

# User Manual for Distributed Flow Routing Model: 1K-FRM-event

Version 1.34, February 12, 2013

Yasuto TACHIKAWA and Tomohiro TANAKA

*Dept. of Civil and Earth Resources Eng., Kyoto University, Kyoto 615-8540, Japan*  
[tachikawa@hywr.kuciv.kyoto-u.ac.jp](mailto:tachikawa@hywr.kuciv.kyoto-u.ac.jp)

## 1. Introduction

1K-FRM-event is a distributed flow routing model based on kinematic wave theory. The source programs are coded using C++ language and works under Microsoft windows environment and the Linux environment. All program source codes and input topography data in the Asian region are provided from the home page of Hydrology and Water Resources Research Laboratory at Department of Civil and Earth Resources Engineering, Kyoto University:

<http://hywr.kuciv.kyoto-u.ac.jp/products/1K-DHM/1K-DHM.html>

The input topography data is generated using HydroSHED 30 second digital elevation model and flow direction data. Forcing data to drive 1K-FRM-event such as runoff generation or precipitation is supposed to be hourly gridded data. At first, a method to develop topography data for 1K-FRM-event is described. Then, by using the developed data, an example of rainfall-runoff simulation using 1K-FRM-event is demonstrated. Figure 1 shows entire structure of the coordinate system used for 1K-FRM-event flow routing simulation.

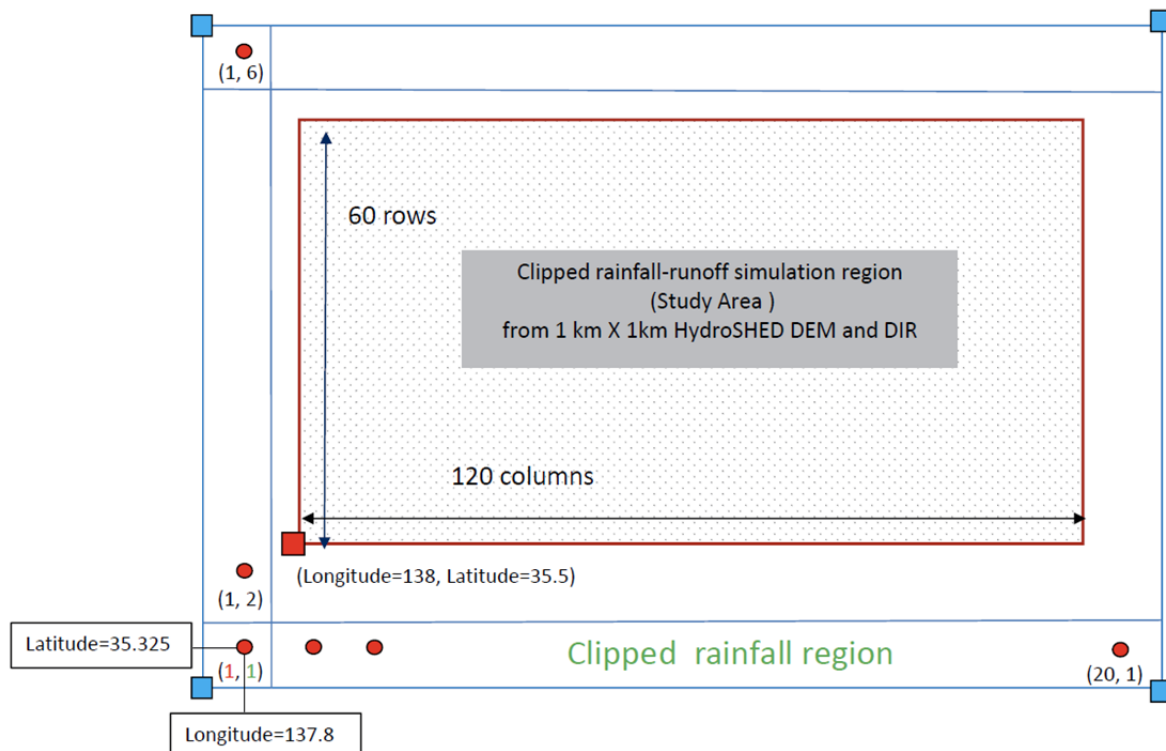


Figure 1: Geographic relation of flow routing simulation area and forcing data grid region.

## 2. Preparation of Topographic Data for 1K-FRM-event

### 1) Hydrosched2topo

Hydrosched2topo is a program to generate input topography data for 1K-FRM-event, which is obtained from 1K-FRM homepage.

### 2) Input data

The topographic data for a flow routing model 1K-FRM-event is generated by using the HydroSHED (<http://hydrosheds.cr.usgs.gov/>) 30-second digital elevation and flow direction data. The DEM and flow direction data with ESRI grid format data for entire Asia (55E,-10S to 180E, 60N) from HydroSHED were downloaded and they were converted to short integer 2-bite binary data format to reduce the file size. These two files:

as\_dem\_30s\_clipped\_55—10-180-60.bin (246MB)  
as\_dir\_30s\_clipped\_55—10-180-60.bin (246MB)

are obtained from 1K-FRM homepage. These two data are input files for hydrosched2topo.

