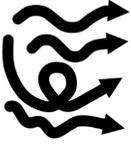# Flowfields

Introduction & summary

In this tool (Flowfields) you can compute flowfields in a region of interest, including different boundary conditions, local bathymetry and breakwater structures.
The tool requires a shoreline, cross-shore profile and tidal characteristics (you can inspect the generic data availability in the top panel of the tool). Breakwaters will also be used in case they are provided. Breakwaters fully block flow, and can be changed interactively.

The flowfields tool uses the hydrodynamic solver of Delft3D Flexible Mesh. The domain/grid is defined by the user, and generic bathymetry data is automatically interpolated to the model. Boundary conditions are based on generic tide data, and can be defined as water levels or velocities. Finally, any breakwaters are implemented as thin dams in the model, these can be moved interactively.

How to use the tool

1. Open a new project
2. Start in Generic data by defining a Coastline, bathymetry (a cross-shore slope, spatially varying bathymetry is optional) and tidal characteristics.
3. If required, define breakwater structures.
4. Open the Flowfields tool.
5. Choose a grid resolution and click on the "Draw grid outline" button ( ).
   Now, drag a grid extent on the map, while keeping your left mouse button pressed:

6. Press the "Rotate/move grid outline ( ) button to move and rotate your domain.
   a. Hold CTRL to rotate the grid outline. the blue box indicates the grid extent, while the arrow indicates the offshore direction.
   b. Hold SHIFT to move the grid outline. the blue box indicates the grid extent.
7. The type of model boundaries can be set in the section 'boundary conditions'. You can choose from the following types of boundaries:
   a. Water levels, defined on the offshore boundary
      These are defined by the tidal components in generic data
   b. Currents, defined on the lateral boundaries
      These are defined by the max. horizontal tidal velocities in generic data
8. A spin-up time can be defined (in HH:MM:SS format)
   a. The model will only show output on the map after the spin-up time has passed

9. By pressing the PLAY button ( ), the simulation is started

10. After completing the spin-up period, the model will indicate the water levels and flow velocities:
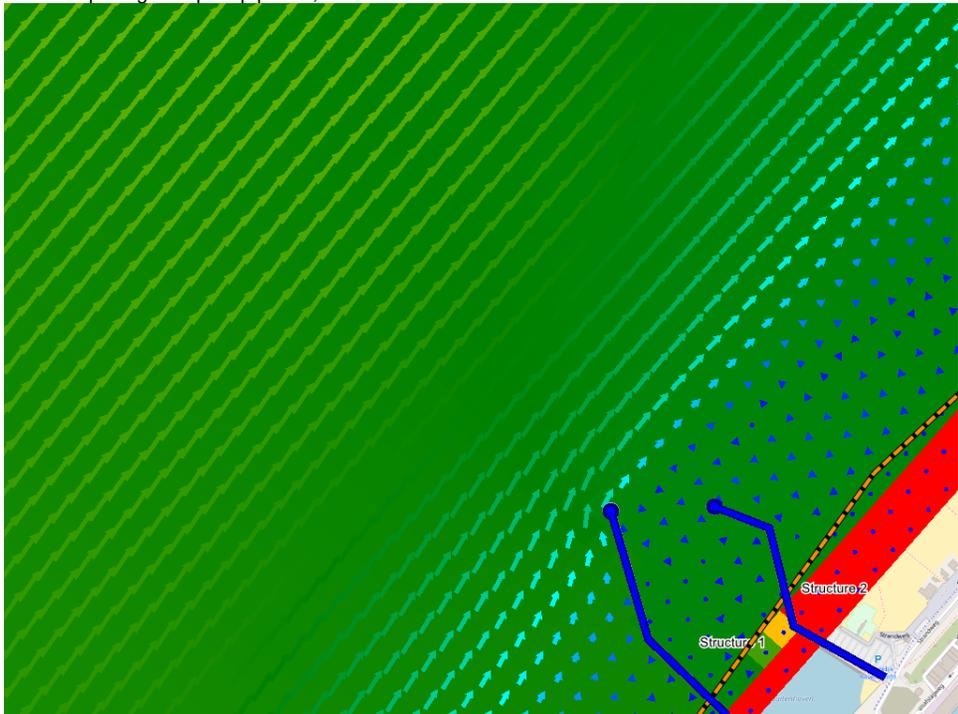
11. The plotted variables can be changed in the section "Visualization".
    a. A combination of 'currents', 'water levels', 'bathymetry' and 'grid' can be made.
12. During the simulation, any breakwater can be moved around by clicking on the button "Move points of temporary structures or delete structure" (

