

- The harbour can best be modelled with, for instance, a grid of 100 x 2 cells grid with $dx = 100$ m and $dy = 1000$ m. Note that this grid appears in FLOW as 102 x 4, of which the first and last rows and columns are dummy (Fig. 1)

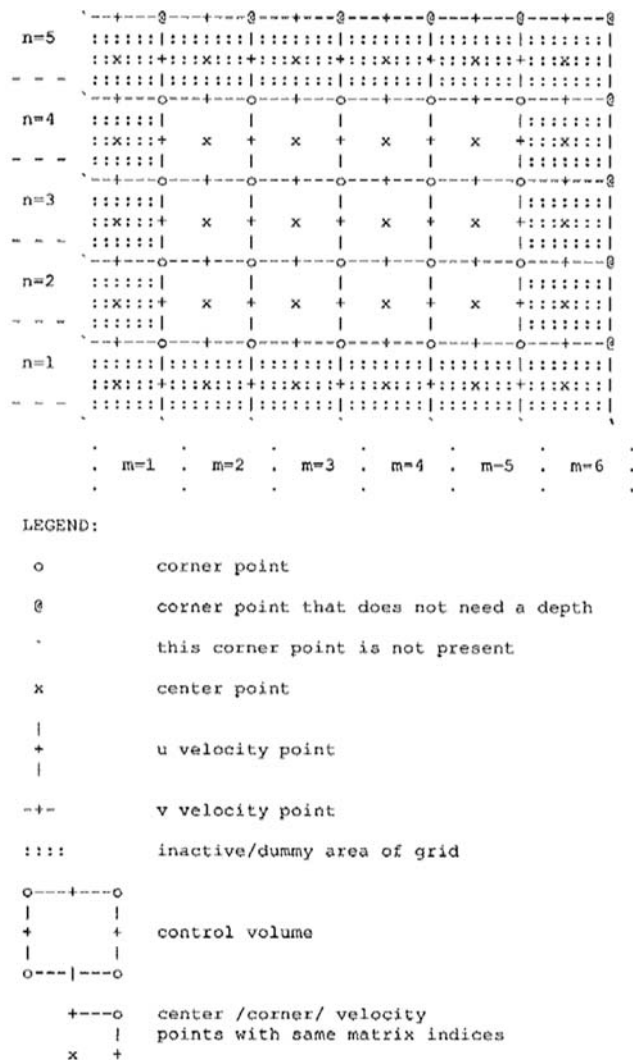


Fig. 1: Grid definition example: A 4x3 grid of active volumes requires a 5x4 grid of active corner points, and a 6x5 active numerical grid which has 5x3 active u velocities 4x4 active v velocities

- Use the Delft3D grid editor RGFGGrid to make a new grid file. Select from the menu Operations > Create rectangular grid. Note that a 0 is considered as no data value so the origin is by default at (1,1).
- Open Delft3D and select the directory where you will run the simulation. We advise that you choose d: to start and create a separate directory per simulation in that drive. In addition remember to copy this to your own USB key or network drive to ensure that you have a copy as d: is accessible to anyone who logs in to the PC. **It is recommended to keep track of what every simulation contains in an Excel file.**

- FLOW SETUP
 - Go to the FLOW module.
 - Start the FLOW input editor.
 - Go through the input items one by one.
 - Description: Describe shortly what the simulation contains and what is meant for. This is not necessary yet in this experimental stage.
 - Domain: select the grid *.grd file that you just created. Note that when you load a 100 x 2 grid, the input editor says the number of grid cells in m and n direction are 102 and 4 respectively. The grid editor also counts the inactive rows and columns, whereas the "Create rectangular grid" option in the grid editor does not.
 - Time: Set reference time always to a sensible date like January 1" 2009 00:00.
 - Time: Simulate for instance 5 days, which contain 10 tidal cycles.
 - Time: Estimate the time step using the Courant wave number. Delft3D-FLOW is implicit, **so wave Courant numbers above one are also stable.**
 - Boundaries: By default all boundaries are closed, so the grid represents a lake. Add for instance one open boundary on the west short side and name it West or W for example.
 - You can add the boundary graphically after clicking in the menu View> Visualisation area. The boundary locations are stored in a separate file with extension *.bnd.
 - Boundaries: Impose a tide at the boundaries using the forcing type: harmonic (time series are less convenient). Frequencies are in deg/hr, so 30 is the S2 tide, and 0 is a constant offset. Each boundary segment has 2 end points, which are referred to as point A (start) and B (end). Start by making one boundary at the west side, and give points A and B the same values. The harmonic forcing data are stored in a separate file with extension *.bcr. You can add additional components by clicking the table menu, but for this exercise you only need the default two components: one constant and one tidal component (Fig. 2). For an example calculation you could use the following equation:

$$\text{water level} = A_0 + A_1 \cos(\omega_1 t - \phi_1) + A_2 \cos(\omega_2 t - \phi_2)$$

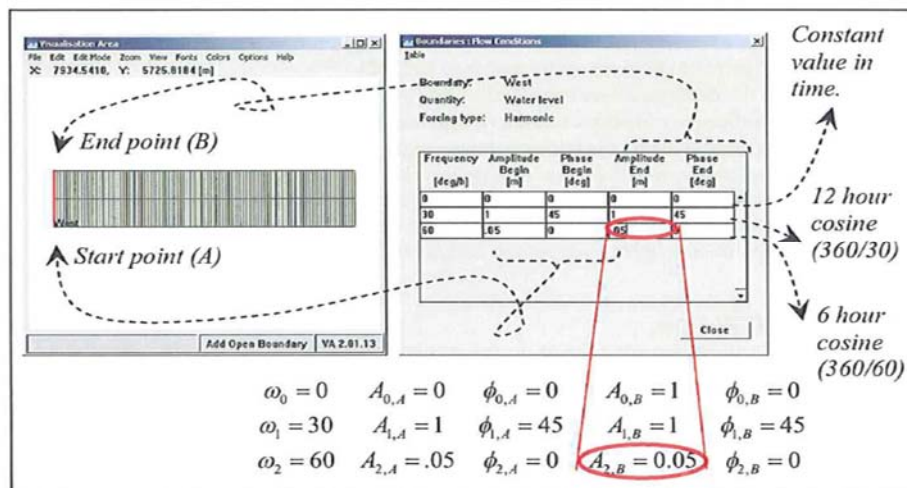
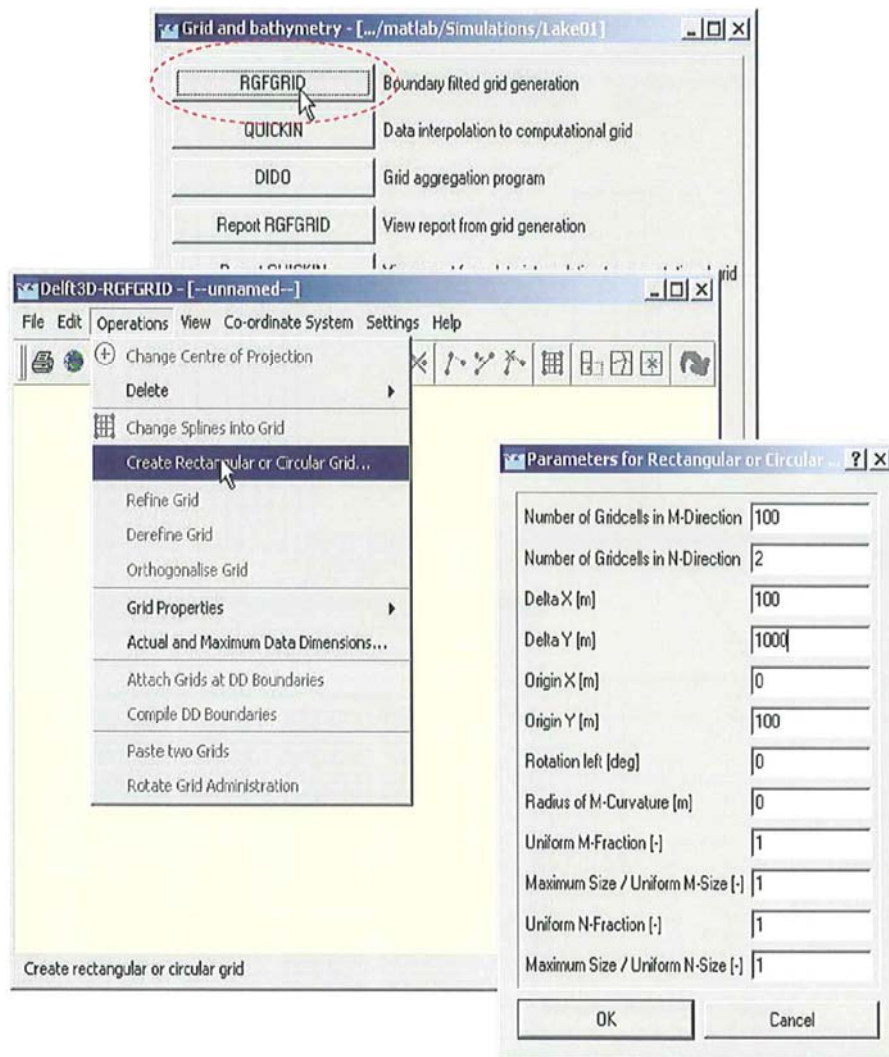


