

Introduction to flood inundation modelling

Computer models are widely used in geosciences to simulate environmental processes. The best know of these are probably the weather and climate models, which are used to simulate the dynamics of the atmosphere for the purpose of forecasting tomorrow's weather or estimating the impact rising CO₂ concentration in the atmosphere might have on climate. Although these models are often very complex and their simulation subject to uncertainty, they provide invaluable data, which when analysed, can be used by environmental managers to help make decisions, assess risks or formulate policy.

We can apply much the same approach used by climate scientists to almost any part of the earth system (e.g. cryosphere, hydrosphere, lithosphere) to make predictions of how a change in something might impact on something else. In this case we are going to look at the hydrosphere and specifically how a large wetland system called the Inner Niger Delta in West Africa might be affected by future changes in river discharge.

River discharge is the volume of water in cubic meters (m³) passing a point on the river every second (e.g. discharge = m³s⁻¹ of water passing a point on the river). To do this we will use a sophisticated numerical model developed by the Hydrology Group at the University of Bristol called LISFLOOD-FP (<http://www.bristol.ac.uk/geography/research/hydrology/models/lisflood/>). Therefore, the aim of this clinic is to provide an introduction to recent research into flood inundation modeling and apply that to a real problem. In the process we will learn how to analyse data from numerical computer models within a GIS environment.

The LISFLOOD-FP model you are using today is unique and almost certainly the most detailed numerical model of the Inner Niger Delta in existence. In this clinic, we will look at the data sets needed by the model and then run a simulation for ourselves of the annual flooding on the Inner Niger Delta in Mali. The simulation will predict the flood inundation between 2002 and 2009 given the inflow discharges for every day of that period. After the simulation has run, we will view some of the results in ArcMap (ESRI) or QGIS (<https://qgis.org/en/site/>). In groups of 3-4 you should then discuss potential scripts/tools you might need to write (Python (also within QGIS), R, Matlab, etc) to process the raw model outputs into useful information. At the end of this session, you should be able to set up and run the model as well as analyse some of the results obtained!

Download the data

You can download the clinic data and model folder from Google Drive using the following link:

https://drive.google.com/open?id=1o9MSiBev1zKGY9An0Tt-a_g-Ulrq4kiH

Viewing the model raster data (in QGIS)

First let's take a look at the input data needed by the model. The model represents the earth surface using a raster grid, so it's easy to view this in a GIS. Open a new QGIS project and save it with a memorable filename in a location of your choice. The default coordinate system setting in QGIS is the one we need (EPSG 4326, WGS_1984), which you will find in the predefined coordinate systems under:

The most important data set used by the model is the Digital Elevation Model (DEM). This is a raster data layer where each cell in the raster contains the ground elevation in that cell. This data set is much like a remotely sensed image that you may be familiar with except that each pixel contains a measure of elevation rather than electromagnetic radiation intensity. These were measured by a radar satellite mission called the Shuttle Radar Topography Mission <http://www2.jpl.nasa.gov/srtm/>, which you might want to read up on.

The DEM data are stored as an ASCII raster text file which you can view using any text editor. To view the ascii data right click on the file and select "Edit with SciTE in New Tab" or open with any other text editor. The text editor will display the raw data starting with the following header

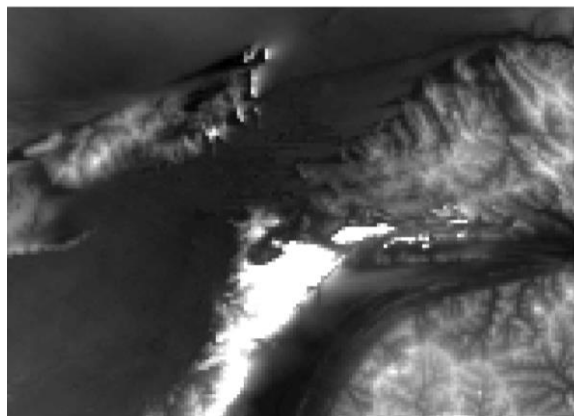
```
ncols    165
nrows    118
xllcorner -6.050000
yllcorner 13.375000
cellsize  0.033333
NODATA_value -9999.000000
```

This header describes the number of columns and rows in the raster data set, the coordinates for the lower left corner of the raster, the size of each raster cell (in decimal degrees) and the value used to represent no data cells. Below this header are the elevation of each cell in the raster.