### 3) Parameter data

All Necessary topographic input files to run the 1K-FRM-event are created by executing the hydrosched2topo.cc. The program creates additional ESRI ASCII files and those files can be imported to ArcGIS to prepare maps of the study area. First, you need to specify the study area by editing the parameter data file

hydrosched2topo/clipArea.dat

This file is a text data file. The study area is determined as a rectangular region by setting longitude and latitude as shown in Figure 2.

- The first row specifies the number of columns for the study region. The value must be an integer value.
- The second row specifies the number of rows for the study region. The value must be an integer value.
- The third row specifies Longitude (degree) of South West (lower left) corner of the study area. The value is a real value.
- The forth row specifies Latitude (degree) of South West (lower left) corner of the study area. The value is a real value.

The grid size is supposed to 30 second. The longitude (second) of j-th cell center is set as follows:

$$\text{longitude} = (\text{int})(\text{SWlon} * 3600) + 30 * j + 15;$$

$$\text{latitude} = (\text{int})(\text{SWlat} * 3600) + 30 * i + 15;$$

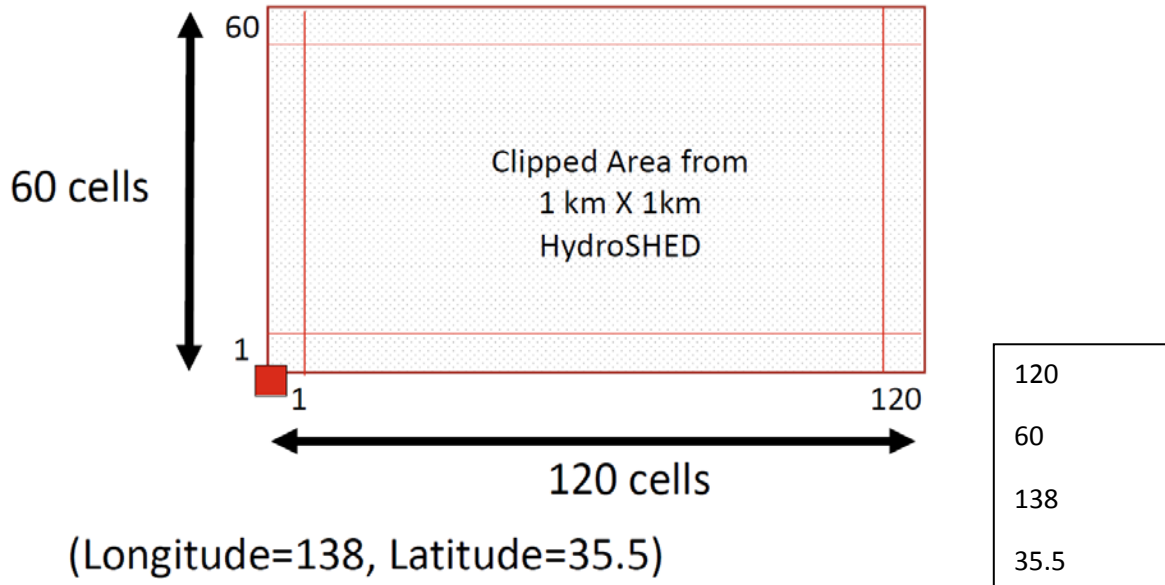


Figure 2: Setting study area. An example of clipArea.dat.

4) Generation of topographic information for 1K-FRM-event

Compile hydroshed2topo.cc and run hydroshed2topo.exe.

5) Output files

All output files used for the 1K-FRM-event are created in the folder:

hydroshed2topo/output

The files generated are below:

- **modDem.bin**: This is the corrected (pit removed) elevation information of the study area. The data was written in real 4-byte binary data format.

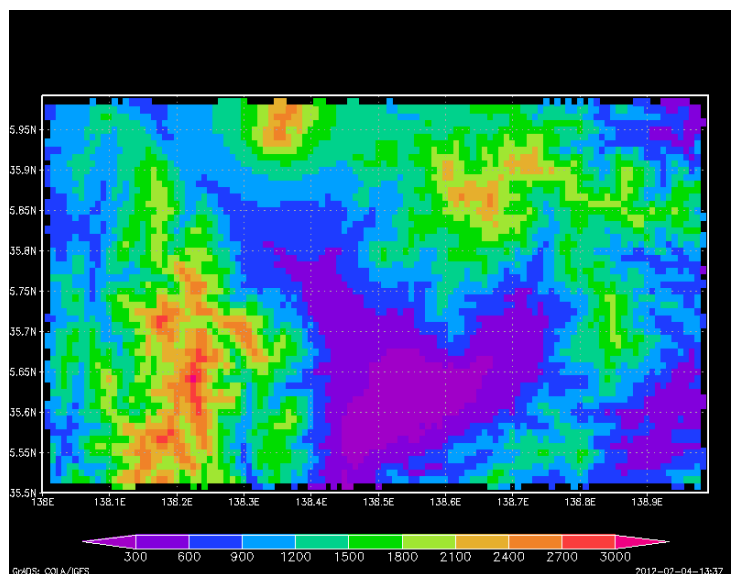


Figure 3: A sample of moddem.bin.

