User Manual

FUNMAP

Model Tsunamis
Create Maps

The apps and user manuals were created by **Raquel P. Felix,** with parts of the code adapted from the original codes from FUNWAVE-TVD GitHub repository (https://github.com/fengyanshi). The codes created were further refined using AI tools to improve organisation and efficiency. Both the apps and manuals underwent testing and were reviewed and edited by **Elaine Tan Hui Zhi, Masashi Watanabe, Andrea Verolino, Puah Jun Yu, and Adam Switzer.**

Please cite this paper when you use the applications:
Felix, R., Tan, E. H. Z., Watanabe, M., Verolino, A., Puah, J. Y., & Switzer, A. D. (2025).
Funwave-based open-access mapping applications (FUNMAP) applied to Tsunami modelling from the Manila Trench to Manila Bay, Philippines. Geoscience Letters.
https://doi.org/10.1186/s40562-025-00422-5

Table of Contents

Introduction

There is a growing interest in numerical modelling of tsunamis, generated not only by earthquakes (e.g., the 2011 Tohoku, Japan tsunami) but also by volcanic meteotsunamis (e.g., the 2022 eruption at Hunga Tonga) and landslides. Here, we introduce two standalone user-friendly applications with a graphical user interface (GUI) designed to improve the efficiency of both tsunami model preparation and post-processing stages. The use of these apps eliminates the need to analyse large text files and code, thereby making tsunami modelling more accessible to all users.

We named the apps as **"FUNMAP"** as they are built upon the **FUNWAVE-TVD v.3.6 model**, which is a fully nonlinear Boussinesq wave model. Moreover, the name easily communicates the intended purpose of the apps which are to aid in modelling tsunamis and creating maps. The first app is called the **FUNMAP Input app**. Its function is to assist the user in setting up the input file needed to run the tsunami simulation. In this app, users also have the option to run the simulation on the current computer used to run the app. The second app is the **FUNMAP Output app** which functions to visualise the results from the tsunami simulations.

The FUNWAVE-TVD v.3.6 model is the latest version of the FUNWAVE model. The initial version was developed by Kirby et al. (1998), building on the work of Wei et al. (1995). The TVD version used in FUNMAP apps follows the modifications by Shi et al. (2012). Our apps integrate various tsunami generation methods provided in the version 3.6., including ship-wave generation (Shi et al., 2018), meteotsunami generation (Woodruff et al., 2018), and sediment transport and morphological changes (Tehranirad et al., 2016; Malej et al., 2019). FUNWAVE can model both global and coastal-scale tsunami generation, propagation, and inundation.

Installation and Setup

Both apps are developed using the App Designer within the MATLAB programming language. However, users do not need to have the MATLAB software and license to run them. The apps only need MATLAB Runtime (non-license), which is a standalone set of shared libraries enabling the execution of MATLAB-related components. The required MATLAB Runtime is included in the installation package.

For installation, users simply need to download the zip file from the GitHub link provided, unzip it, and double-click on the installation files. The installation package contains the installers for Input and Output FUNMAP apps, along with the needed MATLAB Runtime. Users only need to select the directory where the executable file will be saved, and then the applications are ready to run. Restart the computer after installation to ensure it runs efficiently.

a. Version Information

The user manual refers to the first version of the applications. We released the first public version of the application in 2025. The apps and the user manual are uploaded in GitHub. Further revisions or updates will be posted in https://github.com/raquelpfelix.

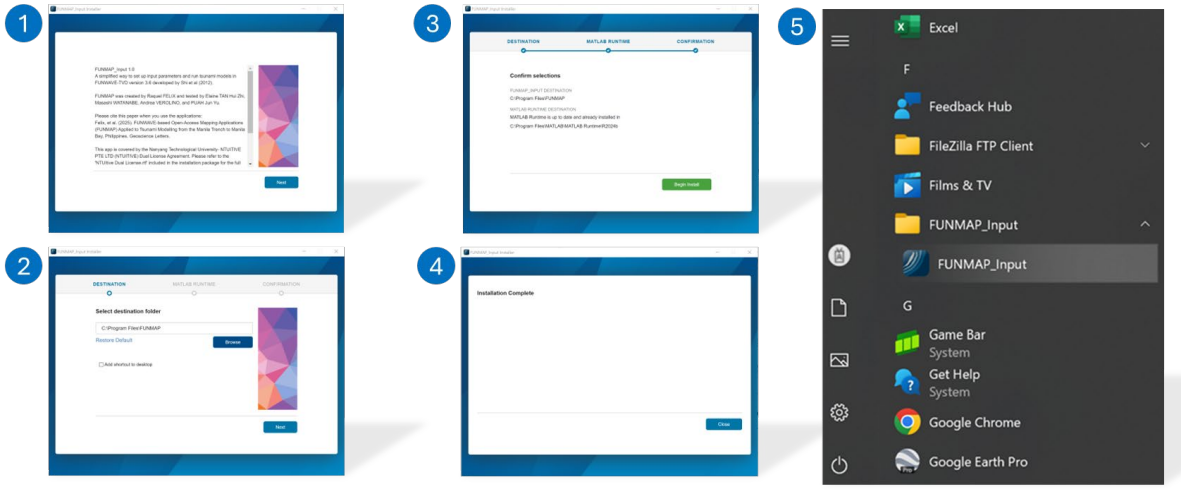
b. Disclaimer

c. Acknowledgments

We thank Shi et al. (2012), the creators and developers of FUNWAVE-TVD version 3.6, for making its source code, examples, and detailed instructions publicly available. All the files used for studying FUNWAVE-TVD and creating the FUNMAP apps can be found at https://fengyanshi.github.io/build/html/index.html. The FUNWAVE-TVD Google community is also a reliable source for resolving issues and can be accessed at: https://groups.google.com/g/funwave-tvd.

Install FUNMAP apps

Option 1: Install as a **standalone app**, which does not require a MATLAB license



Installation Issues on macOS

If you encounter issues during installation, follow these steps based on your Mac's processor type:

For Apple Silicon (M1, M2, M3 chips):

You need to install Rosetta to run Intel-based applications. Follow this guide for instructions:
https://www.howtogeek.com/702982/how-to-launch-the-intel-version-of-a-universal-mac-app-on-apple-silicon/

For Intel-based Macs:

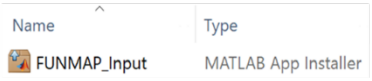
You may need to update file permissions before running the app. Open **Terminal** and enter the following commands:

chmod -R 755 /Downloads/FUNMAP_Input_MacOS.app
sudo xattr -cr /Downloads/FUNMAP_Input_MacOS.app

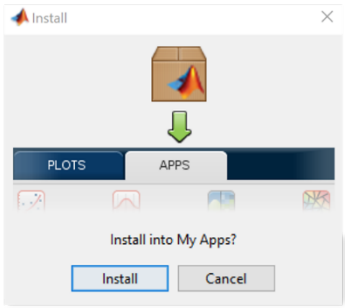
Then, try running the installer again.

Install FUNMAP apps
Option 2: Install as an **in-app feature within MATLAB**

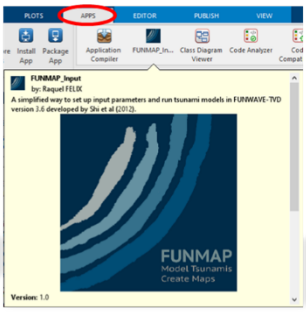
1



2



3

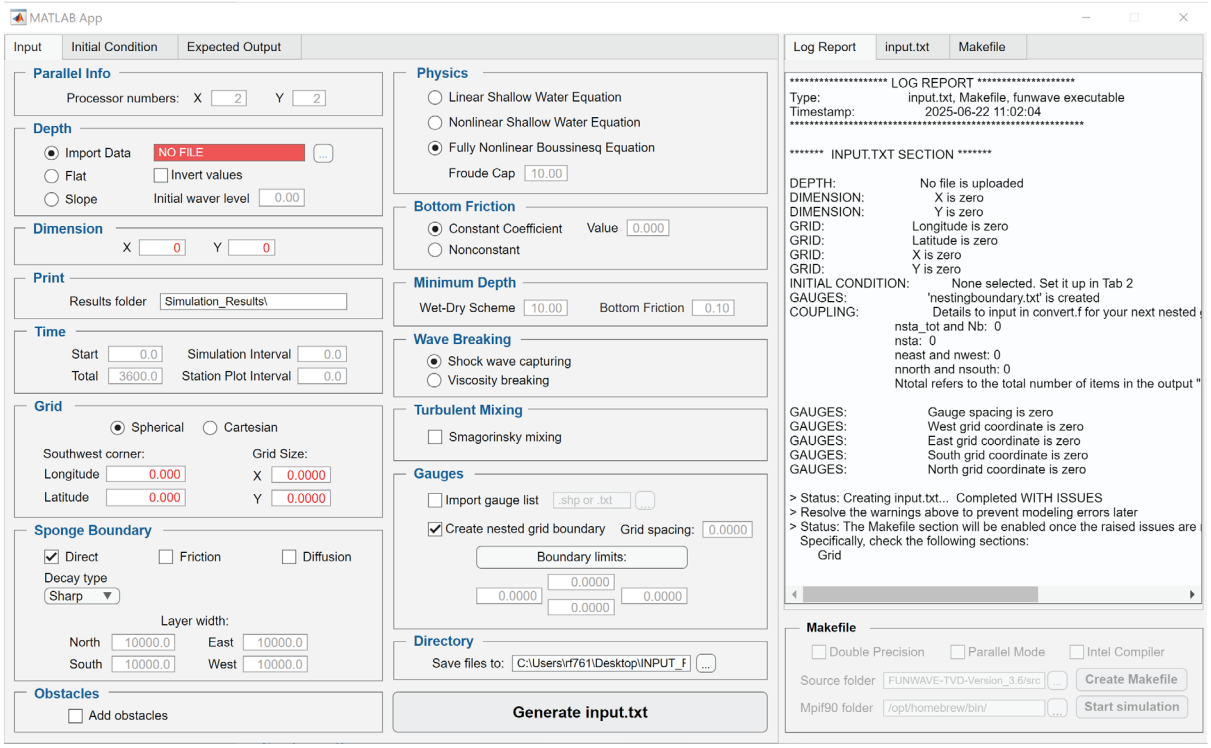


FUNMAP Input App

FUNMAP Input App assists users in preparing input files, generating the executable FUNWAVE, and starting tsunami simulations. The first two panels are used for setting up input parameters, while the third panel previews the files generated by the app.

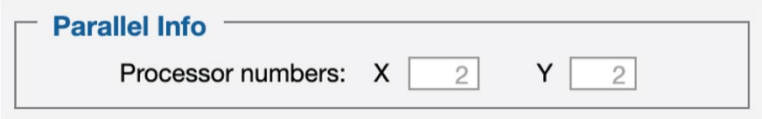
TAB 1: INPUT

This tab allows modification of most input parameters in the 'input.txt' file. It is organized into sections that follow the format of the default input.txt provided by FUNWAVE-TVD. The template and the parameter definitions can be found in https://fengyanshi.github.io/build/html/definition.html.



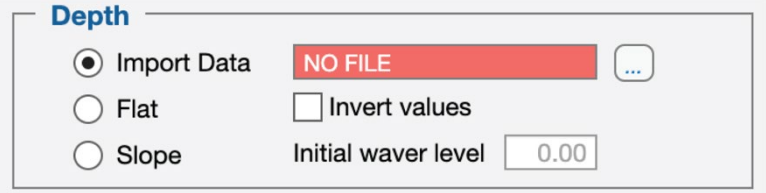
A. Parallel Info

Modify the number of computer processors to use in X and Y directions.

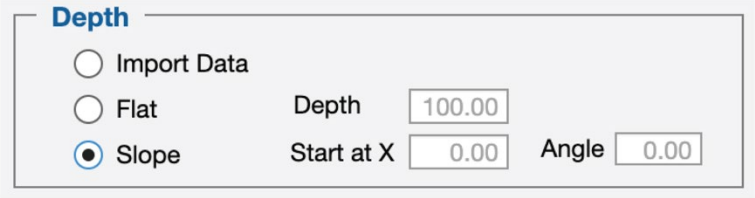


B. Depth

Select the bathymetry configuration. The settings for each option automatically appear when it is selected.



Import Data	Import a file for bathymetry input
Load data	Click the button with three dots. A pop-up window will appear to find the file to import. Accepted formats: .txt and .tif Grid dimension details are automatically extracted from the file. For .tif file, the Southwest Corner coordinates and the grid sizes in the Grid section are also filled. If the uploaded file has a .tif extension, it will be automatically converted to a .txt file. The file will be copied to the 'INPUT_FILES/Input_Data/' folder.
Invert values	To invert positive values to negative and vice versa. The model requires that the water depth values are positive, and topography values are negative. When selected, the modified bathymetry is saved in the 'Input_Data' folder.
Initial water level	Adjust the surface water level of the input bathymetry. This is to incorporate surge, tidal changes, or sea level rise effects.



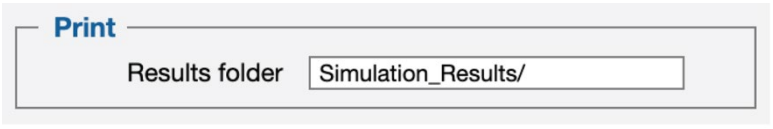
Flat	Use a bathymetry with flat bottom.
Slope	Use a plane beach set up with flat bottom, and a sloping beach part.
Depth	Specify the water depth in meters.
Start at X	The distance along the X-axis (in meters) where the slope begins, gradually transitioning the water depth from flat to sloping towards the beach.
Angle	Adjust the steepness of the slope (in degrees).

C. Dimension

Matrix dimension of the study area in X and Y directions. These are automaticaly filled when a bathymetry file is imported in the 'Depth > Import Data' section.

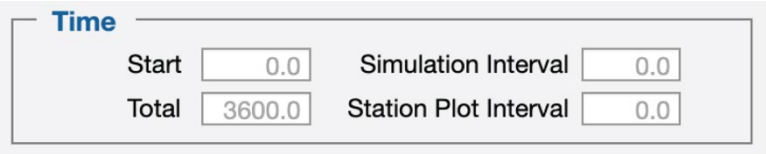
D. Print

The folder name where the tsunami simulation results will be saved. Remember to add the "/"at the end.