To open in QGIS we will first convert the data to the raster format preferred by a GIS software.

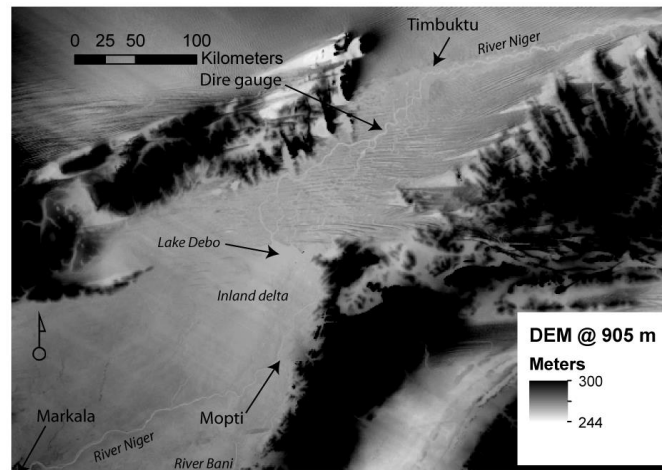
In QGIS go to Raster->Translate(Convert Format)

When the dialogue box opens locate the file *NigerLLdem.asc*. Change the file type of the input file to Arc/Info ASCII Grid (*.asc). Click open. Set the output location to your folder and the type to GeoTIFF. Call the raster you are creating DEM. Finally change the no data to -9999 then press OK. QGIS should now convert the ASCII file into a raster and display the DEM.



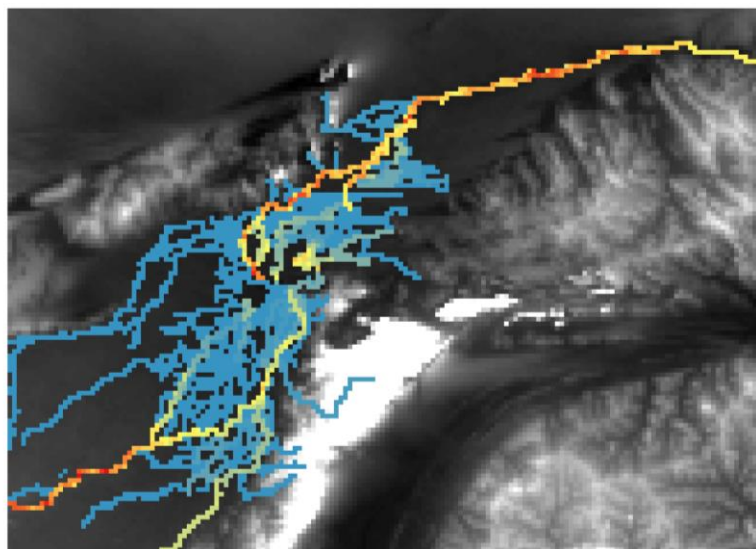
Below is an annotated high resolution version of what you should see. You can find out more details about the locations in this area using Google maps

<http://maps.google.co.uk/maps?hl=en&ll=15.188784,-3.90564&spn=4.44668,8.453979&t=h&z=8>



Use the identify tool to look at the range of elevations in the DEM, which direction do you think the river flows? Use the measure tool to find out how big the area we are modelling is (be careful to check the distance units!).

There are two other raster data layer used by the model *NigerLL.width.asc* and *NigerLL.banks.asc*. Open these in the same way as the DEM. Once open you may want to right click on the data sets in the Table of Contents side bar and select properties. On the Styletab you can set the colour ramp and stretch type, as well checking the Display Background Value box which will make the zeros in the raster layer transparent.