- **flowDir.bin**: Flow direction information. The direction convention is shown in Figure 3. The data format is integer 4-bite binary.

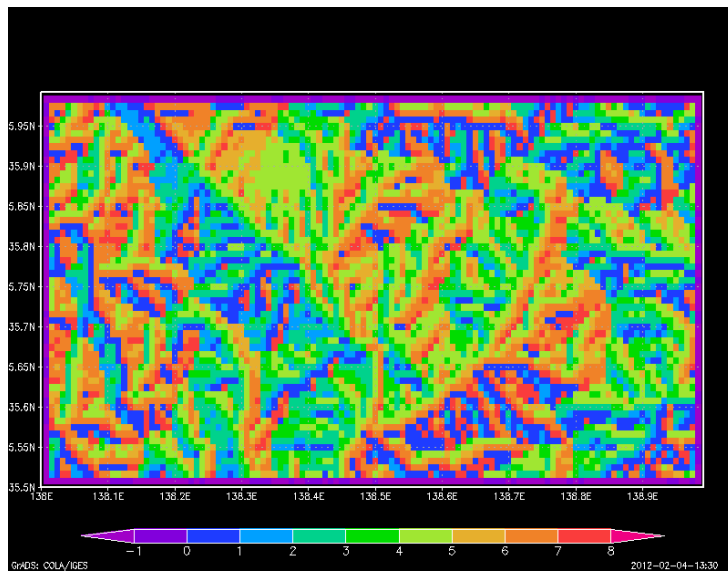


Figure 4: A sample of flowDir.bin.

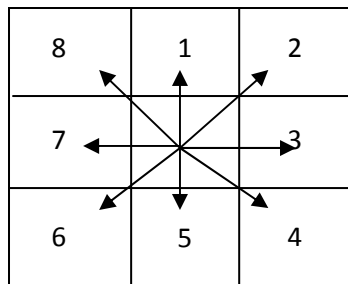


Figure 5: Flow direction specified by the integer value.

- **flowAcc.bin**: Flow accumulation information. The data format is integer 4-bite binary.

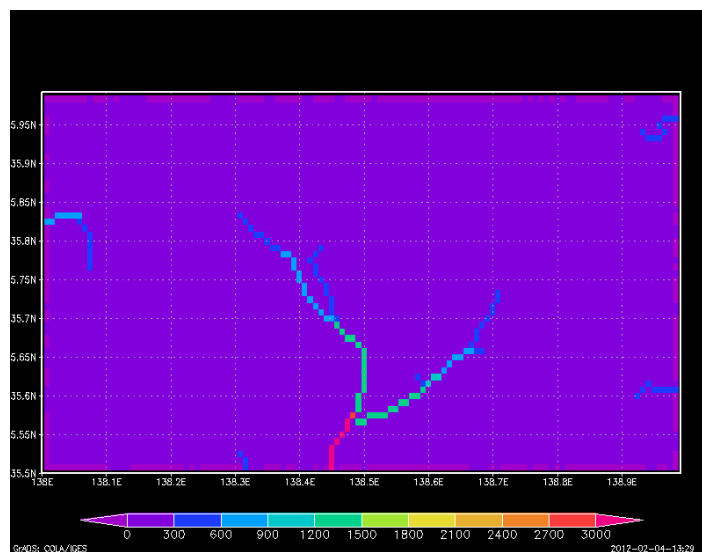


Figure 6: A sample of flowAcc.bin.

- **riverNum.bin**: Each river basin is numbered starting from 1 and then each grid cell is assign a value corresponding to the basin number which the cell is belongs to. The data format is integer 4-bite binary.