E. Time

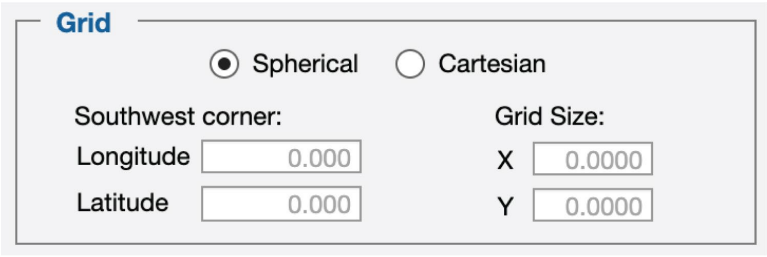
Set up the time parameters for the tsunami simulations. Time unit is in seconds.



Start	Initial time to start the simulation.
Total	Set the total simulation time.
Simulation Interval	Set the time interval between output generation. For example, entering '30' will save hmax and eta files at 30-second intervals. This value is also used as the interval for the screen update that will be displayed in the log report tab of the Preview Panel.
Station Plot Intervals	Set the time interval for recording outputs at virtual gauge stations.

F. Grid

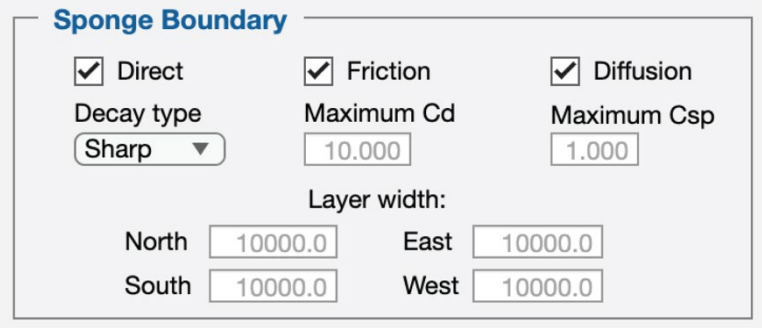
The unit for the Spherical option is in degrees whilst for the Cartesian option it is in meters.



Spherical	Use Spherical grid in the numerical domain.
Southwest Corner	Define the geographical longitude and latitude of the numerical domain.
Cartesian	Use Cartesian grid in the numerical domain.
Grid Size	Specify the size of the grid in X and Y directions.

G. Sponge Boundary

This is used to minimize the wave reflections off the boundaries of the numerical domain.



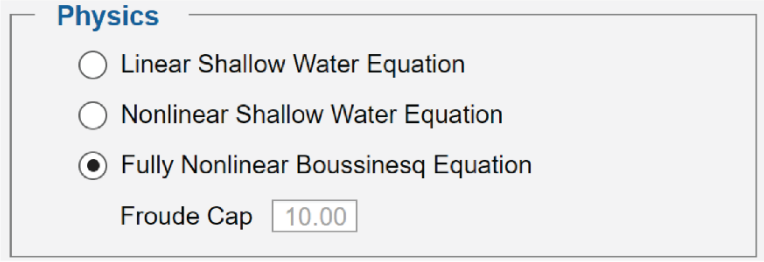
Direct	Activate Larsen and Dancy type of sponge layer (L-D type, 1983).
Decay type *Very mild, Mild, Sharp*	Select the decay type. This will adjust the combination of values of the R_Sponge (decay rate) and A_sponge (maximum decay rate) in the input.txt
Friction	Activate friction type of sponge layer.
Maximum Cd	Set the maximum friction coefficient.
Diffusion	Activate diffusion type of sponge layer.
Maximum Csp	Set the maximum diffusion coefficient.
Layer width	Adjust the thickness of the sponge layer relative to its distance from the north, east, west, and south boundaries. Unit: meters

H. Obstacles

Check the box to include an obstacle structure in the tsunami simulations. The file can be imported by clicking the three-dot button. Note that the matrix values in the file should be a combination of 0's and 1's where they represent the obstacles and water areas, respectively. The format and matrix dimensions of the obstacle file must match those of the input bathymetry file. It will be copied to the 'INPUT_FILES/Input_Data/' folder.

I. Physics

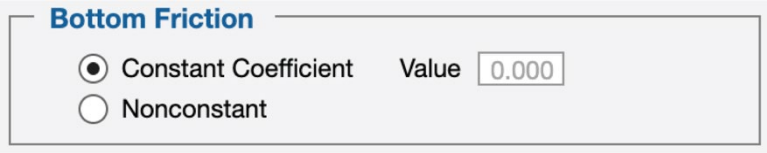
Select the equation to use in the numerical modelling.



Linear SWE	Uses linear shallow water equations. Ideal for large-scale modelling where the tsunami wavelength is longer than the water depth.
Nonlinear SWE	Uses nonlinear shallow water equations. Does not include dispersive terms.
Fully Nonlinear Boussinesq Equation	Uses fully nonlinear Boussinesq equations. Incluldes dispersive terms.
Froude Cap	Upper limit for the Froude number in velocity calculations to optimize efficiency.

J. Bottom Friction

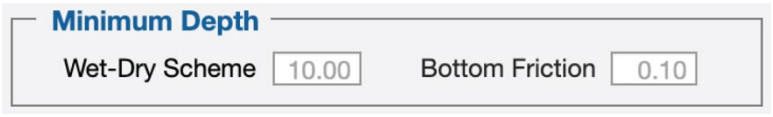
This is to activate the bottom friction feature.



Constant Coefficient	Uses the same friction coefficient in the entire numerical domain.
Value	Input the value to use. Set the value to zero to disable bottom friction.
Non-constant	Uses non-uniform values in the entire domain.
File	Import the file by pressing the three-dot button. Its X and Y dimensions must match the values provided in the 'Dimension' section. The file will be copied to the INPUT_FILES/Input_Data/' folder.

K. Minimum Depth

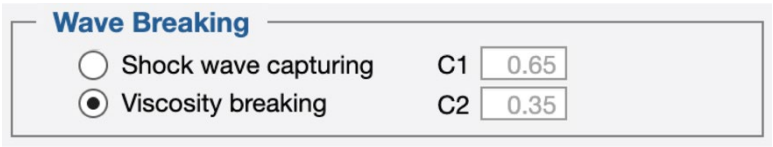
Set the minimum depth boundary limit.



Wet-Dry Scheme	Specify the depth at which the boundary between wet and dry areas in the simulation is established. The FUNWAVE suggests the following values: For lab scale: 0.001 For field scale: 0.01
Bottom Friction	Specify the depth limit for calculations that include bottom friction. The FUNWAVE suggests the following values: For lab scale: 0.01 For field scale: 0.1

L. Breaking

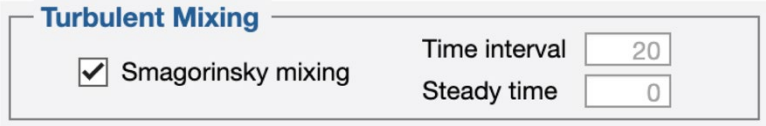
This section allows you to select the type of breaking index to use.



Shock wave capturing	Uses a shock wave capturing scheme.
Viscosity breaking	Uses viscous breaking.
C1, C2	Default values are based on Choi et al. (2018).

M. Mixing

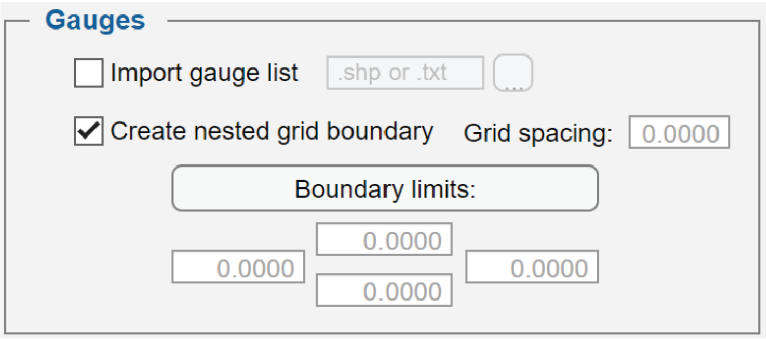
This is to enable the Smagorinsky-like turbulent mixing algorithm. The units for these options are in seconds.

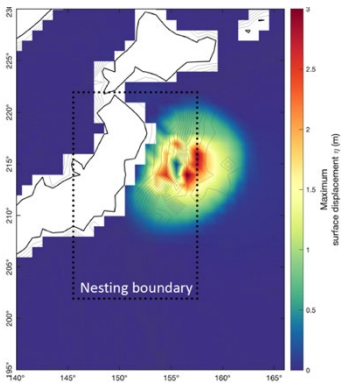


Time interval	Set the time interval for calculating values.
Steady Time	Set the initial time for calculating mean values.

N. Gauges

Set up points to use as virtual gauge stations or as the boundary of the sub-grid layer.



Import gauge list	Check the box to add virtual gauges.
Load File	Load .shp or .txt files containing coordinates. Text file format: two columns, first for latitude and longitude values, no headers. Example: *31.502 138.456* *31.65 135.003* *34.215 140.626* If the uploaded file is in shapefile format, it will be converted to a text file. The file will be copied to the 'INPUT_FILES/Input_Data/' folder.
Create nested grid boundary	Check the box to create points defining the boundary of the nested sub-grid layer. The file will be saved as 'nestingboundary.txt.' More information about setting up the coupling.txt for your nested grid simulation is on the next page. 
Grid spacing	Set the spacing between the points.
Boundary limits	Define the boundary limits of the nested sub-grid layer. Optional: Press the button to automatically extract coordinate and grid spacing information from a .tif file.

A short note on how to prepare your nested-grid simulation

The current version of the app focused on preparing a simulation for one grid layer or the parent/main grid layer. If you're planning to the child/nested-grid layer here are the things that you need to do manually:

1. **Update convert.f**
   - Copy the convert.f' file from `FUNWAVE-TVD-Version_3.6/benchmarks/sph_nesting`
   - Update the values of specific variables. The app provides a note on which sections to edit. Below is an example of the note included in the log report:

```
COUPLING:     Details to input in convert.f for your next nested grid run:
      nsta_tot and Nb:        4500
      neast and nwest:        945
      nnorth and nsouth:      1259

Ntotal refers to the total number of rows contained in each output 'sta_'
file generated after running your model

Update 'fdir' to the directory path where your 'sta_' files and convert.f
file are located

The 'convert.f' file is located in benchmarks/sph_nesting/make_nest_file
```

2. **Copy the convert.f file where your 'sta_' files generated from your previous run are located**

3. **Compile and run convert.f**

   3.1. **On a Local Computer (Using gfortran)**

   ```
   gfortran -o convert convert.f
   ```

   - If a convert.exe is created, type this in the terminal:

   ```
   ./convert
   ```

   - This will generate the coupling.txt file.
   - Copy coupling.txt into the **INPUT_FILES/Input_Data** folder where the other files compiled by the app are located.

   3.2. **On HPC (High-Performance Computing)**

   You can try and add the following lines to your job submission script:

   ```
   #Run job
   gfortran -mcmodel=large convert.f
   ./result.out
   ```

   OR

   ```
   ifort -mcmodel=large convert.f
   ./result.out
   ```

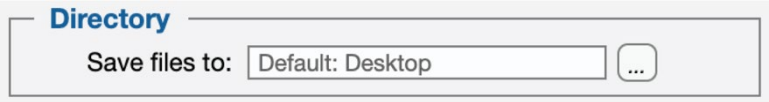
4. **Update input.txt file generated by the app. Copy the texts below.**

```
  ! --------------- INITIAL UVZ -------------------------
--
INI_UVZ = F
  ! ----------------- COUPLING -------------------------
  ! if do coupling, have to set -DCOUPLING in Makefile
COUPLING_FILE = Input_Data/coupling.txt
```

This assumes that the coupling.txt has been moved to the same location (Input_Data folder) as the other files compiled by this app.

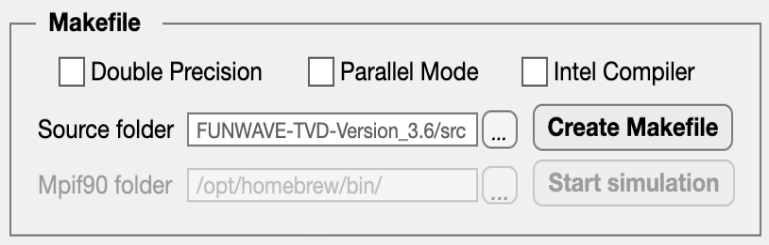
O. Directory

Set the directory path where the 'INPUT_FILES' folder and its subfolders (Input_Data, Log_Report) will be saved. Locate a directory by pressing the three-dot button. If left blank, it will automatically be set to the Desktop location.



P. Makefile

Modify this section to generate the Makefile and compile the executable FUNWAVE. Whilst most modules in the Makefile are automatically adjusted based on the inputs from 'Input' and 'Initial Condition', additional options are provided here to configure the use of double precision, parallel mode, or the Intel compiler. This section is only activated when there are no issues associated with Makefile is encountered when generating the input.txt file. Any issues that need to be addressed first is shown in the log report.

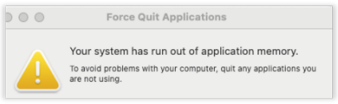


Double Precision	Use double precision. Unchecked box – single precision
Parallel Mode	Use parallel mode. Unchecked box – serial mode
Intel Compiler	Use intel compiler
Source folder	Specify the folder directory containing all the .o, .F, and .f90 files. For example, in version 3.6, the directory is 'FUNWAVE-TVD-Version_3.6/src/'.
Create Makefile	Press the button to create the Makefile. This file is generated in the directory specified in 'Source folder'. At the same time, a copy is saved in the 'INPUT_FILES' for easy access and modification if the user plans to compile the code and create

	the funwave.exe on another computer or a High-Performance Computing cluster. When the Makefile is successfully created, the 'Mpif90 folder' option and the 'Start simulation' button are enabled.
Mpif90 folder	Specify the directory containing the MPI F90 Compiler. The default paths are as follows: • For Mac OS: /opt/homebrew/bin/ • For Windows: C:\Program Files (x86)\MPICH2\bin\ To find the path on your computer, type the following command in the terminal: • On Mac OS: *which mpif90* • On Windows: *where mpif90* This is only necessary if the user intends to create the funwave.exe on the computer where they are using the app.
Start simulation	Press the button to generate funwave.exe and start the tsunami simulation. As the simulation progresses, updates will be shown in both the preview panel and 'INPUT_FILES/LOG.txt.' The funwave.exe generated from the 'Source folder' directory is copied to 'INPUT_FILES', where the simulation is executed, and results are saved. To stop the simulation prematurely, the following methods can be used: • **Mac OS**: Press Ctrl + C in the terminal where the simulation is running. • **Windows**: Press Ctrl + C in the Command Prompt or PowerShell window where the simulation is running.