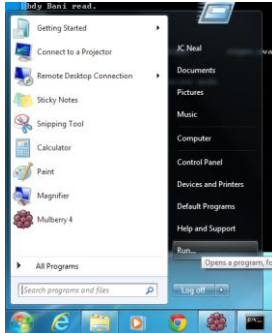
The width data set is a map of river channel widths in every cell of the DEM, while the banks data set is an estimate of the riverbank elevation that has been derived from the DEM. Together these three data sets describe the physical attributes of the land surface over which we are going to simulate flood inundation.

Running the model

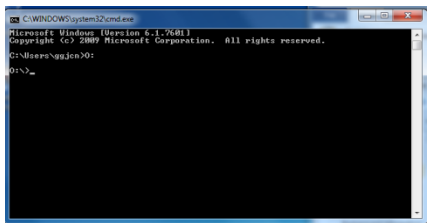
In the folder Inundation_model you will see an executable file called LISFLOOD.exe (if you are using Windows). This is the computer model that we will use to simulate flood inundation on the Delta.

The model runs from the windows command line. To open the windows command line and run the model you will need to do the following:

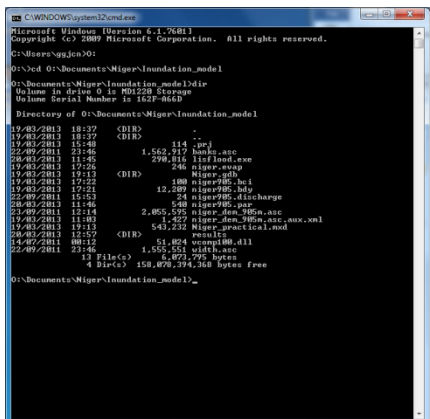
1. Got to start->run



2. In the dialogue box type the letters cmd then press OK, a command window will open. With this we can send instructions directly to the windows operating system. You will see that the windows currently says C:\Users\YOUR USER NAME.
3. First we need to change the drive to the location of the LISFLOOD model. On my machine the files are on the o: drive. To change to this drive type
O:
into the command window and press enter. However your files may be on a different drive and if you are using the local c: you don't need to change drive!



4. We now need to move to the folder containing the model. On my machine this is `O:\Documents\Niger\Inundation_model`
But yours will be different if you have saved the data somewhere else.
5. To change to another folder type `cd` followed by the location of the model, for example `cd O:\Documents\Niger\Inundation_model`
6. Typing `dir` and pressing enter will tell you which files are in the folder



7. We can now run the executable. Type the following into the command line and press enter

```
lisflood_win_double.exe -v NigerLL.par
```

The model will start running and display the something similar to the following

```
C:\WINDOWS\system32\cmd.exe - lisflood -v niger905.par
O:\Ncd\O:\Documents\Niger\Inundation_model>lisflood -v niger905.par
*****
LISFLOOD-FP version 5.8.2
*****
Loading parameters...
Using sub-grid channels and acceleration formulation
SGC changed to 0.04500
SGC multiplier changed to 0.10000
SGC exponent changed to 0.72000
Loading parameters done.

Start Date: 28/3/2013
Start Time: 12:45:18

Loading DEM: niger_dem_905m.asc
483x426
UL corner: (150686.187468,1518877.742858)
NODATA_value: -9999.000000
Done.

Loading SGCbank: banks.asc Done.
Loading SGCwidth: width.asc Done.

Loading boundary condition file: niger905.bdy
QGR at point (164218.000000,1514707.000000) Niger
QGR at point (138602.000000,1511131.000000) Niger
NREE on E side start 1518877.742858 and 1896496.738644 using slope 0.000004
Done.

Loading time varying boundary conditions: niger905.bdy
bdy Niger read.
bdy Dam read.
Done.

Perosity off
Weir off
Loading time varying evaporation: niger.evap
Done.

Starting time steps: acceleration mode
SGC mode
```