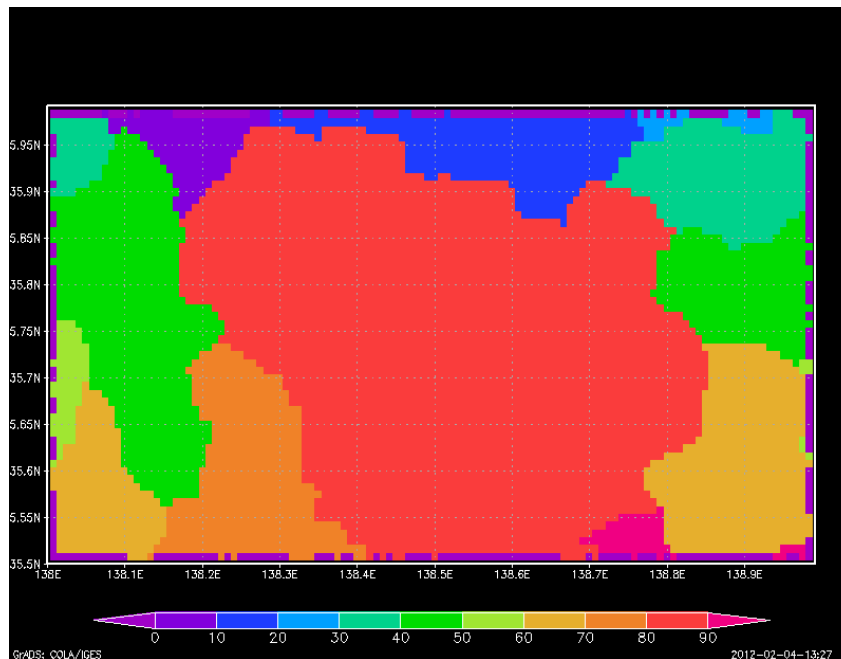


Figure 7: A sample of riverNum.bin.

- **riverNumList.txt**: This is a text file which contains:

Column 1: river basin number of the study area,  
 Column 2: the basin area (acmulated number of grids).  
 Column 3: column number starting from 1 (starting from West),  
 Column 4: row number starting from 1 (starting from South),  
 Column 5: longitude (second),  
 Column 6: latitude (second),  
 Column 7: longitude (degree), and  
 Column 8: latitude (degree)

If you want to calculate the discharge data only at some selected basins using 1K-FRM-event or 1K-FRM, you can delete all the other basins from this text file and keep only the necessary basins.

All data begins from the South-West grid and lining to East, then, moving to the North in the next line, which are typical GrADS data format. Sample GrADS control files are attached in hydroshed2topo to visualize the data.

```

1 5 10 59 497085 129555 138.079 35.9875

2 3 14 59 497205 129555 138.113 35.9875

3 4 16 59 497265 129555 138.129 35.9875

4 4 17 59 497295 129555 138.137 35.9875

5 4 18 59 497325 129555 138.146 35.9875

...

```

Figure 8: An example of riverNumList.txt.

Additional files are also created, which are ESRI ASCII files to be imported to ArcGIS to visualize the study region:

- **cutDem.asc**: The DEM data for the study region. This is the original data clipped for the study region.
- **cutDir.asc**: The DIR data for the study region. This is the original data clipped for the study region.
- **modDem.asc**: The DEM data for the study region. This is the corrected (pit removed) elevation information of the study area, which corresponds to **moDem.bin**.
- **flowDir.asc**: The Flow Direction data for the study region, which corresponds to **flowDir.bin**.
- **flowAcc.asc**: Flow accumulation value information, which corresponds to **flowAcc.bin**.
- **riverNum.asc**: The river basin numbers of the study area, which corresponds to **riverNum.bin**.
- **distance.asc**: The distance from the river mouth.

### 3. Preparation of Forcing Data

It is necessary to create a text file of effective hourly forcing data (spatially distributed lateral input data to slope and river such as rainfall intensity or runoff generation intensity data). If the data duration is shorter than simulation time, remaining rainfall is assumed to zero. A part of a sample data file is shown in Figure 9. The rainfall data file name is “rain.dat” and should be placed at

1k-frm-event/input/rain.dat

The first line of “rain.dat” is Year, Month, Day, Hour, Column Number, and Row Number of spatially gridded rainfall data. From the next line, rainfall intensity values (mm/hr) are stored beginning from the North-West grid and lining to East, then, moving to the South in the next line. Each data is delimited by white space characters (SPACE, TAB, or CR). You can change the rainfall intensity data for your study purpose.

1979	12	1	1	5	6
0.1	0.2	0.3	0.4	0.5	
0.6	0.7	0.8	0.9	1.0	
1.1	1.2	1.3	1.4	1.5	
1.6	1.7	1.8	1.9	2.0	
2.1	2.2	2.3	2.4	2.5	
2.6	2.7	2.8	2.9	3.0	
...					

Figure 9: Part of rain.dat. The 1<sup>st</sup> Row is: Year, Month, Day, Hour, Column Number, Row Number. From the 2<sup>nd</sup> to 7<sup>th</sup> Rows, effective rainfall intensities (5 columns and 6 rows) in mm/hr are stored.

The geographic structure is shown in Figure 10. The rainfall intensity data represents the intensity of each grid center. The time interval of the rainfall intensity must be hour. Figure 11 shows the relation between the rainfall-runoff simulation region and rainfall grid region. For runoff simulation, the nearest grid of rainfall intensity is selected.