Run the model using PC: **In-app feature within MATLAB**

Issue: MATLAB uses a lot of the computer memory.



Workaround:

1. Quit MATLAB
2. Open 'Terminal' and go to the directory where the 'funwave' is located and start the simulation.
   Example:
   cd /Users/felixraquel/Desktop/INPUT_FILES/
   mpirun -np 4 ./funwave

Ensure that the px*px value in the terminal matches the values in input.txt

```
! ------------------PARALLEL INFO-----------------------------
!    PX,PY - processor numbers in X and Y
!    NOTE: make sure consistency with mpirun -np n (px*py)
PX = 2
PY = 2
```

Trouble Shooting: Start Simulation

There will be an issue if the MPI F90 Compiler is not found by the app.

For Windows OS user, a workaround that the user can try is to firstly install a Windows Subsystem for Linux (WSL).

1. Open the 'Command Prompt' and type:
   ```
   wsl –install
   ```

2. Restart the computer.

3. Open the 'Command Prompt' and type:
   ```
   bash
   cd Desktop/FUNWAVE-TVD-Version_3.6/src/
   make clean
   make
   ```

4. When the compilation is successful, open 'File Explorer.'

5. Go to the folder where the compiled file is located.
   Example: C:\Users\rf761\Desktop\FUNWAVE-TVD-Version_3.6\src

6. In the folder, find the file named *funwave*

7. Copy the *funwave* file.

8. Go to the 'INPUT_FILES' folder created by the Input App
   Example: C:\Users\rf761\Desktop\INPUT_FILES

9. Paste the *funwave* file.

10. Open the 'Command Prompt' and type:
    ```
    bash
    cd Desktop/INPUT_FILES/
     mpirun -np 9 ./funwave                    %This will start the simulation
    ```

Note: When using mpirun, make sure the number specified is equal to the product of PX and PY. These values (PX and PY) are defined in the input.txt. See the screenshot in the previous page.

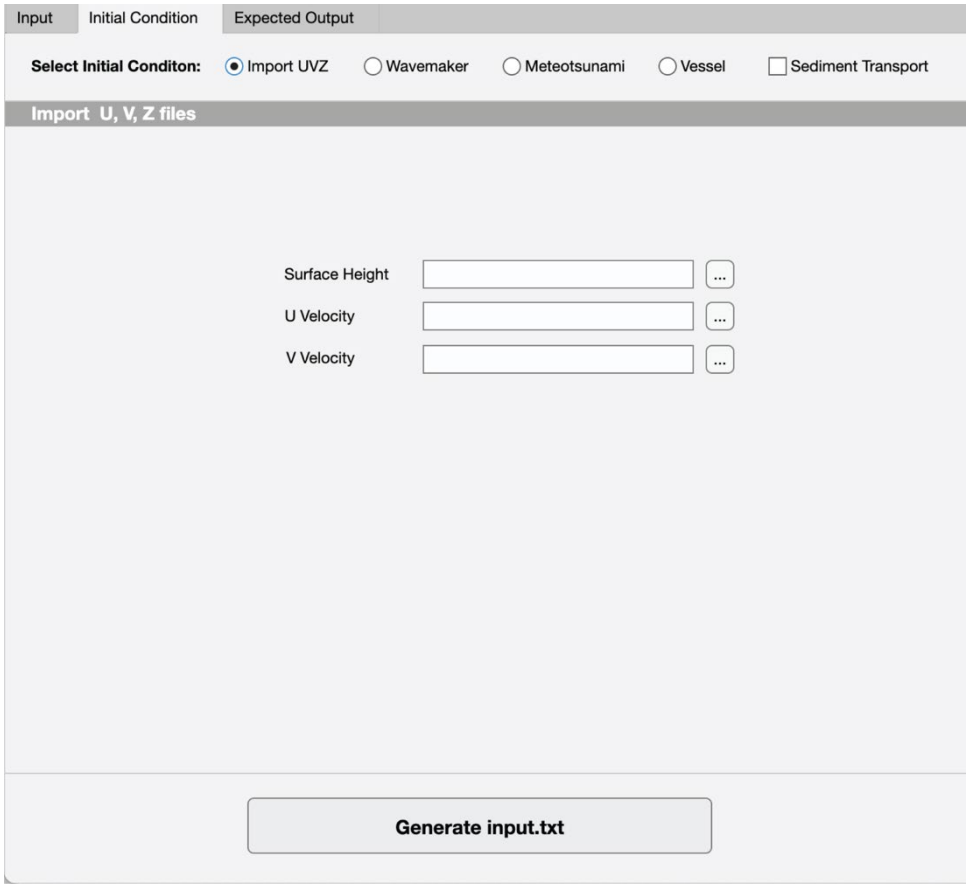
GENERATE THE FILES

By pressing the ‘Generate input.txt’ button, the following files are generated: input.txt, and the Log_Report_InputPrep.txt. The input.txt file is saved in the main folder (INPUT_FILES) whilst the Log report is saved in the ‘Log_Report’ subfolder. All issues encountered by the app are listed in the log report.

The preview of the input.txt and log report are displayed in the third panel of the app. Even when the inputs in the ‘Initial Condition’ and ‘Expected Output’ tabs are not yet filled in, the function of this button will still work and generate the files in order for the users to cross-check if values they input are correctly reflected in the input.txt.

TAB 2: INITIAL CONDITION

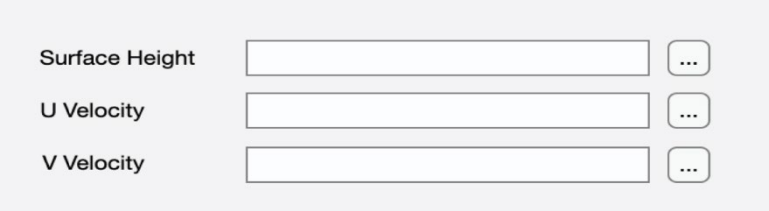
There are four options available for setting up the tsunami source, including another option to add sediment transport in the modelling. The detailed setup for each option is displayed only when clicked.



A. IMPORT UVZ

The tsunami is initiated using U, V, Z files imported in the app. Loading the file can be done by clicking the three-dot button. The file format and matrix dimension of all these files must match the input bathymetry.

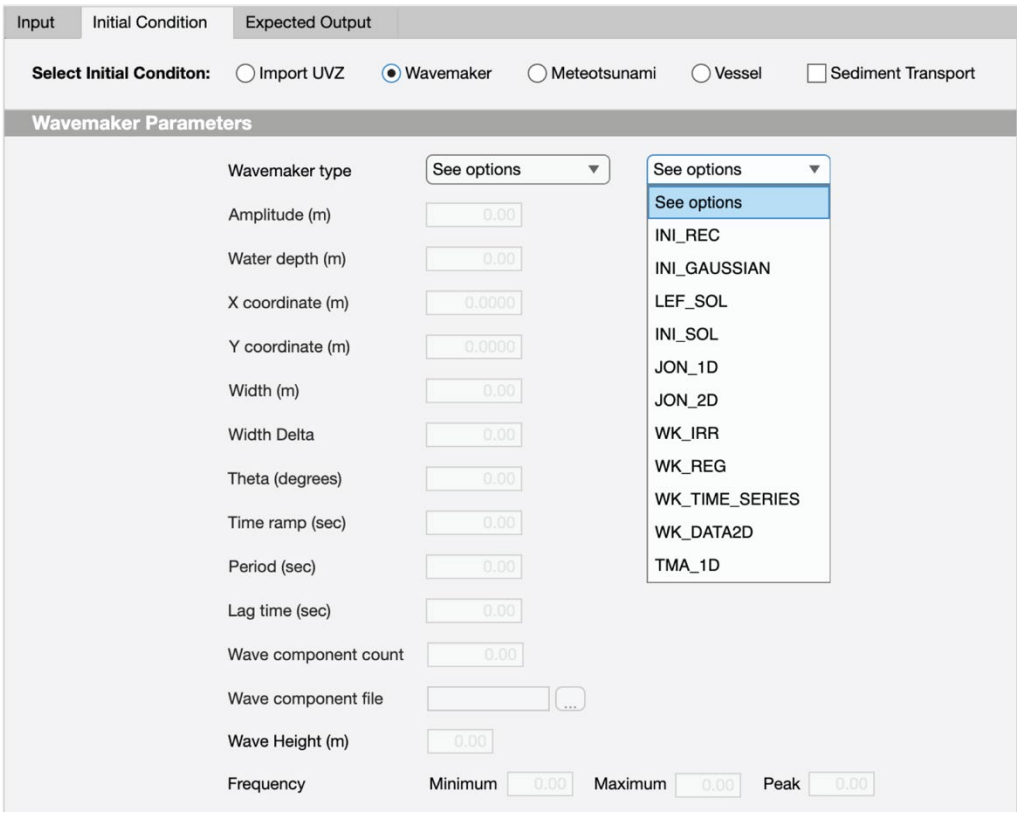
The files should be in .txt format. If you have a .tif file, it can be converted to .txt using Input Tab > Depth Section > Import Data (see page 9).



Surface Height	File containing the sea surface displacement data (Z).
U Velocity	File containing the X-component of the velocity data.
V velocity	File containing the Y-component of the velocity data.

B. WAVEMAKER

Select this option to initiate the tsunami using a wavemaker. The textboxes for the listed parameters are enabled based on the selected option under the 'Wavemaker type' dropdown list.

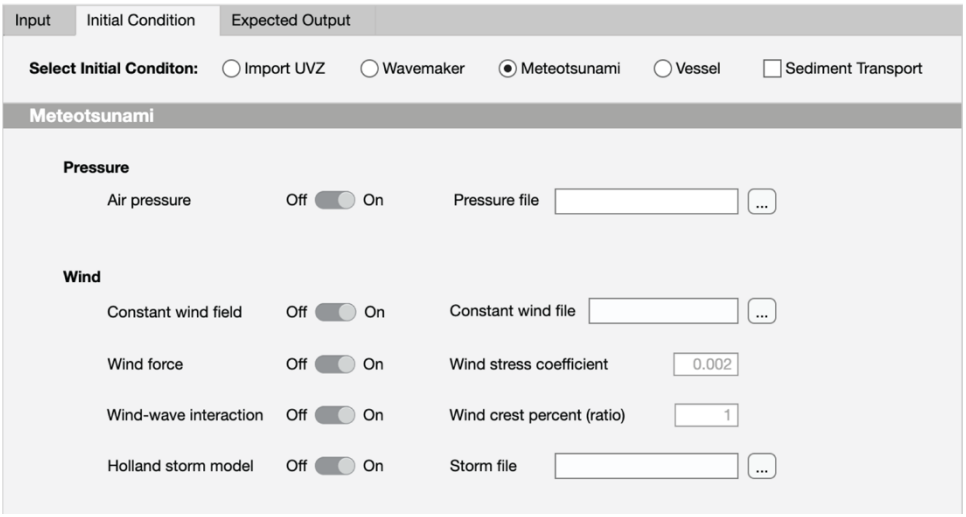


Wavemaker type	
INI_REC	Rectangular hump.
INI_GAUSSIAN	Gaussian hump.
LEF_SOL	Left boundary solitary.
INI_SOL	Solitary wave.
JON_1D	JONSWAP 1D
JON_2D	JONSWAP 2D
WK_IRR	TMA spectrum wavemaker of Wei and Kirby (1999)
WK_REG	Internal wave maker by Wei and Kirby (1999)
WK_TIME_SERIES	Performs fast Fourier Transform (FFT) on a time series to extract each wave component before employing the wave maker by Wei and Kirby (1999).
WK_DATA2D	2D directional spectrum data is provided in the Wave component file.
TMA_1D	TMA 1D

Amplitude (m)	Height of the initial sea surface displacement.
Water depth (m)	Water depth at the location of the wavemaker.
X coordinate (m)	Coordinate in X direction. For INI_REC, it refers to the location of the center of the rectangular hump.
Y coordinate (m)	Coordinate in Y direction. For INI_REC, it refers to the location of the center of the rectangular hump.
Width (m)	For INI_REC or INI_GAUSSIAN, it refers to the width of the rectangular hump. For WK_REG, it is the width in Y direction.
Width Delta	Delta width parameter. Default values: 0.3 ~ 0.6
Theta (degrees)	Direction of regular wave.
Time ramp (sec)	The period during which wave height gradually increases from zero to its maximum.
Period (sec)	Time period of the regular wave.
Lag time (sec)	Lag time associated with the solitary wave generated at the left boundary.
Wave component count	Number of the wave components.
Wave component file	Load the file that contains the wave component. Refer to the FUNWAVE manual for the content format.
Wave Height (m)	Wave height used in Wei and Kirby (1999)
Frequency	Peak, minimum and maximum thresholds of the frequency to be used in Wei and Kirby (1999).

C. METEOTSUNAMI

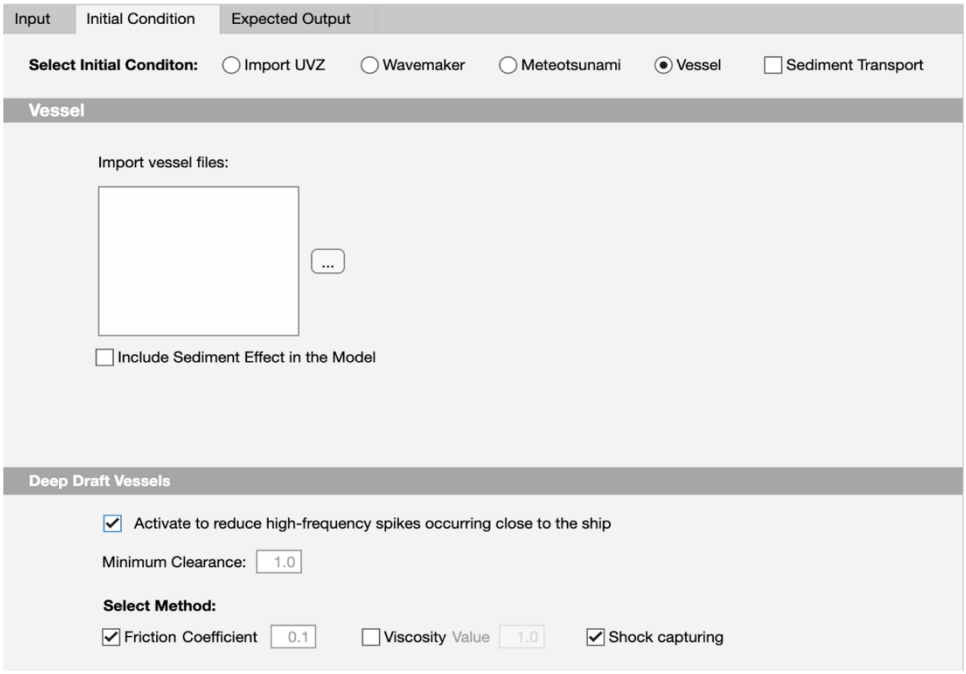
Select this option to initiate the tsunami using atmospheric disturbances.



