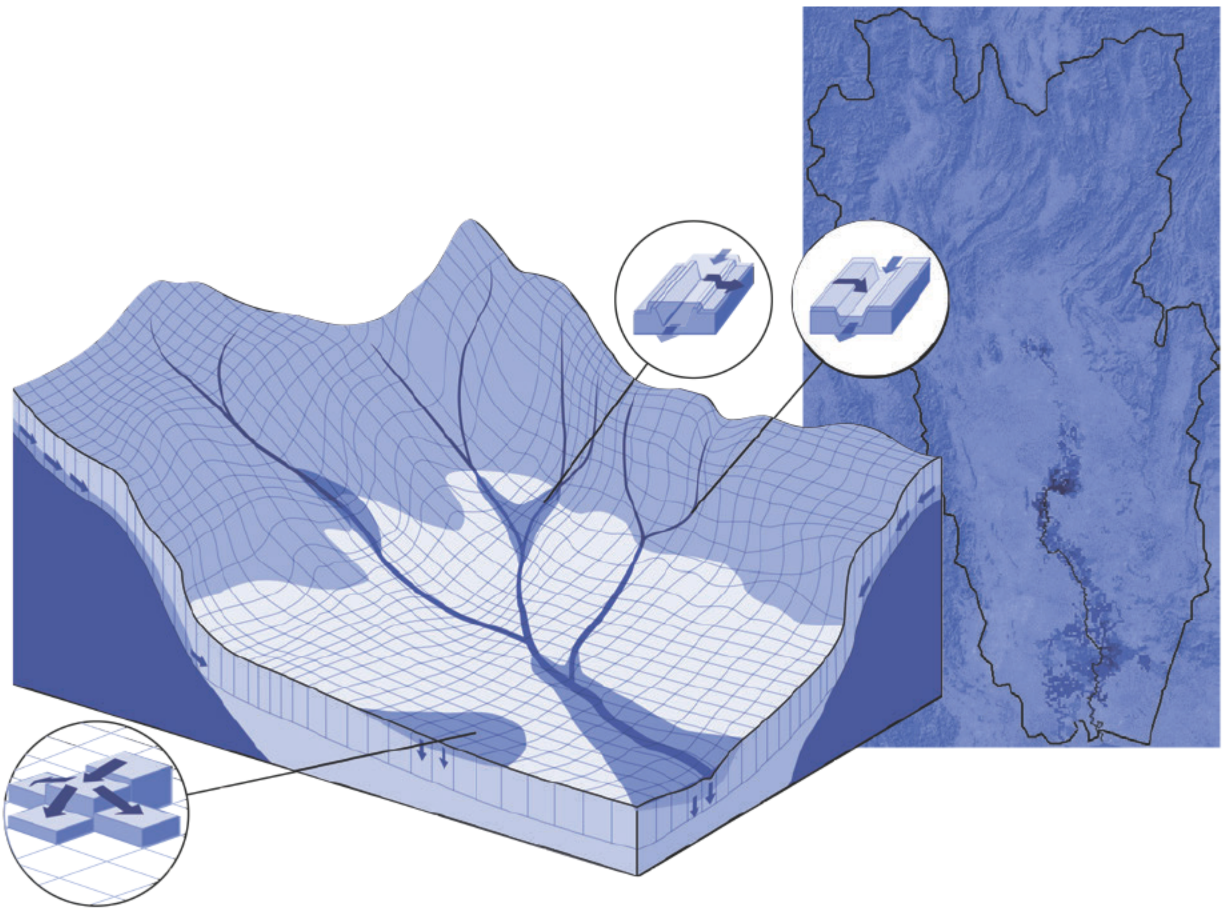


ISSN 0386－5878
Technical Note of PWRI No.4277

ICHARM Publication No.27

Rainfall-Runoff-Inundation (RRI) Model Technical Manual

February 2014



ICHARM

PWRI

UNESCO
United Nations Educational, Scientific and Cultural Organization

International Centre for Water Hazard and Risk Management under the auspices of UNESCO (ICHARM)
Public Works Research Institute(PWRI)