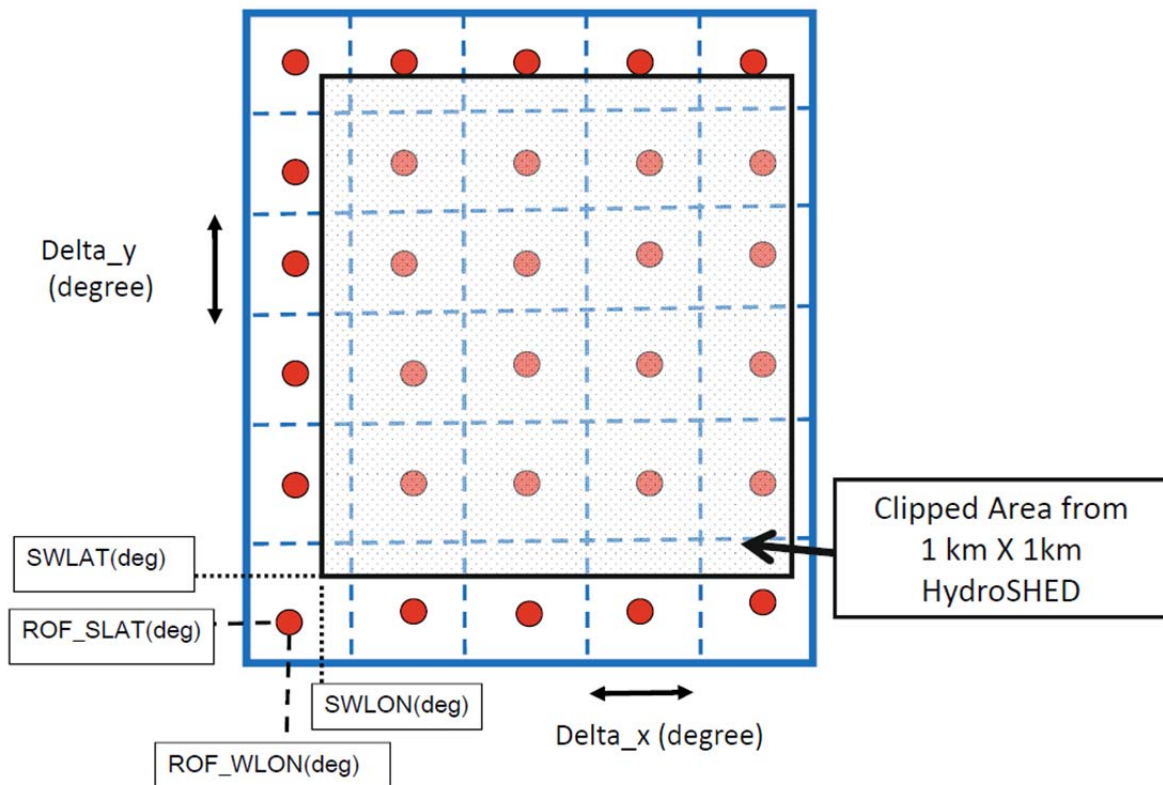


Figure 10: Structure of coordinate of the rainfall grid system.

The grid size of forcing data is set in the parameter setting file for 1K-FRM-event. The grid center of longitude (degree) and latitude (degree) of j-th column and i-th row for input data are set as follows:

longitude = ROF\_WLON + j\*Delta\_x; // set longitude

latitude = ROF\_SLAT + j\*Delta\_y; // set latitude

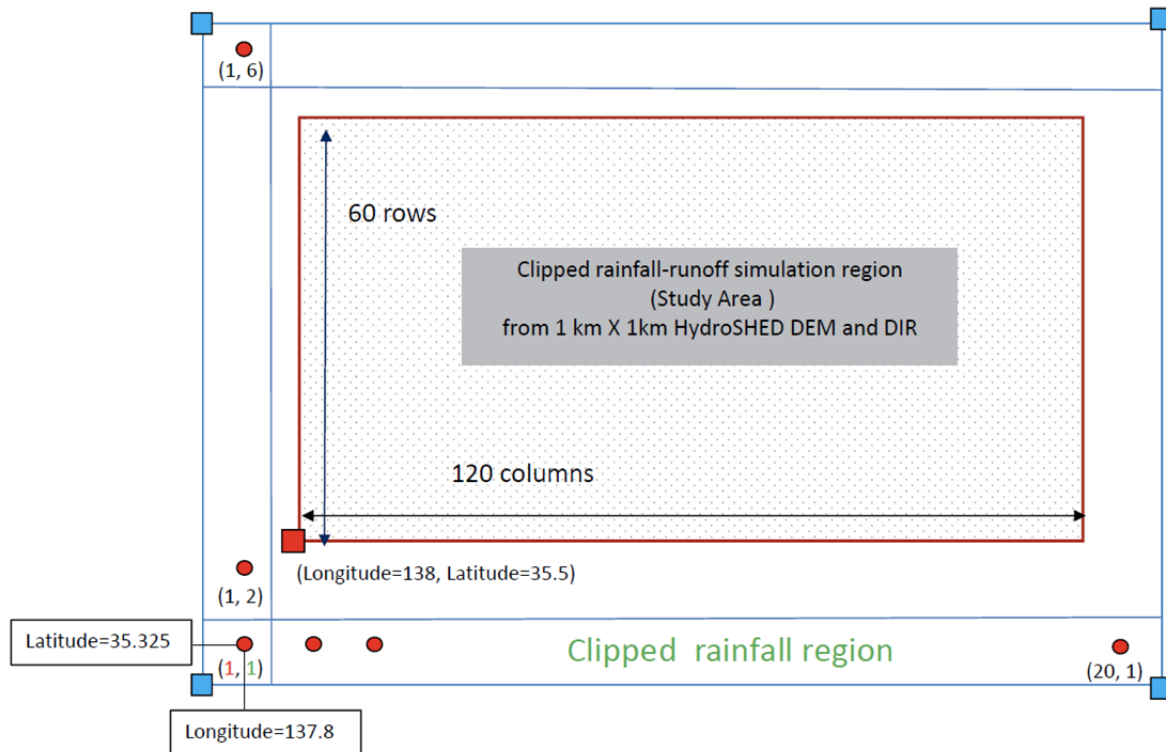


Figure 11: Geographic relation of rainfall-runoff simulation area and rainfall grid region.

#### 4. Flow Routing Simulation Using 1K-FRM-event

Figure 11 shows the geographic relation of the rainfall-runoff simulation area and the rainfall grid region for the example data. The procedures for runoff simulation are as follows:

- 1) **Topography data setting:** Copy the topographic information files (above mentioned five files)

from hydroshed2topo/output/ to 1k-frm-event/topoData/

- 2) **Rainfall data setting:** Set the rain data and place rainfall data file at

1k-frm-event/input/rain.dat. (The file name must be rain.dat.)



3) **Runoff simulation parameter setting:** Edit the runoff simulation parameter file

1k-frm-event/param-event.dat

by using any text editor. Delimiter of each data is white space characters.

120	Number of columns of the study area
60	Number of rows of the study area
138.0	Longitude (degree) of South West corner of the study area
35.5	Latitude (degree) of South West corner of the study area.
5	Number of columns of the input forcing data
6	Number of rows in the input forcing data
0.1875	Input data grid size of x-direction (degree)
0.1875	Input data grid size of y-direction (degree)
137.75	West corner longitude of input data (degree)
35.25	South corner latitude of input data (degree)
10	Simulation duration (days)
600	simulation time step (sec)
20	number of spatial divisions
0.65	$0.5 < \theta \leq 1.0$
0.5	Manning's roughness coefficient for slope (m-s unit)
0.03	Manning's roughness coefficient for river (m-s unit)
250	grid accumulated number to differentiate slope and river
0.0	Initial runoff height for kinematic wave model (mm/hr)

Figure 12: An example of param-event.dat.

The parameter values to set in param-event.dat are as follows:

- From 1<sup>st</sup> row to 4<sup>th</sup> row, the parameters of the study region is specified:

```
int Col=120           // number of column for study area.
int Row=60           // number of row for study area.
double SWLON=138.0   // Longitude (degree) of north west corner of the study area.
double SWLAT=35.5    // Latitude (degree) of north west corner of the study area.
```

- From 5<sup>th</sup> row to 10<sup>th</sup> row, the parameters of the rainfall data is specified:

```
int ROFnum_col = 5    // Number of column of input rainfall data
int ROFnum_row = 6    // Number of row of input rainfall data
double Delta_x = 0.1875 // Input data grid size of x-direction (degree)
double Delta_y = 0.1875 // Input data grid size of y-direction (degree)
```

```
double ROF_WLON = 137.75 // West corner longitude of input data (degree)
double ROF_SLAT = 35.25 // South corner latitude of input data (degree)
```

- The 11<sup>th</sup> row is a parameter to determine the simulation term.

```
int Ndays = 10 // Simulation duration (days)
```

- From 12<sup>th</sup> row to 14<sup>th</sup> row, the parameters for the finite difference condition are specified. Usually, you do not need to change these parameter values.

```
int Time_Step = 600 // Simulation time step (sec)
int Nspace = 20 // Number of spatial divisions
double Theta = 0.65 // weight value 0.5 < theta <= 1.0
```

- From 15<sup>th</sup> row to 17<sup>th</sup> row, the parameters for the kinematic wave model are specified.

```
double Man_slope = 0.5 // Manning's roughness coefficient for slope (m-s unit)
double Man_channel = 0.03 // Manning's roughness coefficient for river (m-s unit)
int Threshold_basin = 250 // grid accumulated number to differentiate slope and river
```

- The 16<sup>th</sup> row is a parameter to determine the initial condition.

```
double initialROF = 0.0 // Initial runoff height (mm/hr)
```

#### 4) Rainfall-runoff simulation

Move to the folder “/project/1k-frm-event” and run

```
“1k-frm-event.exe”
```

All runoff simulation results are created in

```
1k-frm-event/output
```

The files created are below:

- **dischargeDaily.bin:** The file contains the average daily flow (m<sup>3</sup>/sec).
- **dischargeHourly.bin:** The file contains the hourly flow (m<sup>3</sup>/sec).
- **dischargeHourlyMaxInDay.bin:** The file contains the maximum hourly flow value for each day (m<sup>3</sup>/sec).