8. We can do a further check that the model is running by looking at the CPU load on the computer. Press Ctrl-Alt-Delete then select Start Task Manager and click on the Performance tab. The CPU usage will be 100% most of the time, indicating that the model is using all the computer's computational resources.
9. A folder called results will have appeared in the folder Inundation_model where the model will write its output
10. When the model has finished you will get the message displayed on the image below, and the command line will become active again

```
Finished.
Finish Date: 28/3/2013
Finish Time: 13:24:39

Total computation time: 39.33 mins

O:\Documents\Niger\Inundation_model>
```

The model should take under 10 minutes to run! While this is happening take a look at the hydrological inputs to the model and the model parameters described below.

Model input data and parameters

Let's take a closer look at the numerical model you have just set running. In the folder Inundation_model right click on the file *NigerLL.par* and select Edit with SciTE in New Tab or open in any other text editor.

This file contains the information necessary to run the simulation including file names and locations and the main model and run control parameters. The following items are listed in the file:

Item name	Description	Value in the Niger test case
DEMfile	Digital Elevation Model file name	NigerLLdem.asc
Resroot	Root for naming of results files (e.g. root.mass, root-0001-wd.asc etc)	res
Dirroot	Relative or absolute path for the directory where results files are to be placed. The directory is created if it doesn't exist already. If this keyword is omitted the results files are placed in the directory in which the model was executed	Results
Simtime	Total length of the simulation in seconds (real value)	220838400.0 (7 years)
initial_tstep	<i>Fixed time step model</i> Model time step in seconds (real value) <i>Acceleration and Adaptive time step model</i> Initial guess for the optimum time step and maximum possible time step.	500.0
Massint	Interval in seconds at which the .mass file is written to	86400.0 (1 day)
Saveint	Interval in seconds at which results files are saved. Note each file is saved with a sequential number stamp, eg results-0001-wd.asc	2628000 (1 month)
bdyfile	Name of file containing information on time varying channel inflow discharge for the Niger and Bani rivers	NigerLL.bdy
Bcifile	Name of file identifying the location of the Niger and Bani inflows	NigerLL.bci
Evaporation	Name of file containing evaporation data.	NigerLL.evap
SGCwidth	Raster data set of channel widths	NigerLL.width.asc
SGCbank	Raster data set of channel bank heights	NigerLL.banks.asc
SGCchangroup	Raster data set that in combination with SGCchanprams tells the model how deep the river channel are and what their frictions are	NigerLL.region.asc
SGCchanprams	See above	NigerLL.pram
fpfric	Manning's n value for floodplain	0.05
latlong	Tells the model to use a geographic coordinate system	N/A
SGCbhf_mode	Tells the model to use a particular mode	N/A

Now open the file *NigerLL.bdy* using the same method as the file *NigerLL.par*. This file is used to specify time varying inflow to the model for the river Niger and Bani. For each time varying boundary condition the format for the file is as follows:

Line 1: Comment line, ignored by LISFLOOD-FP.

Line 2: Boundary identifier (this will be Niger or Bani).

Line 3: Number of time points at which boundary information is given followed by a keyword for the time units used (either 'days', 'hours' or 'seconds').

Line 4: Value1 Time1

Line 5: Value2 Time2

etc....

Line i: Valuei Timei

Where Value is the discharge (m^3s^{-1}) and Time is the simulations time in sections. In this case time zero is the 1st of January 2002. With this file we can simulate any time-series of river discharge by changing the values in it. For example increasing the Values in the file by 10% will increase the discharge in the river by 10%. You can use excel to read and edit the data in these files and you can find the source data used to create this file in the Excel file *Niger_flow_data.xlsx* in the resources folder. For convenience I've provided you with two alternative .bdy files *NigerLL_150pc.bdy* and *NigerLL_50pc.bdy*... these will increase or decrease the discharge in the river by 50%, respectively, and can be used to simulate the effect of changing river discharge on flood inundation in the delta. To run these you will need to change the bdyfile in the *NigerLL.par* file to match the alternative bdyfile you want to run through the model. When doing this you might also want to change the name of the results folder after *dirroot* so that you do not overwrite you previous results.