Fig. 2: Relation between harmonic components (with respective amplitudes and phases) and boundary end points (always 2). Therefore in this case 12 values need to be specified for the 3 harmonic components at one boundary.

- Note that water levels are defined positive upward with respect to the reference level $z=0$, while bottom depth are defined positive downward with respect to $z=0$. A depth of 10 m and a boundary offset value of 1m thus give a water depth of 11m. However for this case we recommend a depth of 11 m and a 0m offset, so that $z=0$ is the equilibrium water level.
 - Monitoring: add a few monitoring stations at strategic locations. For real models these **positions are typically oil platforms, harbours, instrumentation and last but not least** project locations. At these stations you can save high-frequency model output. These **data are stored in a separate file with extension *.obs**.
 - Output: Store map output to file (where information on whole grid is stored) often (trim***.dat). You can make (a) cross section and (b) plan view plots with these data. For small intervals and with large models this will give very big files (maximum possible is 4.2 GB). You do not need to save data to the communication file, that is only needed when you use also other modules (like waves, water quality, morphology)
 - Output: at a few strategic locations you can save high-frequency data. Write time series (history) output at predefined observation points every timestep (stored in trih-***.dat). You can look at time series with these data. The difference between history and map data is most useful for long mns on large grids. For the small short term grids in this exercise the distinction is small.
- Use a separate directory for every new simulation you want to keep. When doing a second simulation, copy the relevant input files of the previous nm to a new directory and edit those files. The important input files are:
 - *.mdf overall ASCII editable input file
 - *.bnd boundary locations
 - *.bch boundary harmonics
 - *.grd curvi-linear grid definition
 - *.enc curvi-linear grid enclosure file (comes with *.grd)
 - *.dep spatially varying depth field (optional)
 - *.ini spatially varying initial water levels (and, optionally, velocities, salt)
 - tri-diag* ASCII file with error messages
 - runid ASCII contains the RUNID of the last run you performed.
 - Remember to set the working directory every time for each run. The first letters of the *.mdf file are the so-called unique RUNID. The RUNID may not contain spaces. It is easiest to keep these the same for every run, and only change the directory name like harbour01, harbour02, etc. Make a list of all runs in for example Excel, so after a while you still know what you did in each run.
 - Give sensible names to the boundaries and observation points, like 'west01', 'L/4', 'middle_of_harbour'. By default the names are the (m,n) coordinates which is not always useful.
 - By default there is online visualisation. When it runs and you want to stop it, press the <s> key until the online visualisation notices you and stops. We suggest that you turn off online visualization and view the output files while the model is still running rather than use the online visualisation.
 - Visualization (post processing) of results with either:
 - QuickPlot Gill (see also QuickPlot manual)
 - You do not need to close the FLOW input GUI when you start running the model. The same applies for matlab and Quickplot.
 -