The data are stored in GRADS binary data format and can be visualized using GRADS (<http://www.iges.org/grads/>). All data arranges from west to east at the South row, then North row. GrADS control file:

```
- flowAnalysisGradsDaily.ctl
- flowAnalysisGradsHourly.ctl
```

are attached. A sample program is also attached in ./output:

- pointdata-event.cc

which extracts time series ascii data from the above binary data.

## 5) Compile

If you want to change the source code, compile a modified code using:

- Linux environment: The executable is created by using the GNU g++ compiler as following command

```
% g++ -o 1k-frm-event 1k-frm-event.cc cellfunc-event.cc roffunc-event.cc
```

For Intel C++ compiler of Linux version

```
% icc -o 1k-frm-event 1k-frm-event.cc cellfunc-event.cc roffunc-event.cc
```

- MS Windows environment: Microsoft Visual Studio 2008 or 2010 with C++ compiler is useful. You can download a free version of Visual C++ Express Edition 2008 or 2010 from Microsoft web site.

The solution file *1k-frm-event.sln* should be created according to the Microsoft Visual Studio to use Visual Studio environment. You can also use the command prompt window of the Microsoft Visual Studio. In this case, after opening the command prompt window, you can compile the program as:

```
% cl 1k-frm-event.cc cellfunc-event.cc roffunc-event.cc
```

For Intel C++ compiler of windows version

```
c:\> icl 1k-frm-event.cc cellfunc-event.cc roffunc-event.cc
```

## 5. Output Data Analysis

### 5.1 Visualization of time and space change of discharge using GRADS

GRADS is a free visualization software mainly used by meteorologists. The software works on Windows and Linux/Unix environment. GRADS is available from <http://www.iges.org/grads/>.

You can use an example of GRADS control files “flowAnalysisGradsDayly.ctl” and “flowAnalysisGradsHourly.ctl” in “/project/misc/gradsControlFiles”. Please see the detail information at the web page. Figure 13 is a spatial distribution of discharge at the study region.

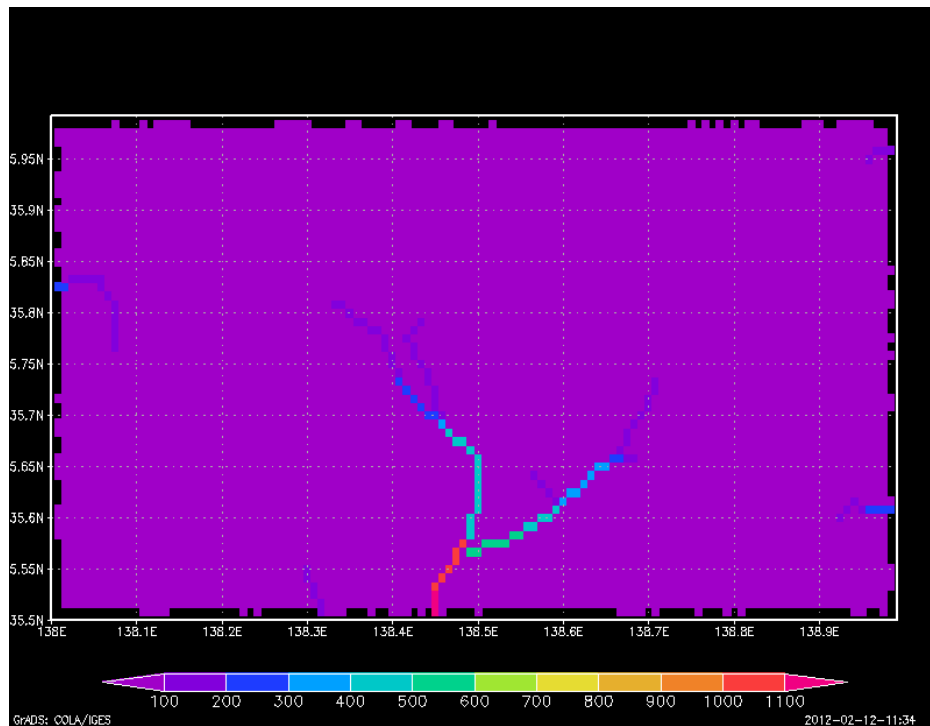


Figure 13: Spatial view of simulated discharge using GrADS.

## 5.2 Visualization of time and space change of discharge using makeImage.exe and IrfanView

If you do not have GrADS, you can generate PPM format image files and see it using a free image viewer IrfanView. The procedure to see the animation of time and space change of discharge is as follows:

- (1) Run the image generation program makeppm.exe, which is downloaded from 1K-FRM home page. This program reads the GrADS format output hourly discharge data **dischargeHourly.bin** and generate PPM format image files under image folder. To run the program,

- 1-1) Open the folder “/project/1k-frm-event”
- 1-2) Click “makeppm.exe”
- 1-3) Image files are generated under “/project/1k-frm-event/image”.

- (2) Install IrfanView. This is a very popular free image viewer. Click

misc/irfanview/irview428\_setup.exe

The viewer is installed and you will find the icon on your MS-Windows desktop.