Air Pressure	Uses air pressure effect.
Constant wind field	Uses a constant wind field. Upload the wind data.
Wind force	Uses wind effect.
Wind stress coefficient	Value used in the quadratic formula.
Wind crest percent (ratio)	The ratio of the forced wave crest height to the maximum surface elevation. For storm surges, the default value is 1.
Wind-wave interaction	Activates wind-wave interaction by Chen et al. (2003).
Holland storm model	Uses the Holland model. Upload the data in the Storm file.

D. VESSEL

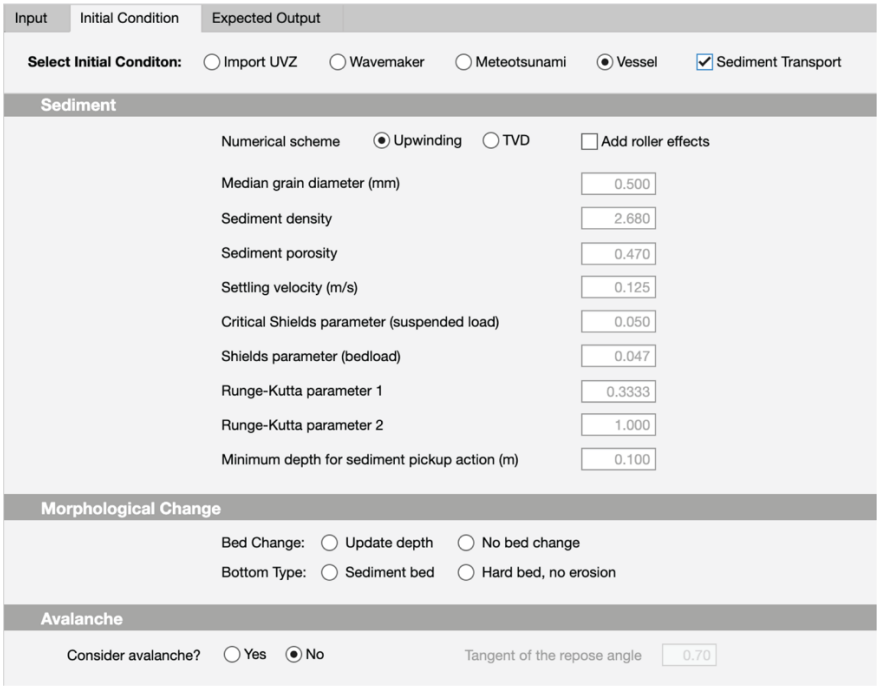
This option uses the ship-wake model in the simulation.



Import vessel file/s	Load the vessel files by clicking the three-dot button. All files must be in the same folder. A preview of the files is shown on the textbox. Refer to the FUNWAVE manual for the template of the vessel file.
Include Sediment Effect in the Model	Check the box to include the sediment effect.
Activate to reduce high-frequency spikes occurring close to the ship	Check the box to address the instability issue that may arise when modelling a large vessel with a draft close to the channel depth. Auto-enables all the options listed below.
Minimum Clearance	Set the minimum clearance
Select Method	Select methods to use to solve the high-frequency spikes.
Friction	Utilizes the dissipation method for damping waves similar to a sponge layer. Suggested values: 0.1 – 1
Viscosity	Similar to the friction method but its wave damping rate is usually lower. Suggested values: 0.1 – 5
Shock capturing	Utilizes the shock-capturing method.

E. SEDIMENT

This provides the initiation setup for sediment transport and morphological change. In the first section, the provided parameters are for non-cohesive sediments.



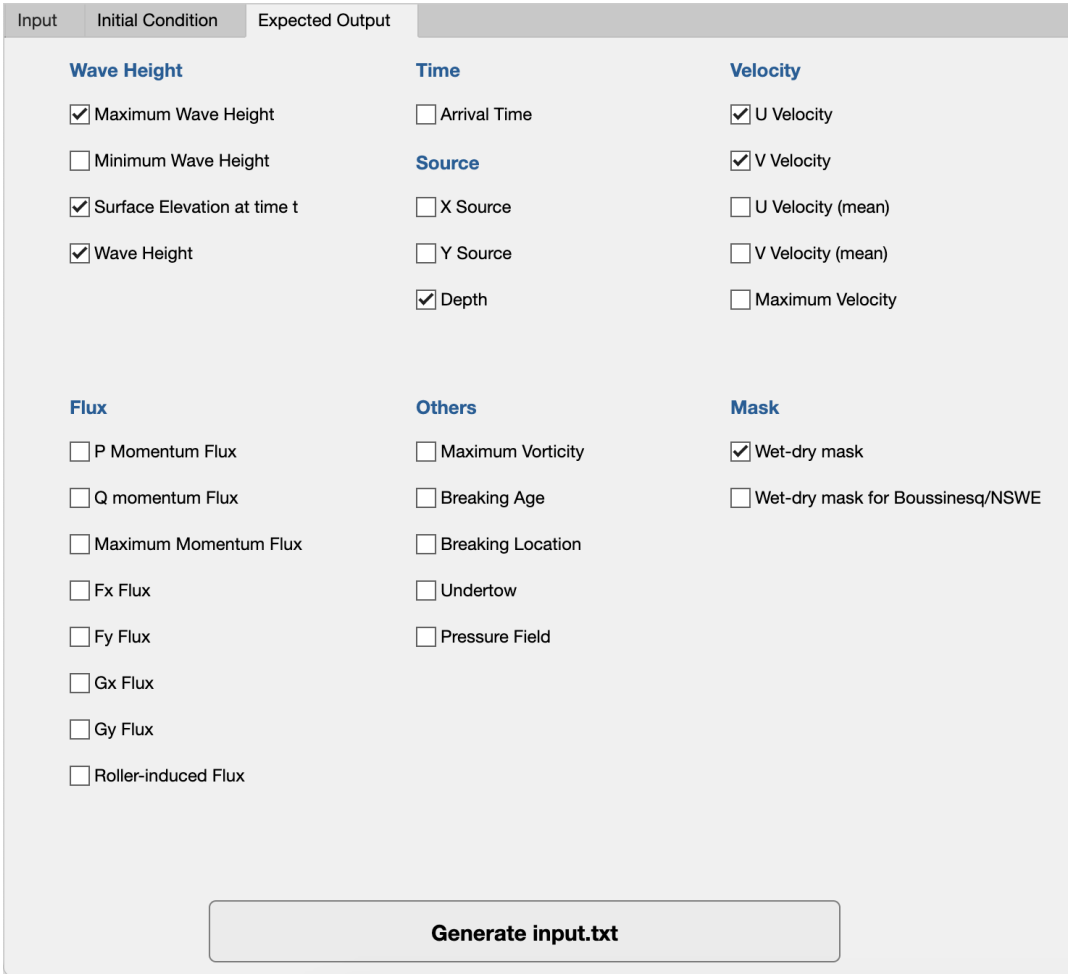
Numerical Scheme	Scheme to use for solving the advection-diffusion equation Upwinding: default scheme TVD: Total Variation Diminishing
Median grain diameter (mm)	D_{50} of the grain diameter
Sediment density	Sediment gravity
Sediment porosity	Porosity of the sediments
Settling velocity (m/s)	Rate at which sediment particles settles.
Critical Shields parameter (suspended load)	Value defining when sediment particles begin to move under the influence of fluid flow.
Shield parameter (bedload)	Default θ_{cr} value is based on Meyer-Peter and Muller (1984).
Runge-Kutta parameter 1	Coefficient values to use in the calculation of Runge-Kutta method.
Runge-Kutta parameter 2	
Minimum depth for sediment pickup action (m)	Set the minimum depth to calculate the sediment pickup action.

Morphological Change	
Bed Change	
Update depth	Bathymetry is updated in the simulation
No bed change	Bathymetry remains constant
Bottom Type	
Sediment bed	Select to allow bed erosion.
Hard bed, no erosion	Select for no erosion. Upload the file containing the distribution of the erodible layer thickness in the 'Hard Bottom File' option. The file format must be the same with the input bathymetry.

Avalanche	
Consider avalanche	Select ‘Yes’ to enable simulation with avalanche consideration. If yes, it enables the option below.
Tangent of the repose angle	The tan Φ repose angle, the steepest angle at which the sediments remain stable.

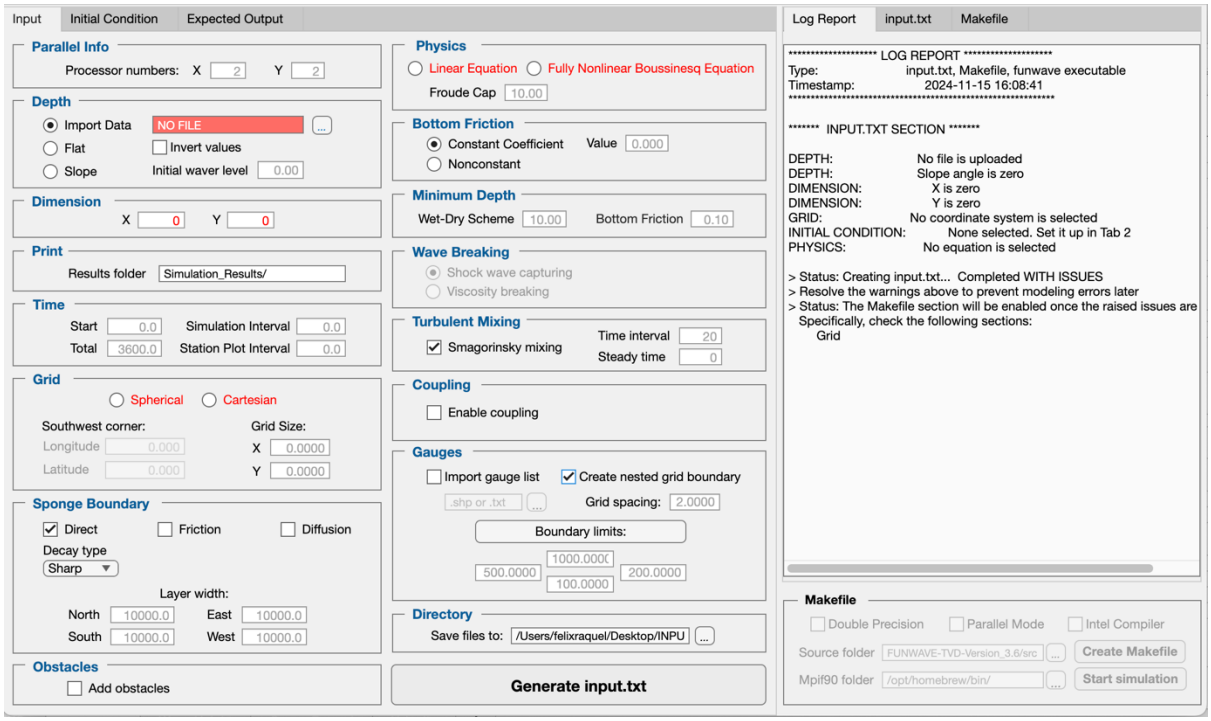
TAB 3: EXPECTED OUTPUT

In this tab, the user can select multiple types of outputs that they want the tsunami simulation to generate. The input.txt file can also be generated in this tab using the button located at the bottom of the panel.



PREVIEW PANEL

The preview panel is in the third column of the app. It features three tabs where the contents of the generated log report, input.txt, and Makefile can be previewed.

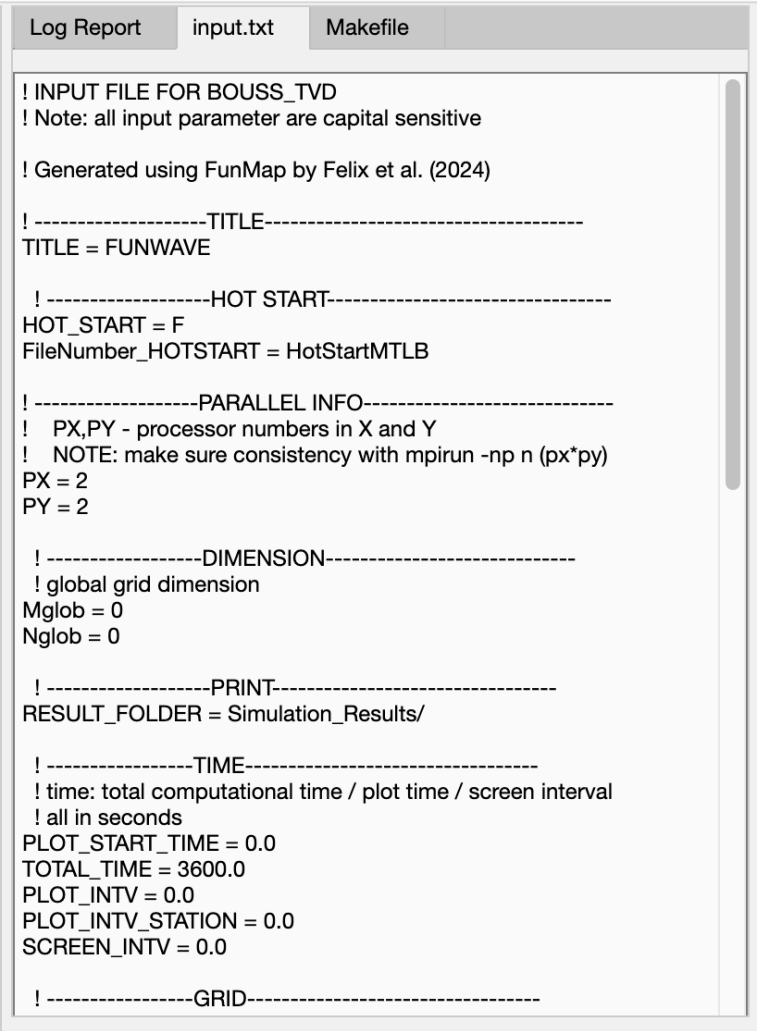


TAB 1: Log Report

This report, saved as ‘Log_Report_InputPrep.txt’, tracks the progress of creating the files: input.txt, Makefile, and funwave.exe. The detected errors or missing files in the input.txt are also highlighted in the app. For instance, if 'DEPTH: No data is selected' is reported, it prompts you to review the 'Depth' section. Sections with errors mentioned in the log report are highlighted in the first and second panels of the app.