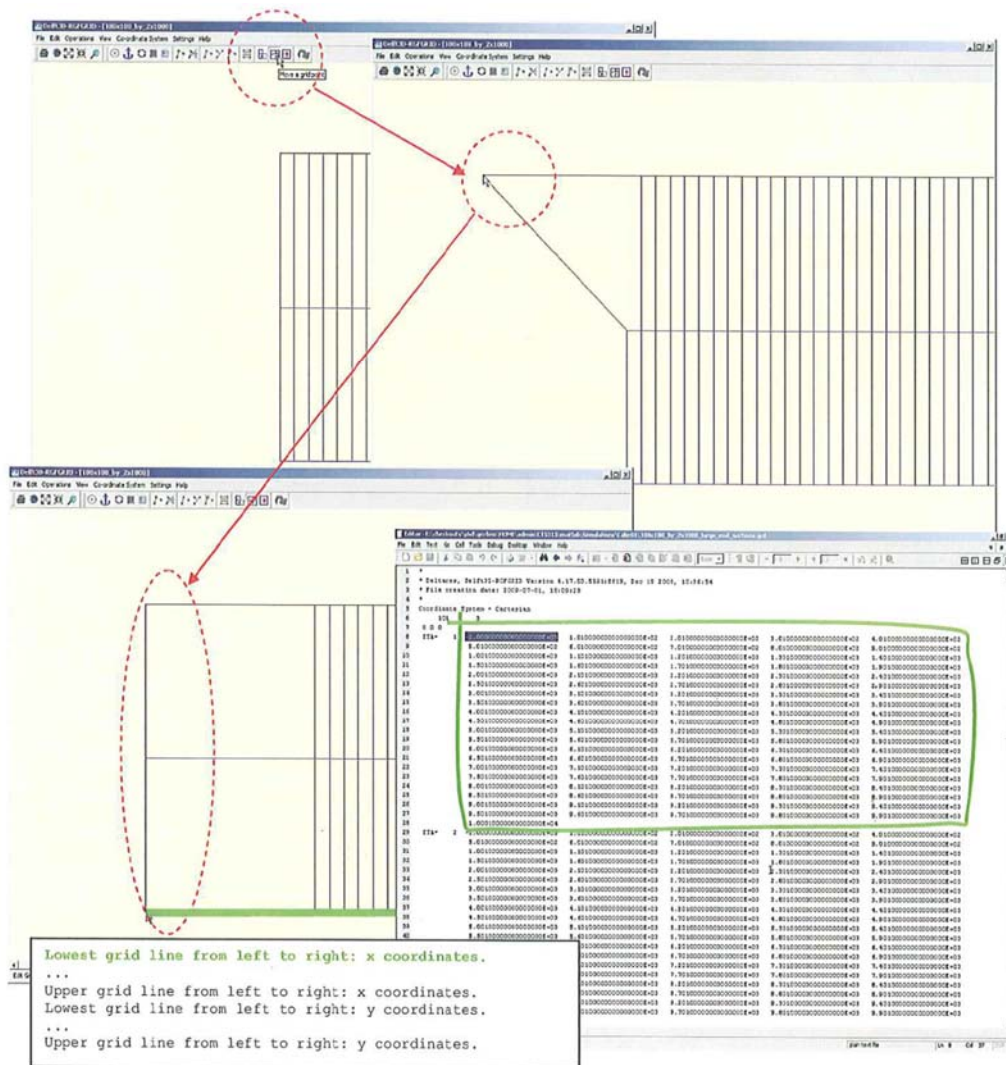
CE5377/CE6077 Delft3D Exercises: General Hints

- For all exercises an orthogonal grid with constant grid sizes has to be used. The figure and steps below show you how to create such an orthogonal grid.
- The grid cells you see in the editor are the active grid cells. Internally, Delft3D adds a band with dummy cells around the grid. The actual grid size will therefore be 1 bigger in each direction.
- Save the grid with a filename that allows you to identify the grid based on the filename.



CE5377/CE6077 Delft3D Exercises: General Hints

For Exercise 3 the effect of grid size near the vertical walls has to be assessed. The orthogonal grid of exercise 1-3 can be modified for thus purpose. The figure below shows how to modify the orthogonal grid created for exercise 1.



- Click the button "Move a grid point".
- Drag the leftmost upper grid corner point to the left, keeping an eye on the orthogonality.
- Export the grid. The grid data can be viewed with a text editor (e.g. wordpad, Matlab editor). Optionally, the grid coordinates can be modified in the text editor to make the grid perfectly orthogonal. In *grd file the upper left number represents the lower right corner. Each block indicated with ETA~ represents one horizontal line. First all x-coordinates are listed, then all y-coordinates.