Reading model results in QGIS

After the model has finished running! Model results files are all written in ASCII text format, so you can view the raw data in a text editor . However, to analyse the results it's more appropriate to visualise the data in a GIS. The file types produced by the model are described at the end of this handout. If the model has not finished running skip to the discussion section and come back to this when the model is done.

Here we are going to concentrate on one particular output, the maximum water depth simulated by the model. Use the ASCII to Raster converter tool in as earlier to read the results file *res_max.asc* which is in the results folder. Do you understand what the data shows? Discuss this with your group and the course demonstrators.

The model has also created a series of additional results that show water surface elevation *res_####_elev.asc* and the depth of water in cells *res_####_wd.asc* and depth of water on the floodplain *res_####_wdfp.asc*. These files are exported at regular monthly time interval with the numbers in the filename #### indicating the month of simulation, take a look at a few of these.

Discussion and some group assignment

For the assignment we are going to investigate how changing the river discharge in the model affects the flood inundation extent in the Inner Niger Delta. We have provided you with files to increase or decrease the discharge by 50% but you can choose to do something a bit different if you want, you might even want to add some realism to your changes in discharge by investigating the literature for potential futures.

Have a discussion in you groups about what you might want to investigate with the model, the article by Zwarts 2010 on the link below goes through some of the social and environmental pressures on the delta. Two or three scenarios (model runs) should be enough to complete the assignment, but you are welcome to do more.

<http://www.wetlands.org/LinkClick.aspx?fileticket=BNx9pX%2fpABM%3d&tabid=56>

Potential GIS operations

To support your project work you might want to use the GIS to calculate derived data sets such as:

1. Duration of inundation
2. Date of initial inundation
3. Volume of water of the floodplain
4. Flooded area over time
5. Average depth of inundation
6. Water surface elevations along the river (very difficult)

Some of these are relatively straightforward problems while others are trickier.

Suggested reading

It may be useful before or after the seminar to follow the hyperlinks and take a look at two accompanying papers:

Bates, P.D. and De Roo, A.P.J., (2000). A simple raster-based model for floodplain inundation. *Journal of Hydrology*, **236**, 54-77.

Neal, J., G. Schumann, and P. Bates (2012), A sub-grid channel model for simulating river hydraulics and floodplain inundation over large and data sparse areas, *Water Resour. Res.*, 48, W11506, (<http://dx.doi.org/10.1029/2012WR012514>)

There is also some online training material that goes into great detail about how the model works and demonstrates some potential flood risk applications:

<http://www.bristol.ac.uk/geography/research/hydrology/models/lisflood/training/>

Background information

The model, LISFLOOD-FP, takes a raster Digital Elevation Model and a file containing the location of the channel and produces a map of inundation extent from these.

The user specifies the parameters:

- Floodplain friction (Manning's n) in each floodplain grid cell,
- Channel friction (Manning's n) in each channel,
- Channel width and depth (in m) in each channel grid cell,
- Inflow rate (m^3s^{-1}) at the upstream boundary (either a constant rate or a hydrograph),
- Water surface slope (m) at the downstream boundary,
- Simulation time (in seconds),
- Initial time step (in seconds)

The model uses simple hydraulic principles to calculate the water depth in each cell of the raster grid. Specifically, we use an explicit finite difference solution of a simplified shallow water equation to route water in the channel. When water depths in the channel exceed the bank height, water is routed onto the adjacent floodplains using a simple storage cell algorithm based on the difference in hydraulic head (water surface elevation) between adjacent cells.