- (3) Click the IrfanView icon and select [File]→[Slideshow]. Then, set the location of PPT image file. You can enjoy the animation.

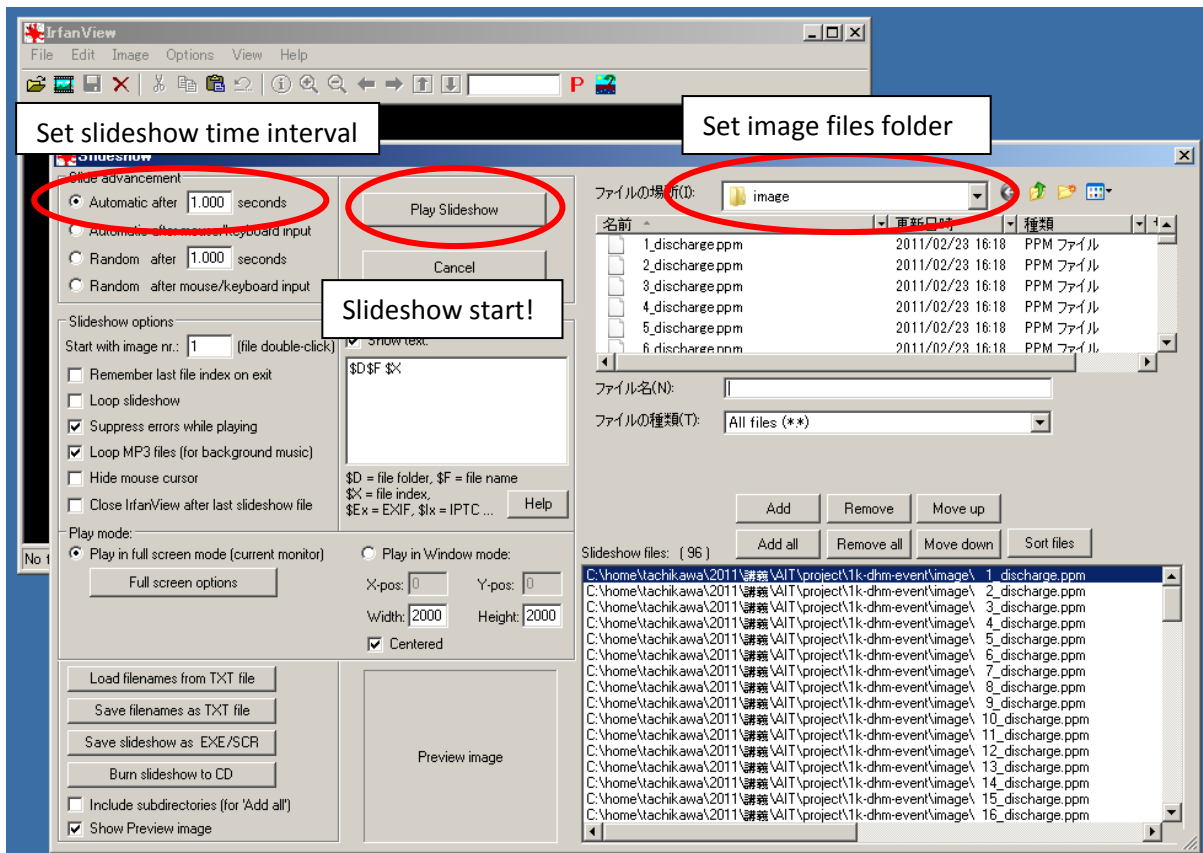


Figure 14: IrfanView Slideshow setting.

- (4) Another method to view the animation is to select [File]→[Open], select the image folder and click the first image. Then, select [View]→[Start/Stop automatic viewing]. You can select [View]→[Display option (window mode)] to enlarge the image.

### 5.3 Time series of discharge data at specific point

The procedures to extract the time series of discharge data at the specific point is as follows:

- (1) Run the time series extraction program pointdata.exe. This program reads the GRADS format output hourly discharge data **dischargeHourly.bin** and generates text format time series data under /output. To run the program,
  - 1-1) Open the folder “1k-frm-event”.
  - 1-2) Click “pointdata.exe”.
  - 1-3) You will be asked to input the column and row number to extract the data. To determine the column and row number, it is useful to see the accumulated catchment data “/project/hydroshed2topo/output/ESRIBasin.asc to examine the

discharge with large catchment size. The row and column number begins from the North West corner starting from 1.

- 1-4) The extracted data is generated under \1k-frm-event\output\Col-Row-discharge.dat. The first column of the data is time count (hour) and the second column is discharge ( $\text{m}^3/\text{sec}$ ).

1	0.0882515
2	0.653777
3	2.2653
4	7.35936
5	14.4867
6	20.414
7	25.6387
8	32.4114
9	43.8638
10	55.3801
...	

Figure 15: An example of time series discharge data extracted by using pointdata-event.exe.

- (2) To draw the hydrograph, you can use Excel or GnuPlot (<http://www.gnuplot.info/>).