SNL-Delft3D-CEC

3D/2D modeling suite for integral water solutions

Building an Advanced Model
Overview

This tutorial will aim to:

- Edit grids with attribute files (polygons)
- Create depth files through interpolation and extrapolation
- Manually add boundaries to geographical grid
- Run a simulation
Delft 3D initialization

- Start the Delft 3D program.
- From the Windows command line, run d3d_menu.bat
Grid

- From the main menu, select Grid
From the Grid and Bathymetry menu, select RGFGRID
In this tutorial, we will create our own 400x370 grid to be edited into the shape of SF Bay.

Go to Operations → Create Rectangular or Circular Grid
Create your own Grid

- Create grid to match these values.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Gridcells in M-Direction</td>
<td>400</td>
</tr>
<tr>
<td>Number of Grid cells in N-Direction</td>
<td>370</td>
</tr>
<tr>
<td>Delta X [m]</td>
<td>255</td>
</tr>
<tr>
<td>Delta Y [m]</td>
<td>255</td>
</tr>
<tr>
<td>Origin X [m]</td>
<td>527310.526</td>
</tr>
<tr>
<td>Origin Y [m]</td>
<td>4141700.246</td>
</tr>
<tr>
<td>Rotation left [deg]</td>
<td>0</td>
</tr>
<tr>
<td>Radius of M-Curvature [m]</td>
<td>0</td>
</tr>
<tr>
<td>Uniform M-Fraction [-]</td>
<td>1</td>
</tr>
<tr>
<td>Maximum Size / Delta X [-]</td>
<td>5</td>
</tr>
<tr>
<td>Uniform N-Fraction [-]</td>
<td>1</td>
</tr>
<tr>
<td>Maximum Size / Delta Y [-]</td>
<td>5</td>
</tr>
</tbody>
</table>
This is what you should see.
RGFGRID

- Select File → Attribute Files → Open Polygon
Select “SFShoreline2.pol”
RGFGRID

This is the result.
Select Operations → Delete → Grid

This will remove all points “inside” of the polygon.
This is the result.
Select File → Export → RGFGRID
**RGFGRID**

- Save the grid file as “400x370.grd”.
- Close RGFGRID and import it again.
Editing the Grid

- Import the newly created grid.

- In order to make creating the boundary easier, grid cells will be deleted to allow for smooth boundary edges instead of jagged.
Editing the Grid

The end goal is to transform the boxed region on the left to the one on the right.
Creating a Linear Boundary

- Remove parts of the grid with the “Block Delete Interior” tool.
- Use the tool to delete grid cells from the bottom, left side, and top of the SF Bay boundary
Creating a Linear Boundary

- Create a block by selecting four points on the grid.

- Have top points be on the same line, and have bottom along the bottom corner of the grid.

- Right click to delete the grid points within the selected grid points.
Creating a Linear Boundary

- The bottom should now be smooth.
- Repeat process for left side and top.
Delete Left cells

Right click to delete cells inside blocked area.
Delete Cells on top

Right click to delete cells inside blocked area.
Saving Your Grid

- Export the completed grid as “400x370.grd”
Exit RGFGGRID and return to the Grid and Bathymetry menu.

Select QUICKIN
QUICKIN

- Select File → Import → RGFGRID and open the file that we just created.

- QUICKIN will be used to add attribute files and interpolate and extrapolate sample data.
This is what you will see.
QUICKIN

- Select File → Attribute Files → Open Samples
Adding a Sample File

- Select test.xyz
- test.xyz contains data samples at each (x,y) coordinate on the grid.
Adding a Sample File

- This is the result. Notice the samples (z-values) range from 2.37 to -112.
Adding a Sample File

- Flip the signs of the sample values

- Select Operations → Combine Samples and Uniform Value → Multiply Samples with Uniform Value...
Adding a Sample File

- Change from -999 to -1, click OK.
- A warning window will pop up. Click Yes.
This is the result.

Notice the difference in the sample values from before.
Grid Cell Averaging

- Select Operations → Grid Cell Averaging

- This will interpolate the samples on to the grid.
Sample Removal

- Select View → Samples → No Samples
Sample Removal

- The removal of the contour indicates that the interpolation was successful.
• Select Operations → Internal Diffusion
• This will extrapolate the depths to the remaining unpopulated cells.
- Select File → Export → Depth
- Save as sfbay.dep
- This depth file will be used in the Flow setup.
Return to the main Delft 3D menu.

Select Flow
Flow

Select Working Directory
Choose the folder currently being worked in.
Return to the main menu and select Flow Input.
**Delft3D-FLOW**

- Select File → New
Delft3D-FLOW

- Under the Description tab, label your .mdf file.
Delft3D-FLOW

- Under Domain → Grid, select Open Grid
Delft3D-FLOW

- Select 400x370.grd
Delft3D-FLOW

- Select Open grid enclosure
Delft3D-FLOW

- Select 400X370.enc
Select Bathymetry and then Select Open
Delft3D-FLOW

- Select the 400x370.dep depth file
Under Time frame, fill in the Reference date, Simulation start time, Simulation stop time, and Time step.
Delft3D-FLOW

- Select the Boundaries Tab
- Then, select View → Visualization Area
Visualization Area

- Select Edit → Open Boundaries
Visualization Area

- Select Edit Mode → Add
Visualization Area

- Using the mouse, drag and add boundaries along the edge of the grid.
Visualization Area

- The left side should look like this.
The boundary on the far right side of the bay (river inlet) could be added to allow for the input of river data.

For the purposes of this tutorial, this will not be necessary.
Labeling Boundaries

Rename the newly created boundaries to north, west, and south depending on the boundary’s location on the grid.
Editing Boundaries

- Change the Forcing type for all boundaries
- Forcing Type → Time Series
Edit flow conditions for all boundaries (north, west, south).
For each boundary, edit flow conditions by entering water level data.

For our purposes, have the flow conditions match this table (entry for “dd” will change depending on your model). Or choose to import water level data from external sites.

Add rows to table by selecting: Table → Insert row above.

Note: Data could be added manually through the GUI or entered in your .bct file.
Click on Open/Save to save the Boundary definitions and Time-series flow conditions
Editing Boundary Flow Conditions

- Save Boundary definition as bnds.bnd
- Save Time-series flow conditions as boundaries.bct
- You have now created a .bnd and a .bct file.
Then under Physical parameters → Constants, add the gravity and water density
Under the Roughness tab, use the Chezy Roughness Formula with Uniform U and V
Physical Parameters

- Under Viscosity, select Uniform Horizontal Eddy Viscosity
Numerical Parameters

- Under Numerical Parameters, match these values.
Monitoring

- Using the visualization area, select observation points and save.
- Or open file of pre-selected observation points.
Under the Output → Storage tab:
After saving the .mdf file, return to the main FLOW menu.

Select start. And choose your .mdf file.

Expected run time: ~7 minutes