ISSN 0386－5878
Technical Note of PWRI No.4277

Rainfall-Runoff-Inundation (RRI) Model Technical Manual

February 2014

International Center for Water Hazard and Risk Management (ICHARM)

INCORPORATED ADMINISTRATIVE AGENCY
PUBLIC WORKS RESEARCH INSTITUTE

1-6, Minamihara, Tsukuba-shi, Ibaraki-ken, 305-8516 JAPAN

Technical Note of
PWRI No.4277

Rainfall-Runoff-Inundation (RRI) Model
Technical Manual

by

Takahiro Sayama

Synopsis：

Rainfall-Runoff-Inundation (RRI) Model simulates various hydrologic processes including rainfall-runoff, stream flow propagation and flood inundation in an integrated manner at a river basin scale. The model solves diffusive wave equations applied to flows on ground surface and subsurface on the two dimensional basis. The model is designed to be applicable even to large river basins with limited data by utilizing satellite-based information. The model application includes flood prediction and risk assessment, in particular at a basin with large floodplains. This technical manual describes the basic concept with equations, step-by-step model application and the advance use of the RRI model.

Key Words：Flood Prediction, Risk Assessment, Distributed Model, GIS and Remote Sensing

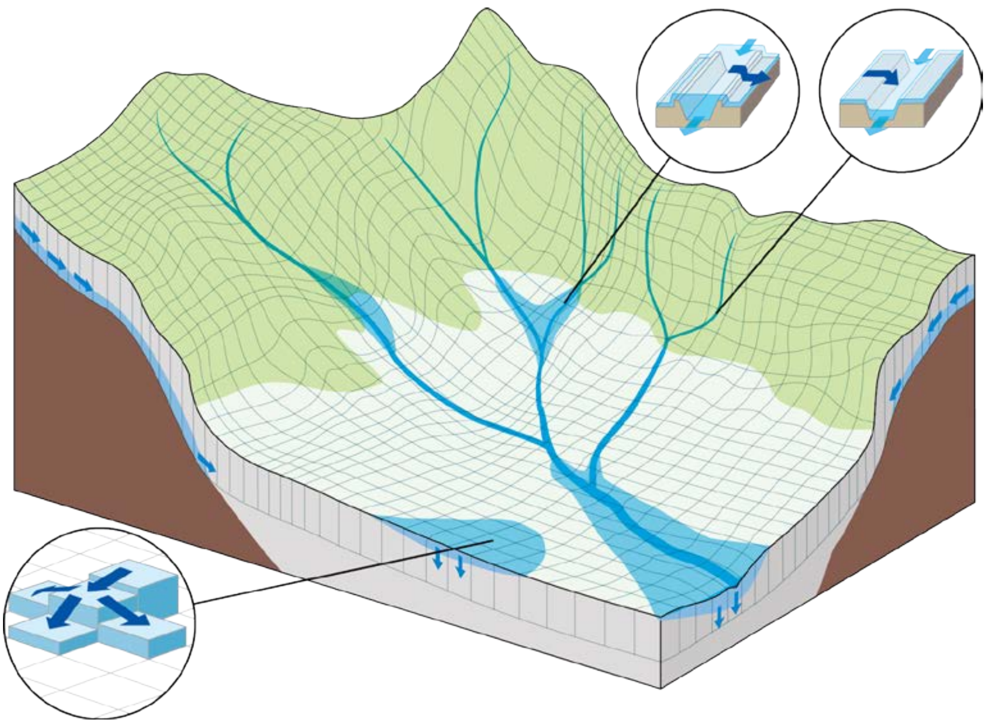
Rainfall-Runoff-Inundation Model Technical Manual