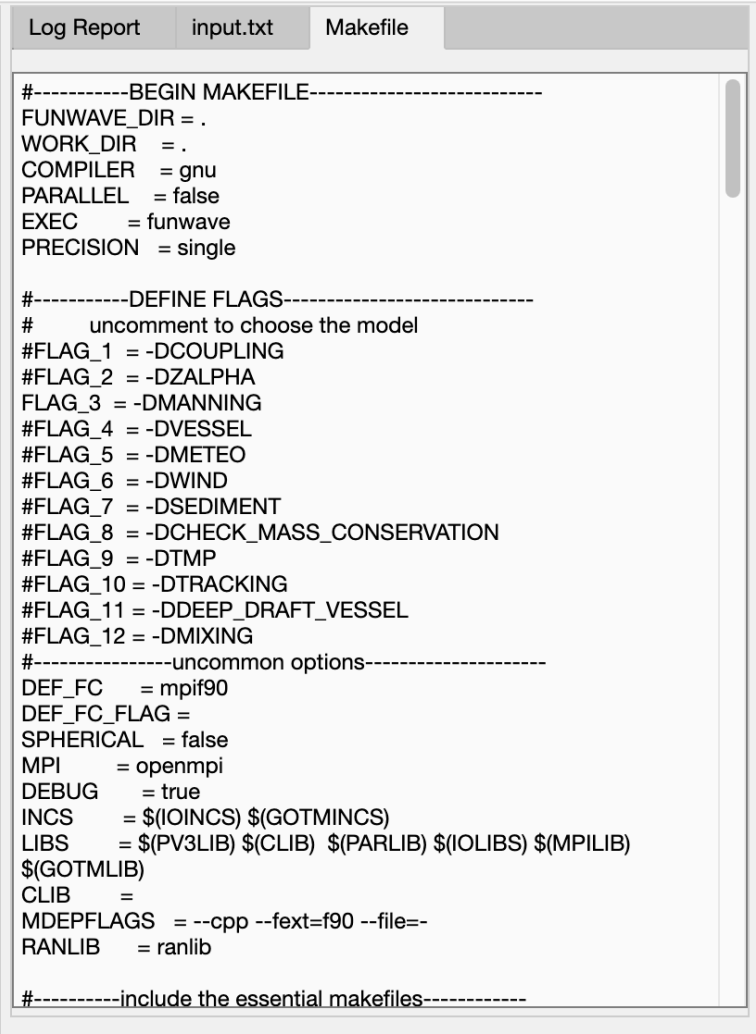
TAB 2: input.txt

This displays the generated input.txt file, allowing the user to review the loaded filenames and entered values.



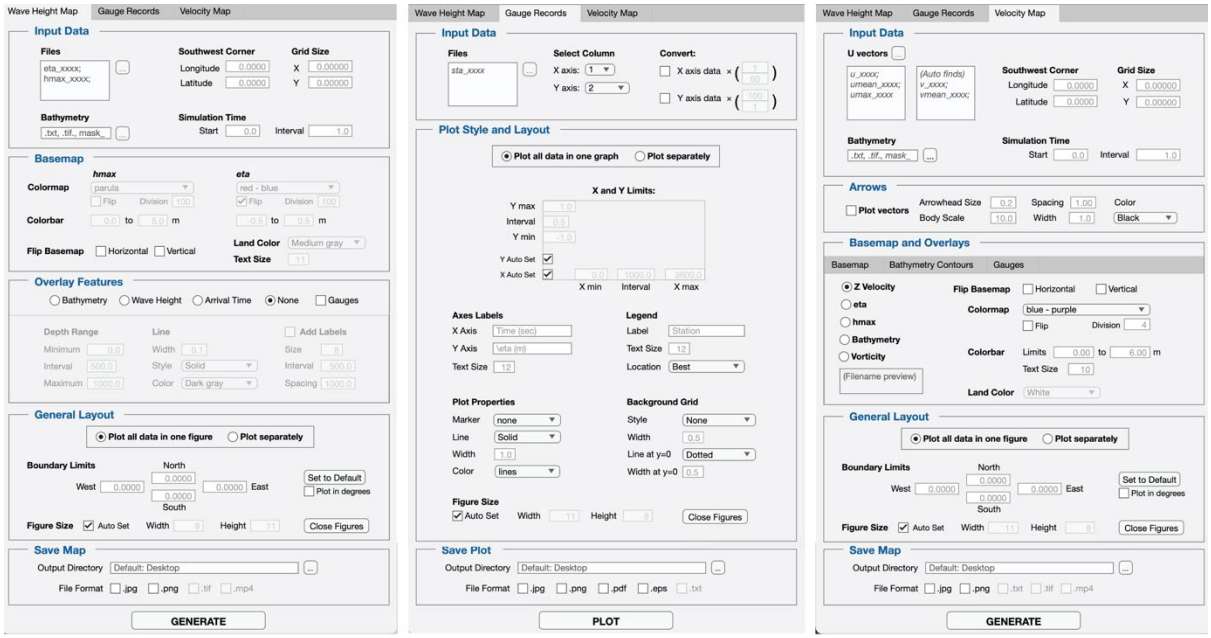
TAB 3: Makefile

This tab displays the generated Makefile. Here, it can be reviewed which FLAGS are activated (i.e., no '#' symbol to the left side). For example, if coupling is enabled, the FLAG_1 must not have the "#" symbol.



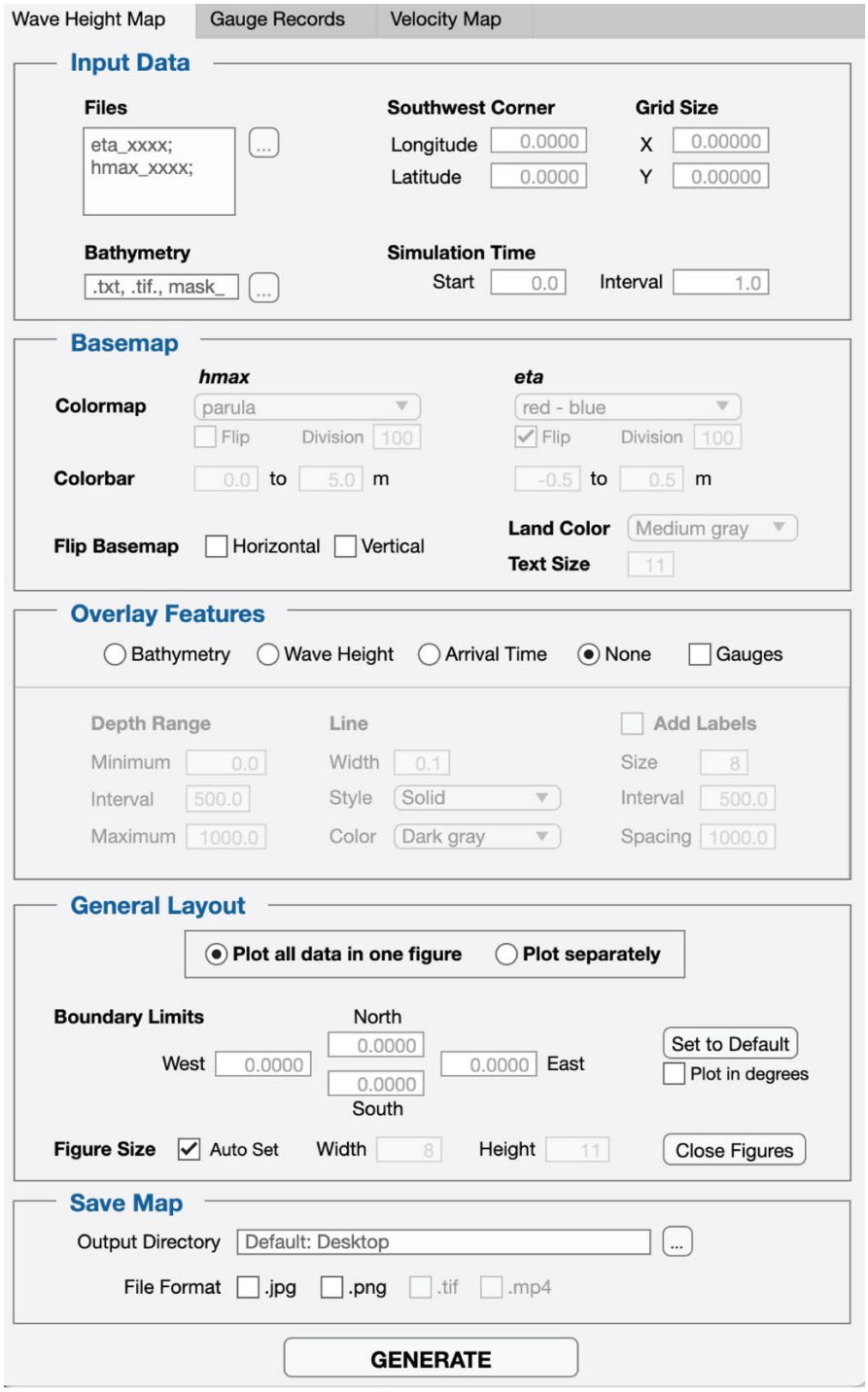
Output FUNMAP App

This application visualizes the tsunami simulation results through maps, animations, and plots. The interface features three main tabs, each independent from the others. The first tab creates wave height maps, the second tab plots gauge records, and the last tab generates velocity maps.

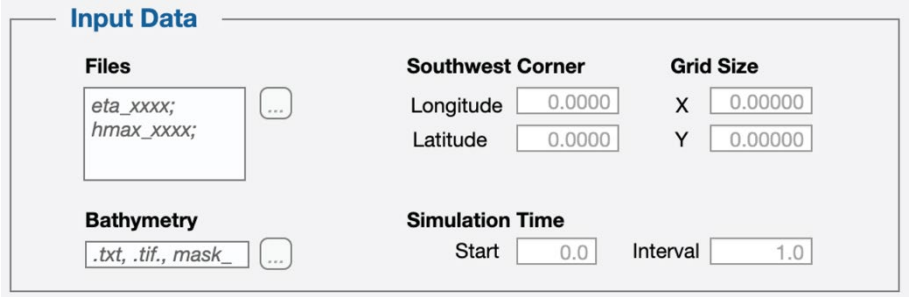


TAB 1: WAVE HEIGHT MAP

In this tab, the 'hmax' and 'eta' files are used to generate wave height maps. The app layout is divided into five sections: importing required files for the basemap, overlaying additional features, adjusting the general layout, and specifying the file type for saving.



A. INPUT DATA



Files	Import the 'hmax' and/or 'eta' files to be used as the basemap by pressing the three-dot button. Importing a combination of 'hmax' and 'eta' files at the same time is possible as long as they are in the same folder directory. The imported files will be listed in the textbox.
Bathymetry	Import the bathymetry file by pressing the three-dot button. The file will set the boundary between land and water bodies, automatically enabling the 'Land Colour' option in the Basemap section. Accepted formats: .txt, .tif Grid size details are automatically extracted from the file. For .tif file, the southwest corner coordinates are also filled.
Southwest Corner	Input the coordinates of the southwest corner used for the tsunami simulation, as specified in input.txt. Unit: degrees, meters
Grid Size	Input the X and Y grid size as specified in the input.txt. Unit: degrees, meters
Simulation Time	Input the simulation timings to be used for figure titles. Unit: seconds. For values >60 seconds, the unit will be automatically converted to minutes or hours. The user can verify whether the conversion is correct by double-checking the LOG.TXT file generated after the simulation. For example, in the line: "PRINTING FILE NO. 1 TIME/TOTAL: 3600.000 / 7200.000" the hmax or eta 00001 entry will be labelled with a title corresponding to 1 hour.
Start	Start time. This is useful when using 'Import UVZ' as the initial tsunami condition, where the imported file contains a time snapshot of nonzero values. This enables readjustment of the figure title.
Interval	Time interval for saving each output file. Input the PLOT_INTV value as specified in the input.txt.

B. BASEMAP

Colourmap	Assign colours to wave heights using built-in and cbrewer2 package options. Visit MATLAB colourmap and cbrewer2 package websites for visuals. Two separate dropdown lists for colourmaps are provided in case both 'hmax' and 'eta' types of files are loaded at the same time.
hmax	It is auto enabled when ‘hmax’ files are imported.
eta	It is auto enabled when ‘eta files’ are imported. It is recommended to use a divergent colour sequence (e.g., red-white-blue) to clearly distinguish between positive and negative waves.
Flip Checkbox	Reverses the colourmap sequence.
Division	Defines the colour transition refinement. A higher value results in smoother transitions. Auto-enabled when a cbrewer2 colourmap is selected.
Bar Limits:	Input both the minimum and maximum values to define the colourbar range to be displayed on the map. For 'eta' files, set symmetrical opposite values for minimum and maximum limits (e.g., -1 to 1 meters) and apply a divergent colourmap. This ensures both positive and negative waves are clearly visualized with equal intensity.
Flip Basemap	Flip the matrix horizontally or vertically.
Land Colour	Choose from eleven colour options, with white as the default. It is auto enabled when a bathymetry file is imported.
Text Size	Sets the text size of the colourbar labels. It also increases the map title text size by 1 unit relative to the input label size.

C. OVERLAY FEATURES

This section provides options to overlay contours generated from bathymetry, wave height, or arrival time, along with virtual gauge stations. The parameters for each option are organized across different tabs.