Thus in the channel the model calculates the flow between cells using:

$$Q_{c,i+1/2}^{t+\Delta t} = \frac{Q_{c,i+1/2}^t - gA^t\Delta t S_{i+1/2}^t}{1 + g\Delta t n^2 |Q_{c,i+1/2}^t| / [(R^t)^{4/3} A^t]} \quad (1)$$

Where: Q_c is the volumetric flow rate in the channel, A the cross sectional area of the flow, S the water surface slope, i is a cell index, g is gravity, Δt is the model time-step and n Manning's coefficient of friction. The hydraulic radius R is found by $R = A/P$, where P the wetted perimeter of the flow.

Floodplain volumetric flow rates are similarly described with the flow of water between cells Q_f calculated using:

$$Q_{f,i+1/2}^{t+\Delta t} = \frac{q_{f,i+1/2}^t - gh_{f,flow}^t \Delta t S_{i+1/2}^t}{1 + g\Delta t n^2 |q_{f,flow}^t| [(h_{f,flow}^t)^{7/3}]} (\Delta x - w_c) \quad (2)$$

Where Δx is the cell width, q is the flow Q divided by Δx , w is the width of any channels in the cell, and n is the Manning's friction coefficient for the floodplain. The flow depth, h_{flow} , represents the depth through which water can flow between two cells, and is defined as the difference between the highest water free surface in the two cells and the highest bed elevation.

Given these floodplain and channel flows the model updates the volume of water in each cell at each time step. The volume of water in each cell is calculated by:

$$V_{i,j}^{t+\Delta t} = V_{i,j}^t + \Delta t (Q_{i-1/2,j}^{t+\Delta t} - Q_{i+1/2,j}^{t+\Delta t} + Q_{i,j-1/2}^{t+\Delta t} - Q_{i,j+1/2}^{t+\Delta t}) \quad (3)$$

Where V is the volume of water in the cell from which we can easily calculate the water surface elevation h , $Q_{i,j}^{t+\Delta t}$ is the flow into or out of the cell from the floodplain and any channels at each cell interface, See Figure 1.