)
    a. The interactive flowfields will automatically update
    b. Note that you can also save your breakwater changes to the generic data (for use in other tools), by pressing the button "Set temporary structures to structures"
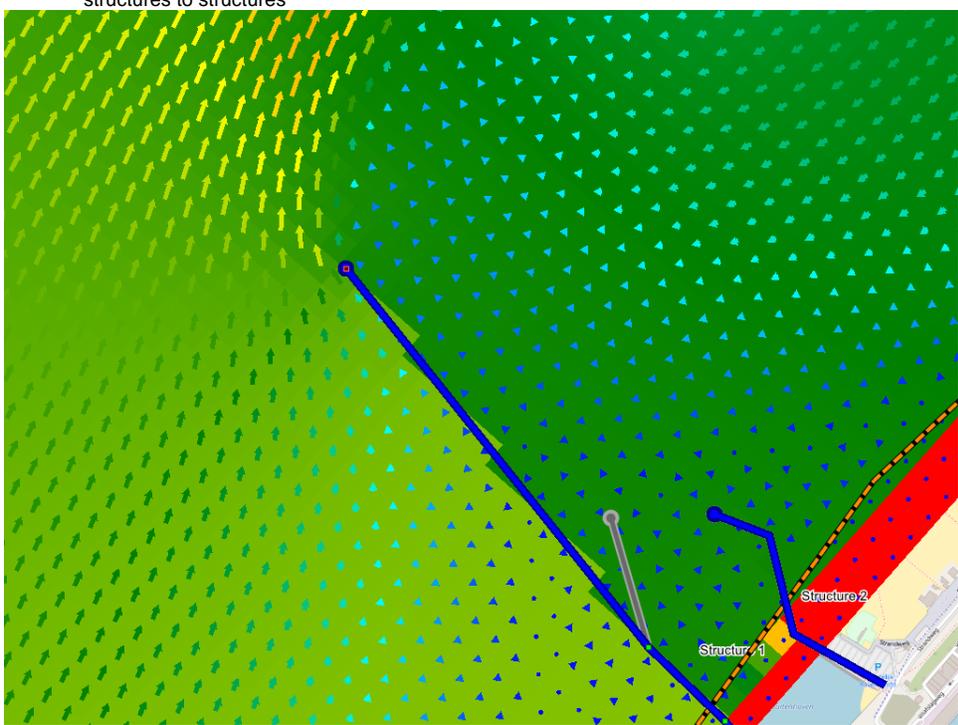
13. You can interact with the model progress:

    a. Click to start or resume the model (depending on the status of the model)

    b. Click to pause the model

    c. Click to compute a single timestep

    d. Click to stop the simulation and redefine a model domain

Please be aware that the speed of a Delft3D Flexible Mesh model (and thus this tool) scales:

- Linearly with with the number of gridcells
  - E.g. increasing the *area* of the entire model domain by a factor 2, will yield a model that is 2 times slower
- To the power 3 (!) related to the defined cell size
  - E.g. changing the cell size from 200 to 100 m (for the same model domain), will yield a model that is 8 times slower (4x as much gridcells (in x and y direction), see item above, while the model timestep also typically decreases by a factor 2)
- Linearly with the maximum flow velocity
  - E.g. increasing the maximum velocity by a factor 2, will yield a model that is 2 times slower (e.g. by changing a flow boundary condition or by flow contraction using breakwaters)

All other tool modifications, such as visualisation, have a limited effect on model and tool performance.

Temporary files

The model files are stored in the following temporary folder, for use outside of CoDeS:

- *C:*\Users*smith*\AppData\Roaming\Deltares\CoDeS\Workdir\ (note that in this case, *C:* is the windows-drive, and '*smith*' the example username

User input fields

A number of user input fields and funtionality is provided to modify the model simulation to reflect the local situation. Default values are provided for each input field. The following inputs are defined:

Model domain

- This section allows you to setup the domain of the Delft3D Flexible Mesh model. Apart from buttons that allow you to draw a grid outline ( ) and move/rotate it ( ), the following keywords are defined:

	Default Value	Unit	Min.	Max.	Brief Description
Cell Size	1000	m	1	Inf	Cell size of the gridcells. Note that the total number of cells may not increase above 25.000
Grid rotation	0.4	<DEGREES>	-Inf	Inf	Rotation of the grid, ideally in between -360 and 360 <DEGREES>

Visualization

	Default Value	Unit	Min.	Max.	Brief Description
Vector scale	1	-	0.0000001	Inf	Scaling of the velocity vectors. Smaller values show larger arrows. Note that negative values flip the arrows

General

Apart from the play, pause, 1-step and stop buttons (, , and), the following keyword is defined in this section:

	Default Value	Unit	Min.	Max.	Brief Description
Spin-up time	0	HH:MM:SS	00:00:00	Inf	Spin-up time (period) after which model results are updated on the map (the spin-up time is ignored during visualization)

Also note that the "current model time" and "time step number" are shown here for information.

Tool limitations

The Delft3D Flexible Mesh model is a rough representation of the area, and governed by the limitations to the Delft3D Flexible Mesh modelling Suite, as well as governed by the quality of the boundaries, bathymetry, etc.

Furthermore, the models are 2-dimensional and do not account for temperature, salinity or sediment transport.

Example cases

The following file (.dsproject) can be downloaded and loaded into CoDeS to serve as an example. It is not based on actual projects, but are intended as illustration.

Unzip the files to a convient folder (but make sure the file structure does not change), and open the .dsproject file from within CoDeS.

Flowfields...xample.zip