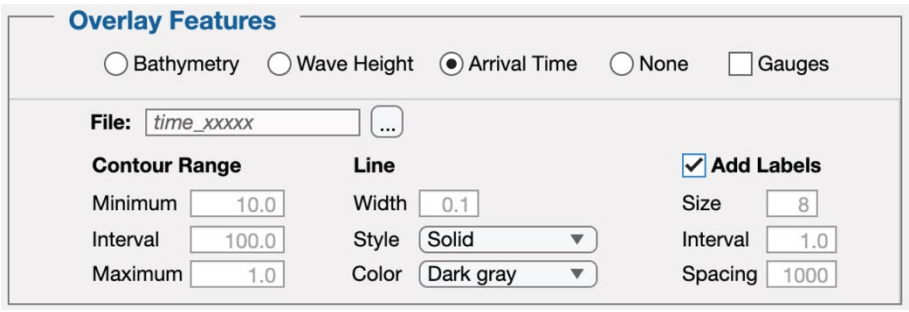
C.1 Bathymetry Contours Tab

Depth Range	Set minimum, interval, and maximum values in meters. Note that water depths are represented as positive values, where higher values indicate deeper waters.
Line	Customize contour line thickness, style and colour.
Add Labels	Add and customize the labels for water depth.
Size	Adjust the font size of the labels.
Interval	Specify label intervals for contour lines. Unit: meters
Spacing	Set the text spacing, with lower values resulting in less crowded text generation Unit: 1 point = 1/72 inch

C.2 Wave Height Tab:

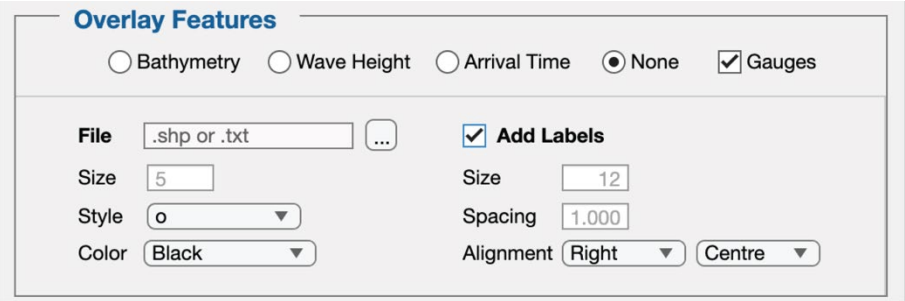
Contour Range	Set minimum, interval, and maximum values in meters.
Line	Customize contour line style, colour, and thickness.
Add Labels	Add and customize the labels.

C.3 Arrival Time Tab:



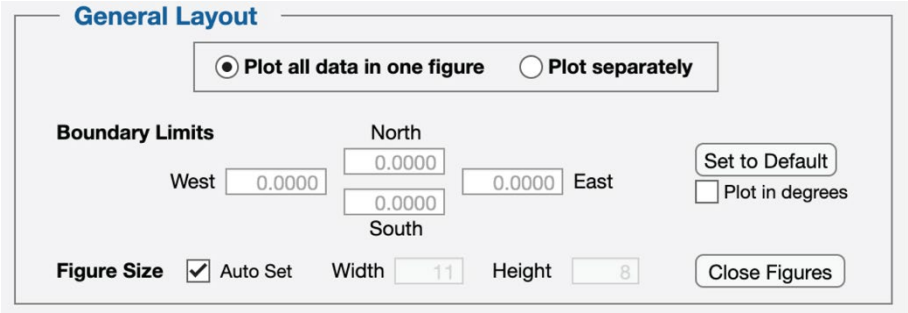
File	Import the 'time' output file for contour generation by clicking the three-dot button.
Contour Range	Set minimum, interval, and maximum values in minutes.
Line	Customize contour line style, colour, and thickness.
Add Labels	Add and customize the labels.

C.5 Gauges Tab



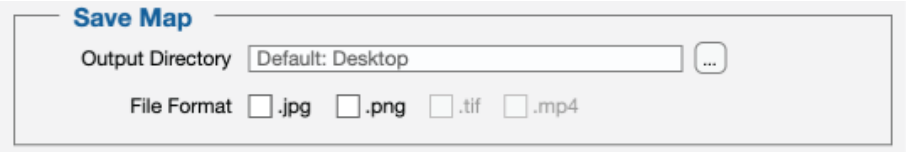
File	Load .shp or .txt files containing the coordinates of the station files. Text file format: Two columns with latitude and longitude values, without headers. Example: *31.502 138.456* *31.65 135.003* *34.215 140.626*
Marker	Adjust the style, size, and colour of gauge markers.
Add Labels	Add and customize the labels.
Size	Adjust the font size of the labels.
Spacing	Set the distance between the marker and the label. It is added to the x-axis on the right side of the marker. Unit: degrees, meters
Alignment	Set the text's horizontal and vertical position relative to the marker.

D. GENERAL LAYOUT



Plot all data in one figure	Show maps as subplots in a single window. Advantage: Easy result comparison. Downside: When too many files are loaded, the maps appear too small to view details clearly.
Plot Separately	Show each map in separate windows. Advantage: Larger, high-resolution maps. Downside: Overlapping figure windows can hinder quick analysis.
Boundary Limit	Adjust the longitude and latitude limits of the map. It is automatically set to match the boundary limits of the imported eta/hmax files.
Set to Default	Reset the boundary limit to the bounds of the eta/hmax files.
Plot in degrees	Convert map tick labels on the X and Y axes to degrees unit. Even with this option checked, input values in the 'Boundary Limit' text boxes should remain in the original numeric values. For example, if the original East limit is 200 units, and you wish to shift it 20 units to the right, input 220 units, not -140°.
Figure Size	Size of the generated figure.
Auto Set	The application determines the optimal size.
Width, Height	Manually adjust the figure size in inches. This option is enabled when 'Auto Set' is unchecked.
Close Figures	Close all currently open figures. Note: Existing figure windows are automatically closed before new ones are generated when clicking the 'Generate' button.

E. SAVE MAP

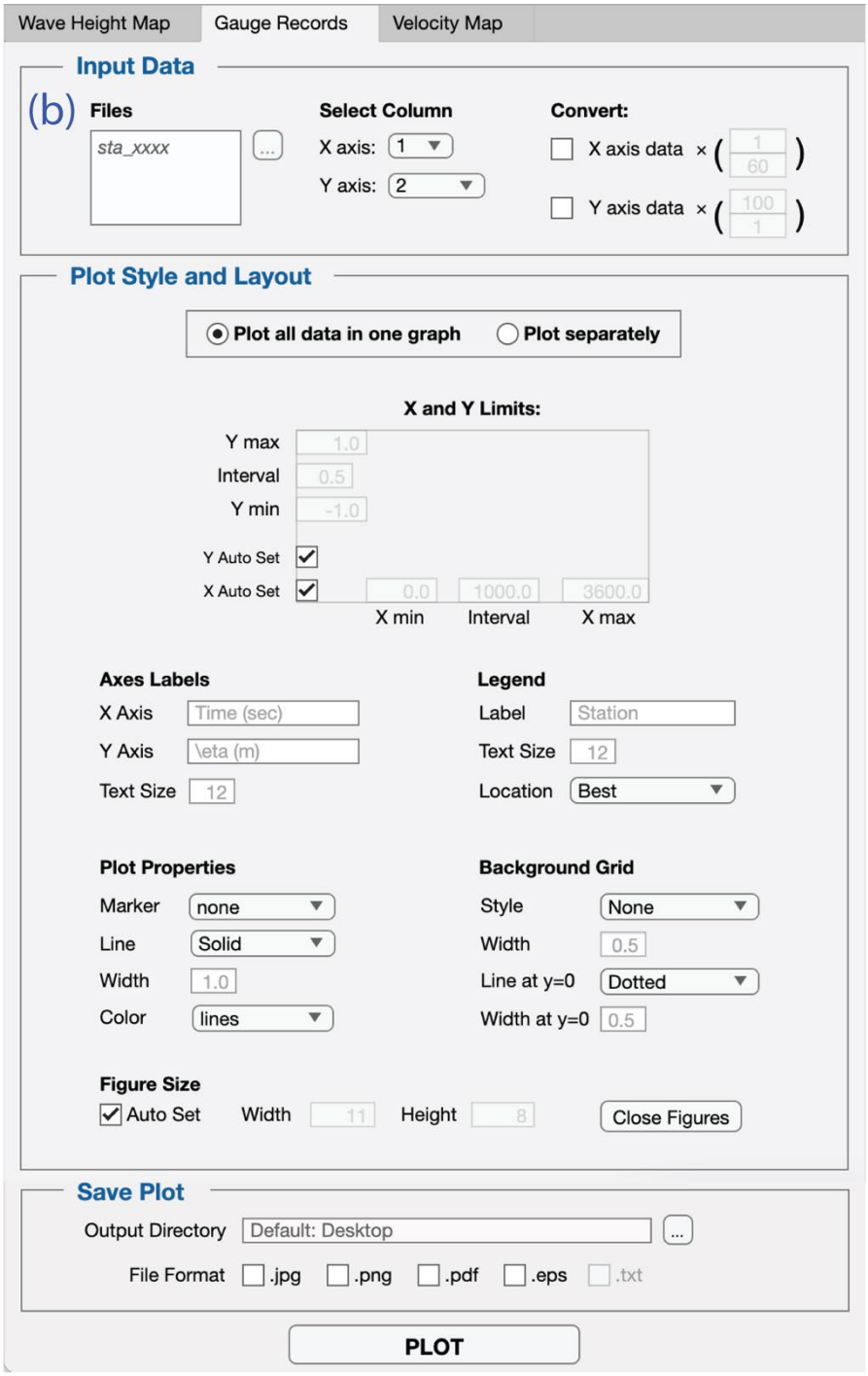


Output Directory	Folder location for saving files. To change the directory, click the button with three dots. A new folder named 'OUTPUT_FOLDER/Figures/' is created at the specified path. Default location is set to Desktop.
Filename	When 'Plot all data in one figure' is selected, the output filename will be "WaveHeight_Output." For the 'Plot separately' option, each output file will have the same name as the corresponding raw file (e.g., eta_00001.png; hmax_0001.tif).
File Format	
.jpg, .png	Image files. Default resolution: 300 DPI.
.tiff	Georeferenced raster file. Use this file format to load the map in GIS applications. It is enabled when 'Plot Separately' is selected.
.mp4	Create an animation. Make it high-resolution by manually setting the width and height of the figure in 'General Layout' section of the app. Frame Rate: Frames or figures displayed per second. The lower the value, the slower the animation. It is enabled when 'Plot Separately' is selected. The output filename will be 'animation_waveheight.'

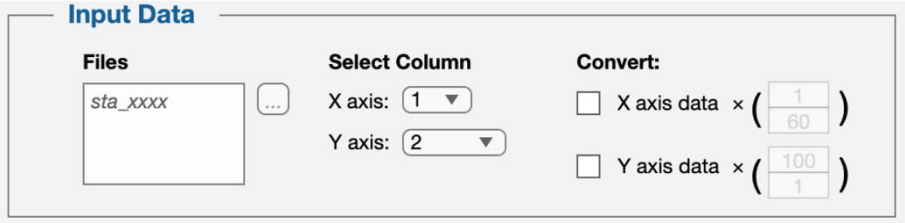
Press the 'Generate' button to create the maps

TAB 2: GAUGE RECORDS

Here, the ‘sta’ files are imported to analyse the data recorded by virtual wave gauges.

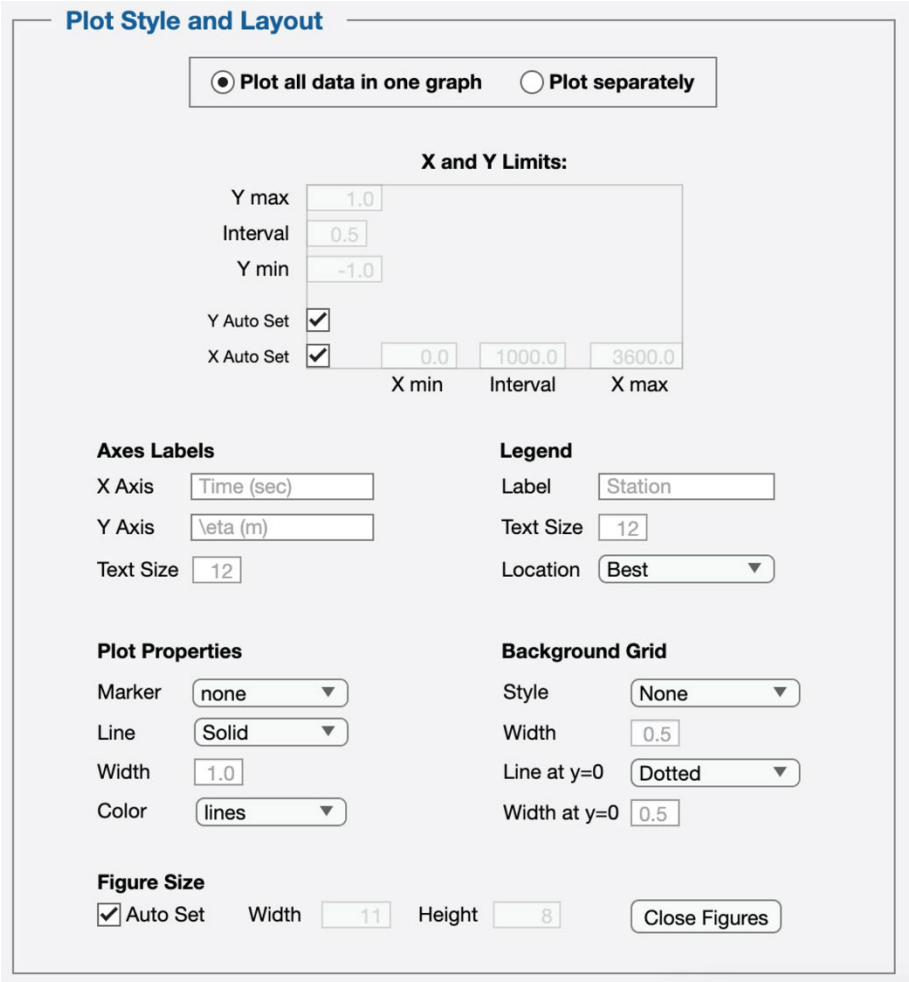


A. INPUT DATA



Files	Click the button with three dots to import files with 'sta' in the filename. Selected filenames will appear in the text box next to the button.
Select Column	Select the columns to use for X and Y axes in the plot. Unit for each column: 1: time (s) 2: eta/hmax (m) 3: u vector (m/s) 4: v vector (m/s) 5: z vector (m/s) as calculated using the values from U and V files.
Convert	Check the boxes to change the units of the X and Y axis values. Input conversion factors as fractions in the corresponding text boxes.