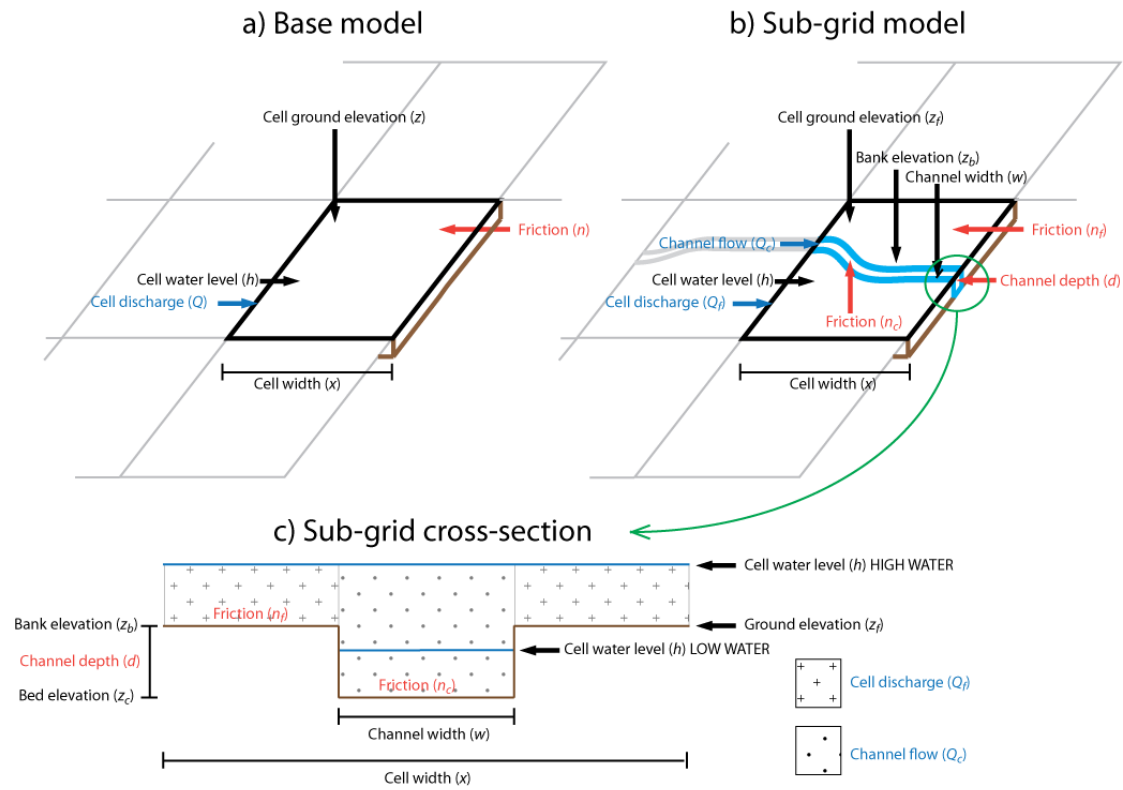


Figure 1: Model schematics for a. Floodplain only cell. b. Floodplain and channel cell c. interface between cells

To accurately simulate the hydrodynamics the model must run at a short enough time-step to prevent oscillations developing in the water surface. This time-step is defined by the well-known Courant-Friedrichs-Lewy condition:

$$\Delta t = \alpha \frac{\Delta x}{\sqrt{gh}} \quad (3)$$

Model input and output file formats (technical information)

Input file formats

Data is input to the model using the file types as described below. Users should note that the file extensions are not fixed, comments can only be used in the parameter file (`.par`) and all items are case sensitive.

Boundary condition type file (`.bci`)

This file specifies boundary conditions not associated with the channel. There can be any number of boundaries on the edge of the domain or at points within the domain itself.

Column 1: Boundary identifier taking a value of N, E, S, W or P and referring to the north, east, south or west boundaries or P referring to a point source

Column 2: start of boundary segment (easting or northing in map co-ordinates) for edge boundaries or easting in map co-ordinates for a point source location

Column 3: End of boundary segment (easting or northing in map co-ordinates) for edge boundaries or northing in map co-ordinates for a point source location

Column 4: Boundary condition type

Column 5: Boundary condition value. This varies according to boundary condition type as indicated in **Error! Reference source not found.**

Possible boundary condition types and their associated values are given below.

Boundary condition type	Description	Value supplied in column 5 of the <code>.bci</code> file
CLOSED	Zero-flux (default option)	None
FREE	Uniform flow	Free surface or valley slope (optional)
QVAR	Time varying flow into domain	Boundary identifier (e.g. <code>Niger</code>) corresponding to data in the user supplied <code>.bdy</code> file

Time varying boundary conditions file (`.bdy`)

This file is used to specify time varying boundary conditions (keywords `QVAR` in the `.bci` file) associated with a point source. For each time varying boundary condition the format for the file is as follows:

Line 1: Comment line, ignored by LISFLOOD-FP.

Line 2: Boundary identifier

Line 3: Number of time points at which boundary information is given followed by a keyword for the time units used (either 'days', 'hours' or 'seconds').

Line 4: Value₁ Time₁

Line 5: Value₂ Time₂

etc.... ...

Line i: Value_i Time_i

Where Value_i is given as mass flux per unit width and has units of m²s⁻¹. In this case, the flux per unit width is multiplied within the code by the cell size to give the mass flux (discharge) in m³s⁻¹.

Digital Elevation Model file (`.asc`)

This file specifies the Digital Elevation Model used by the model. It consists of a 2D raster array of ground elevations in ARC ascii raster format. The file may be manipulated using either the ArcGIS Geographical Information System platforms or manually edited using a text editor. A brief summary of the format is provided below.