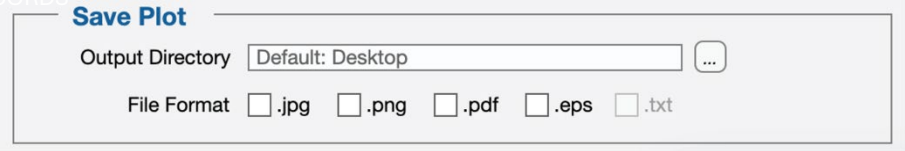
B. PLOT STYLE AND LAYOUT



Plot all data in one graph	One figure is generated showing all data plotted in one graph. It is ideal for comparing and analyzing data.
Plot separately	Display the data from each 'sta' file in separate windows.
X and Y Limits	Modify the X and Y limit of the graph. Default: the intervals and the limits are automatically set based on the minimum and maximum values in the selected data columns. To manually adjust the intervals and the limits, uncheck the X and Y Auto Set tick boxes.
Axes Labels	Modify the labels and font sizes of the X and Y axes. The labels are automatically adjusted based on the data column selected for both axes in the 'Extract Values' part of the Input Data section.

Legend	Adjust the legend of the plotted data.
Label	Text at the beginning of the legend. By default, it is set to 'Station,' which displays sta_0001 and sta_0002 files as 'Station 1' and 'Station 2,' respectively. Leaving it blank will display only the numbers in the legend.
Location	Specifies where to position the legend.
Plot Properties	
Marker	Select the display style for the data points.
Line, Width, Colour	Customize the styles, colours, and thickness.
Flip Checkbox	Reverses the colourmap order. It is auto enabled when ‘Plot all data in one graph’ is selected.
Background Grid	Customize the style and colour of the graph grid.
Figure Size	Size of the generated figure.
Auto Set	The application determines the optimal size.
Width, Height	Manually adjust the figure size in inches. This option is enabled when 'Auto Set' is unchecked.
Close Figures	Close all currently open figure windows. Note: Existing figure windows are automatically closed before new ones are generated when clicking the 'Generate' button.

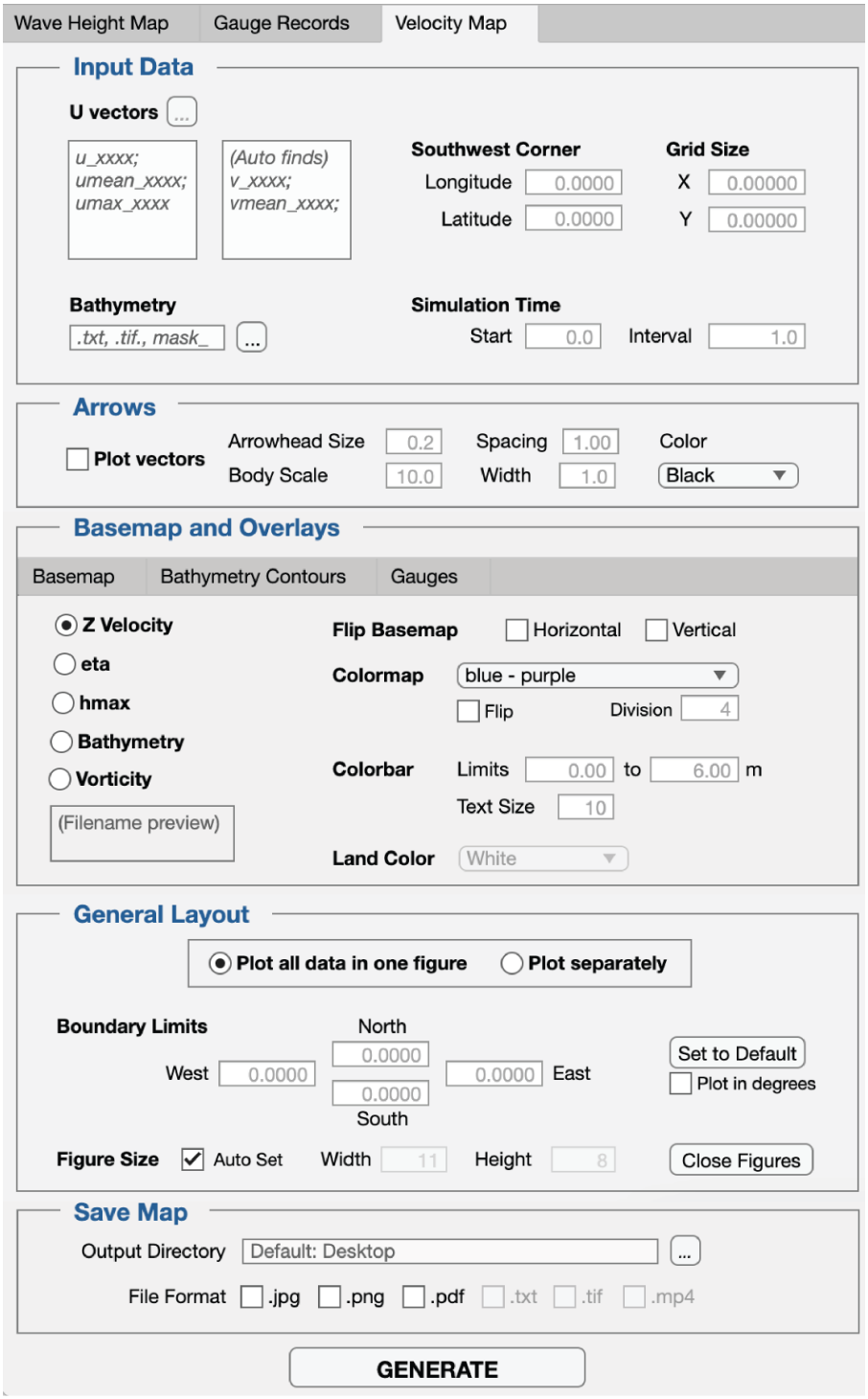
C. SAVE PLOTS



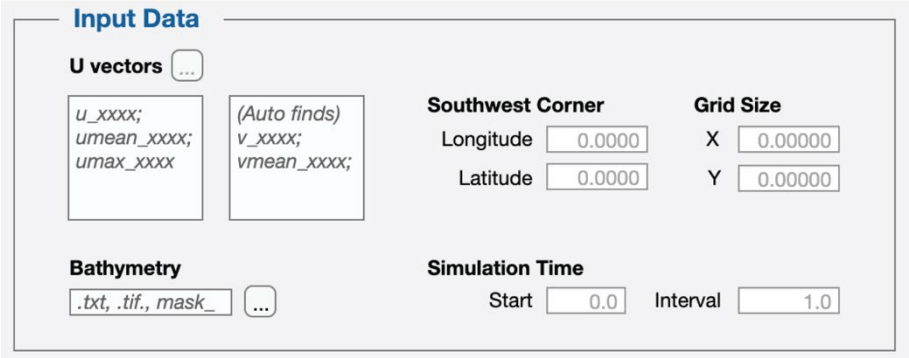
Output Directory	Folder location for saving files. To change the directory, click the button with three dots. A new folder named 'OUTPUT_FOLDER/Figures/' is created at the specified path. Default location is set to Desktop.
Filename	When 'Plot all data in one figure' is selected, the output filename will be "GaugeRecord." For the 'Plot separately' option, each output filename will be "GaugeRecord_" followed by the station number. For example, data from sta_0001 is saved as "GaugeRecord_1".
File Format	
.jpg, .png	Image files. Default resolution: 300 DPI.
.pdf, .eps	Vector-type graphic format. Default resolution: 300 DPI. Ideal for additional editing in graphic design software. For .pdf option: all parts of the map, except for the basemap, are in vector format.
.txt	Save raw data from 'sta_' files as tab-delimited text files. It is enabled when 'Plot Separately' is selected.

TAB 3: VELOCITY MAP

The U and V vector files are used in this tab to visualize velocities.



A. INPUT DATA



U vectors	Load the U-vector files by clicking the three-dot button (e.g., u_00001, umean_00001, umax_00001). The app automatically identifies matching V vector files located in the same directory alongside the U files.
Bathymetry	Import the bathymetry file by pressing the three-dot button. The file will set the boundary between land and water bodies, automatically enabling the 'Land Colour' option in the Basemap section. Accepted formats: .txt, .tif Grid size details are automatically extracted from the file. For .tif file, the southwest corner coordinates are also filled.
Southwest Corner	Input the coordinates of the southwest corner used for the tsunami simulation, as specified in input.txt. Unit: degrees, meters
Grid Size	Input the X and Y grid size as specified in the input.txt. Unit: degrees, meters
Simulation Time	Input the simulation timings to be used for figure titles. Unit: seconds For values greater than 60 seconds, the unit will be automatically converted to minutes or hours for easier readability.

B. ARROWS

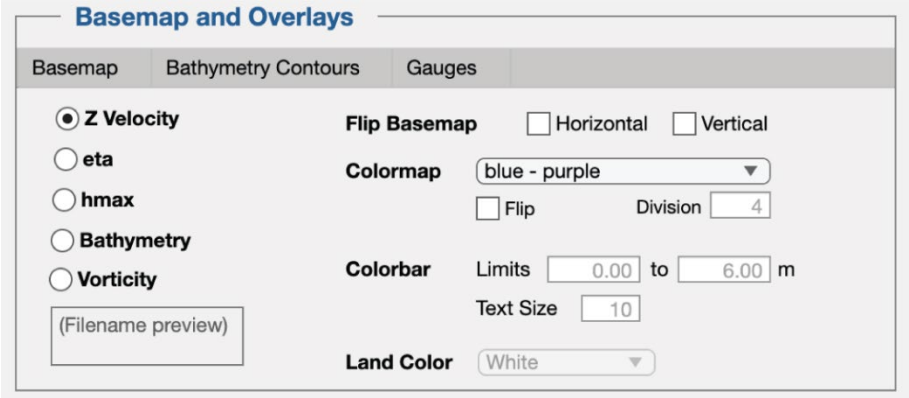
The arrows on the map represent the directional components of the U and V vectors. The length of each arrow corresponds to the magnitude of the vector. The arrow ends are positioned at the centre of each grid.

Plot Vectors	Check the box to plot the vectors.
Arrowhead Size	Modify the arrowhead size relative to the body size. A value 0.2 corresponds to 20% of the arrow body size.
Body Scale	Adjust the line length using a scaling factor. A value of 10 increases the size by 10x.
Spacing	The distance between the arrows. This is used to make the map less clustered, and the vectors clearer.
Width	Adjust the thickness of the arrow's body line.
Colour	Adjust the colour of the arrow.

C. BASEMAP AND OVERLAYS

Customize the properties of the basemap, contours, and gauges.

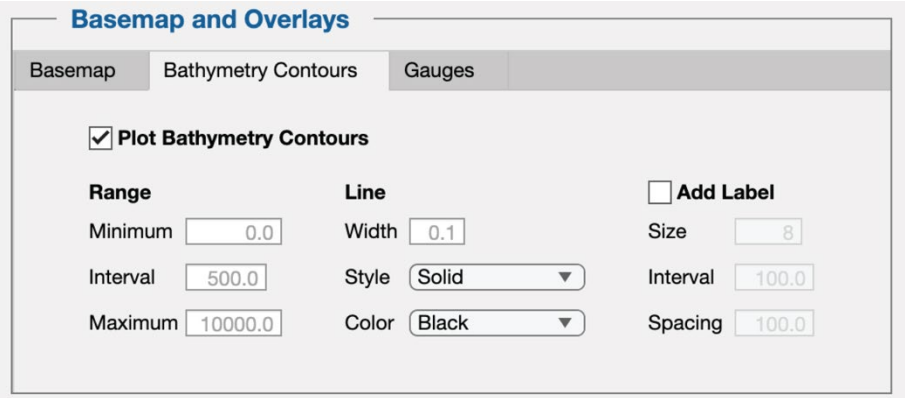
C.1 Basemap Tab



Basemap Options	Select the file type to use as the basemap.
Z Velocity	Z component of the velocity calculated using the U and V vector files imported.
eta	Use the wave height at a specific time snapshot. The app automatically identifies the corresponding eta file with the same timestamp as the uploaded U vector file, located in the same directory as the U vector files. For example, eta_0001 and eta_002 are searched for in the same folder where U_0001 and U_0002 are located. Each eta file is then used as the basemap for its respective U vector file.
hmax	Use the maximum wave height. Similar to the 'eta' option, the app automatically identifies the corresponding 'hmax' file within the same folder as the vector files.
Bathymetry	Use the bathymetry data as the basemap.
Vorticity	Uses the curl function of MATLAB to calculate vorticity.
Flip Basemap	Flip the matrix horizontally or vertically.
Colourmap	Assign basemap colours using built-in and cbrewer2 package options. Visit MATLAB colourmap and cbrewer2 package websites for visuals.
Flip Checkbox	Reverses the colourmap sequence.
Division	Defines the colour transition refinement. A higher value results in smoother transitions.

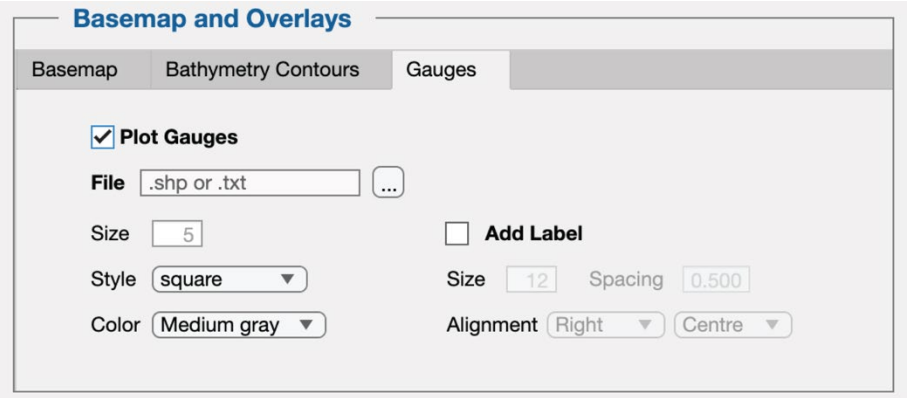
	Auto-enabled when a cbrewer2 colourmap is selected.
Colourbar Limits	Input both the minimum and maximum values to define the colourbar range to be displayed on the map.
Colourbar Text Size	Adjust the font size of the legend.
Land Colour	Choose from eleven colour options, with white as the default. It is auto-enabled when a bathymetry file is imported.

C.2 Bathymetry Contours



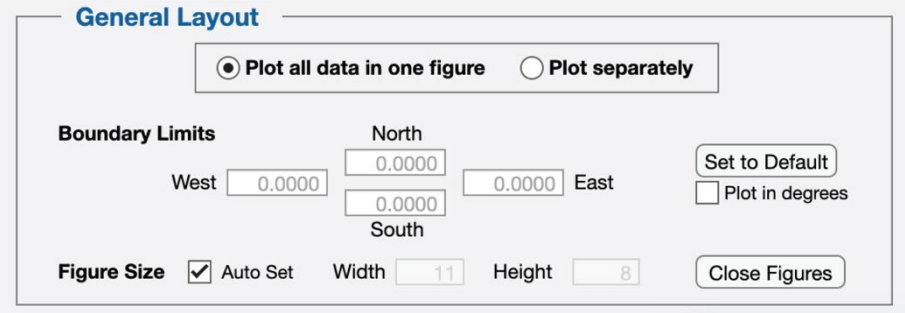
Plot Bathymetry Contours	Check the box to plot the bathymetry contours.
Range	Set minimum, interval, and maximum values in meters. Input positive values: smaller for shallow waters, higher for deeper depths, and zero for the coastline.
Line	Customize contour line thickness, style and colour.
Add Label	Add and customize the labels for water depth.
Size	Adjust the font size of the labels.
Interval	Specify label intervals for contour lines. Unit: meters
Spacing	Set the text spacing, with lower values resulting in less crowded text generation. Unit: 1 point = 1/72 inch

C.3 Gauges Tab



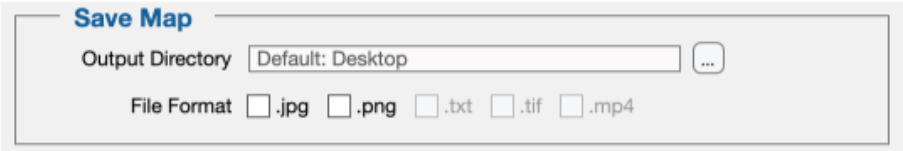
Plot Gauges	Check the box to plot the gauge stations as points.
File	Load .shp or .txt files containing coordinates. Text file format: Two columns with latitude and longitude values, without headers. Example: *31.502 138.456* *31.65 135.003* *34.215 140.626*
Marker	Adjust the style, size, and colour of gauge markers.
Label Alignment	Add labels, setting size and alignment. Set the text's horizontal and vertical position relative to the marker.

D. GENERAL LAYOUT



Plot all data in one figure	Show maps as subplots in a single window. Advantage: Easy result comparison. Downside: When too many files are loaded, the maps appear too small to view details clearly.
Plot Separately	Show each map in separate windows. Advantage: Larger, high-resolution maps. Downside: Overlapping figure windows can hinder quick analysis.
Boundary Limit	Adjust the longitude and latitude limits of the map. It is automatically set to match the boundary limits of the imported eta/hmax files.
Set to Default	Reset the boundary limit to the bounds of the eta/hmax files.
Plot in degrees	Convert map tick labels on the X and Y axes to degrees unit. Even with this option checked, input values in the 'Boundary Limit' text boxes should remain in the original numeric values. For example, if the original East limit is 200 units, and you wish to shift it 20 units to the right, input 220 units, not -140°.
Figure Size	Size of the generated figure.
Auto Set	The application determines the optimal size.
Width, Height	Manually adjust the figure size in inches. This option is enabled when 'Auto Set' is unchecked.
Close Figures	Close all currently open figures. Note: Existing figure windows are automatically closed before new ones are generated when clicking the 'Generate' button.

E. SAVE MAP



Output Directory	Folder location for saving files. To change the directory, click the button with three dots. A new folder named 'OUTPUT_FOLDER/Figures/' is created at the specified path. Default location is set to Desktop.
Filename	When 'Plot all data in one figure' is selected, the output filename will be "Vector_Output." For the 'Plot separately' option, each output file will have the same interval name as the corresponding raw file (e.g., u_00001 and v_00001 will have the filename 'Vector_00001').
File Format	
.jpg, .png	Image files. Default resolution: 300 DPI.
.txt	Save the raw U and V vector files as tab-delimited text files.
.tiff	Georeferenced raster file. Use this file format to load the map in GIS applications. It is enabled when 'Plot Separately' is selected.
.mp4	Create an animation. Frame Rate: Frames or figures displayed per second. The lower the value, the slower the animation. It is enabled when 'Plot Separately' is selected.

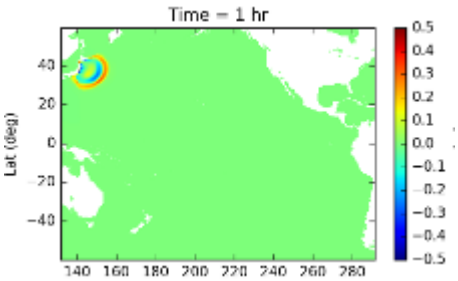
Press this button to create the maps.

Comparison of Sample Maps and App-Generated Maps

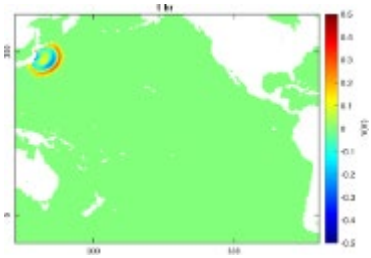
The sample maps were downloaded from the ‘postprocessing’ subfolder in FUNWAVE-TVD-Version_3.6/simple_cases/xxxx, where "xxxx" is the folder name. In each example, the folder name is used as the left column title in the map comparisons below. Note that some maps being compared are not from the exact same snapshot. The goal is to demonstrate that similar maps from the original can be prepared and generated using the FUNMAP apps.

Initial Condition: Import UVZ

Original: tohoku_tsunami

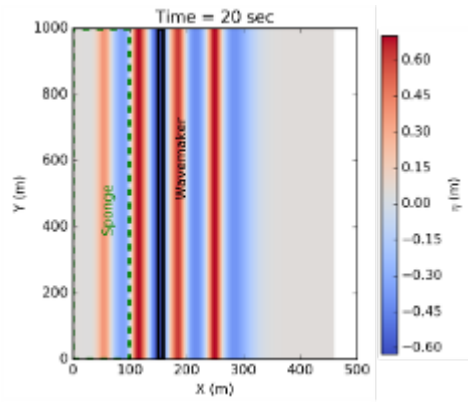


FUNMAP

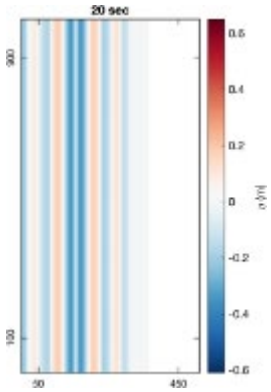


Initial Condition: Wavemaker

Original: beach_2d

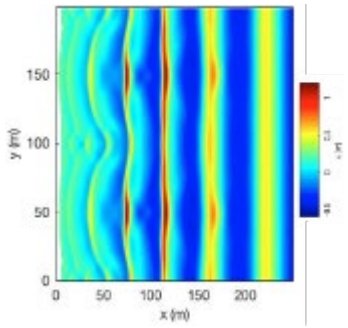


FUNMAP

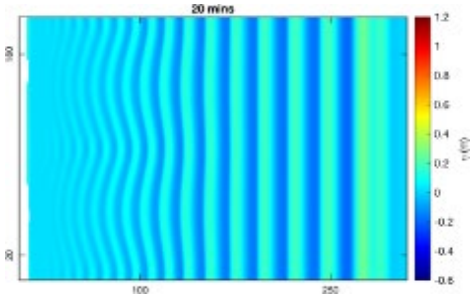


Wavemaker, Sediment Transport

Original: sediment_rip

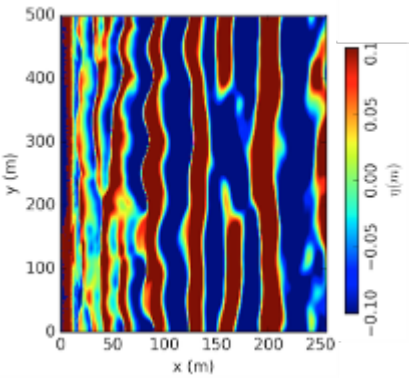


FUNMAP

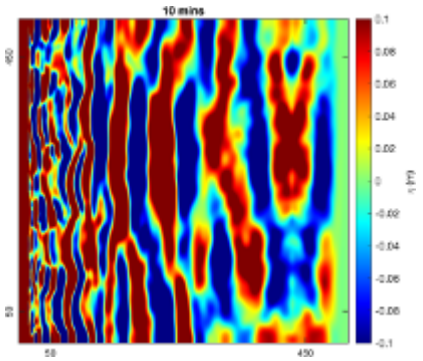


Wavemaker, Wave Height

Original: rip_2d

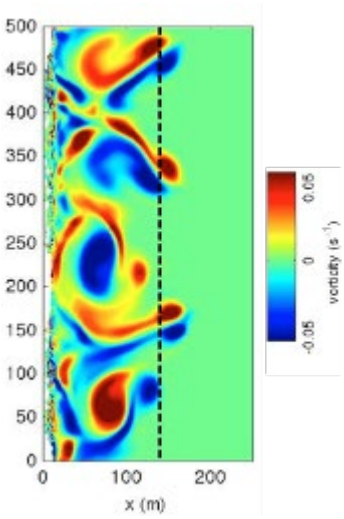


FUNMAP

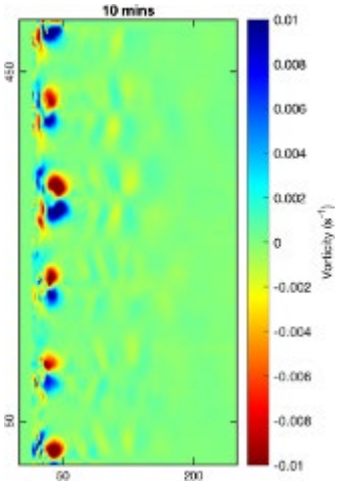


Wavemaker, Vorticity

Original: rip_2d

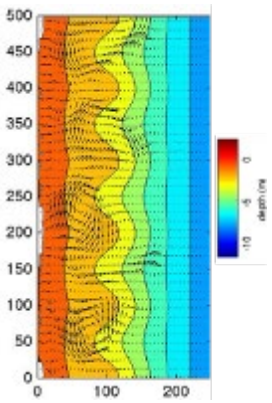


FUNMAP

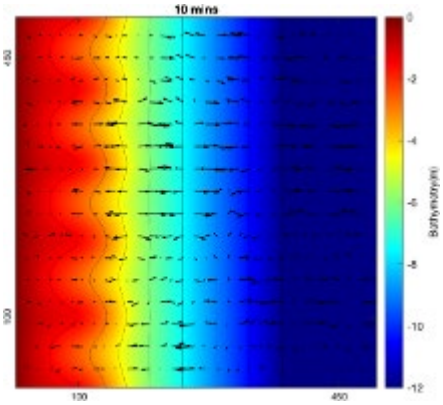


Wavemaker, Vectors

Original: rip_2d



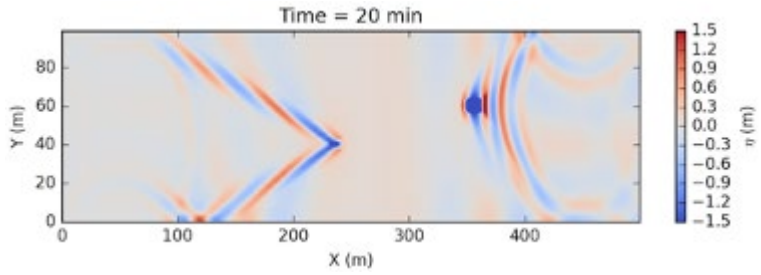
FUNMAP



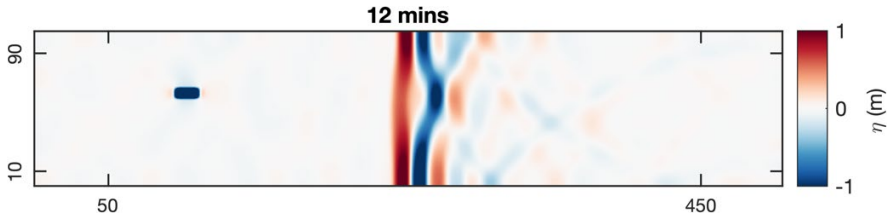
Note: The bathymetry (depth_a15.txt) imported into FUNMAP has been resized to match the dimensions of the output U and V vectors.

Initial Condition: Vessel

Original: flat_bottom



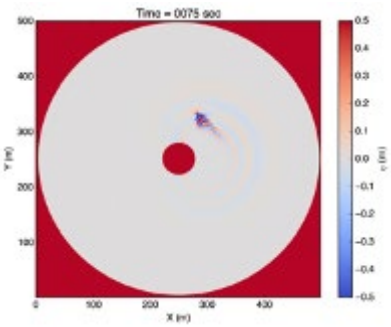
FUNMAP



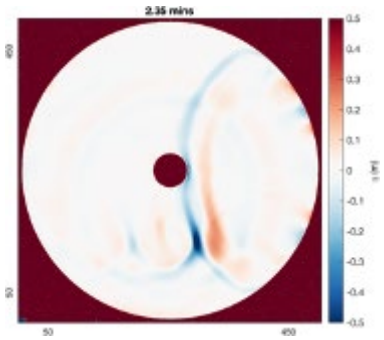
Note: The maps have different time frames.

Vessel, Wave Height

Original: vessel_island_beach



FUNMAP



Note: The maps have different time frames.