The file consists of a 6 line header followed by the numerical values of each data point on the grid as a 2D array of *i* rows and *j* columns. Each line of the header consists of a self-explanatory keyword

followed by a numeric value. As an example, the header for the Niger application is given below (comments in brackets are not part of the file format):

ncols	603	(Number of columns)
nrows	426	(Number of rows)
xllcorner	150686.107468	(X cartesian co-ordinate of the lower left corner of the grid in metres)
yllcorner	1510877.742858	(Y cartesian co-ordinate of the lower left corner of the grid in metres)
cellsize	905.209361	(Cell size in metres)
NODATA_value	-9999	(Null value)

Evaporation data file (.evap)

This file is used to specify a time-varying evaporation rate and is read when the keyword `evaporation` appears in the `.par` file. This sink term is then applied to every model grid cell at each time step to give a spatially uniform evaporation loss over the domain. The file format is similar to the `.bdy` file:

Line 1: Comment line, ignored by LISFLOOD-FP.

Line 2: Number of time points at which boundary information is given followed by a keyword for the time units used (either 'days', 'hours' or 'seconds').

Line 3: Value₁ Time₁

Line 4: Value₂ Time₂

etc....

Line i: Value_i Time_i

Where Value_i is evaporation rate in mm day⁻¹ and Time_i is the time at which this value occurs in the units specified on line 2. The model then linearly interpolates these values to give the evaporation rate at each time step.

Output file formats

During a simulation the model produces a series of results files named according to the `resroot` convention given in the parameter file. These are placed in the `dirroot` directory if this keyword and a directory name are placed in the parameter file. The output files are produced at different time intervals according to specifications made by the user in the parameter file and are described below.

Mass balance output file (.mass)

This file gives details of the model mass balance performance and is written at the interval specified by the keyword `massint` in the parameter file. The output consists of 11 columns of data, space separated:

Column 1: Time. The time in seconds at which the data was saved.

Column 2: Tstep. Time step specified by the user (initial time step in the adaptive model)

Column 3: MinTstep. Minimum time step calculated by the adaptive model during the iteration

Column 4: NumTsteps. Number of time steps since the start of the simulation.

Column 5: Area. Area inundated in km².

Column 6: Vol. Volume of water in the domain.

Column 7: Qin. Inflow discharge in m³s⁻¹.

Column 8: Hds. Water depth at the downstream exit of the model domain.

Column 9: Qout. Calculated outflow discharge at the downstream exit of the model domain in m³s⁻¹.

Column 10: Qerror. Volume error per second in m³s⁻¹.

Column 11: Verror. Volume error per mass interval (`massint` variable in the parameter file) m³.

Column 11: Inf+Evap. Cumulative Infiltration loss over the simulation in 10³ m³.

Synoptic water depth, water surface elevation files (`-xxxx.wd.asc`, `-xxxx.elev.asc` and `-xxxx.wdfp.asc`)

These files consist of a grid of water depths and water surface elevations values in ARC ascii raster format for each pixel at each save interval (saveint) specified in the parameter file. Units are in metres. In this naming convention xxxx is the saveint number. `-xxxx.wdfp` represent floodplain only water depths (i.e. in cells containing a channel this is the depth of water above bankfull depth). By default these output options are turned on but production of each set of files can be suppressed by putting the logical keywords `depthoff` or `elevoff` in the `.par` file.

Maximum water surface elevation file (`.mxe`) and maximum water depth (`.max.asc`)

These files consist of a grid in ARC ascii raster format of the maximum water surface elevation (`.mxe`) predicted by the model for each pixel over the course of the simulation, or the maximum water depth (`.max`). Units are in metres. By default these values are the maximum values over the whole simulation (i.e. over each time step) but if the keyword `mint_hk` appears in the `.par` file then they are the values over each time step for which the `.mass` file is written to (`massint`) instead. Calculating the maximum at the mass interval rather than at every time-step will be computationally more efficient but less accurate (especially if water depths are changing rapidly relative to `massint`). There is currently no keyword to suppress the output of these files.