Table of Contents

1. Outline of Rainfall-Runoff-Inundation (RRI) Model

1.1 Model Structure Overview

Rainfall-Runoff-Inundation (RRI) model is a two-dimensional model capable of simulating rainfall-runoff and flood inundation simultaneously (Sayama et al., 2012). The model deals with slopes and river channels separately. At a grid cell in which a river channel is located, the model assumes that both slope and river are positioned within the same grid cell. The channel is discretized as a single line along its centerline of the overlying slope grid cell. The flow on the slope grid cells is calculated with the 2D diffusive wave model, while the channel flow is calculated with the 1D diffusive wave model. For better representations of rainfall-runoff-inundation processes, the RRI model simulates also lateral subsurface flow, vertical infiltration flow and surface flow. The lateral subsurface flow, which is typically more important in mountainous regions, is treated in terms of the discharge-hydraulic gradient relationship, which takes into account both saturated subsurface and surface flows. On the other hand, the vertical infiltration flow is estimated by using the Green-Ampt model. The flow interaction between the river channel and slope is estimated based on different overflowing formulae, depending on water-level and levee-height conditions.



Schematic diagram of Rainfall-Runoff-Inundation (RRI) Model

Model Features

1) Rainfall-runoff and inundation simultaneously with diffusion wave approximations.
2) Subsurface flow (lateral subsurface and vertical infiltration) is simulated for physical representations of rainfall-runoff processes.
3) One-dimensional diffusive wave river routing and its interaction with the slope model.

1.2 Governing Equations of RRI Model

A method to calculate lateral flows on slope grid-cells is characterized as "a storage cell-based inundation model" (e.g. Hunter et al. 2007). The model equations are derived based on the following mass balance equation (1) and momentum equation (2) for gradually varied unsteady flow.

$$\frac{\partial h}{\partial t}+\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}=r-f \quad (1)$$

$$\frac{\partial q_x}{\partial t}+\frac{\partial u q_x}{\partial x}+\frac{\partial v q_x}{\partial y}=-gh\frac{\partial H}{\partial x}-\frac{\tau_x}{\rho_w} \quad (2)$$

$$\frac{\partial q_y}{\partial t}+\frac{\partial u q_y}{\partial x}+\frac{\partial v q_y}{\partial y}=-gh\frac{\partial H}{\partial y}-\frac{\tau_y}{\rho_w} \quad (3)$$

where h is the height of water from the local surface, q_x and q_y are the unit width discharges in x and y directions, u and v are the flow velocities in x and y directions, r is the rainfall intensity, f is the infiltration rate, H is the height of water from the datum, ρ_w is the density of water, g is the gravitational acceleration, and τ_x and τ_y are the shear stresses in x and y directions. The second terms of the right side of (2) and (3) are calculated with the Manning's equation.

$$\frac{\tau_x}{\rho_w}=\frac{gn^2u\sqrt{u^2+v^2}}{h^{1/3}} \quad (4)$$

$$\frac{\tau_y}{\rho_w}=\frac{gn^2v\sqrt{u^2+v^2}}{h^{1/3}} \quad (5)$$

where n is the Manning's roughness parameter.

Under the diffusion wave approximation, inertia terms (the left side terms of (2) and (3)) are neglected. Moreover, by separating x and y directions (i.e. ignoring v and u terms in equations (2) and (3) respectively), the following equations are derived:

$$q_x=-\frac{1}{n}h^{5/3}\sqrt{\left|\frac{\partial H}{\partial x}\right|}\,\mathrm{sgn}\left(\frac{\partial H}{\partial x}\right) \quad (6)$$

$$q_y = -\frac{1}{n} h^{5/3} \sqrt{\left|\frac{\partial H}{\partial y}\right|} \operatorname{sgn}\left(\frac{\partial H}{\partial y}\right) \quad (7)$$

where sgn is the signum function.

The RRI model spatially discretizes mass balance equation (1) as follows:

$$\frac{dh^{i,j}}{dt} + \frac{q_x^{i,j-1} - q_x^{i,j}}{\Delta x} + \frac{q_y^{i-1,j} - q_y^{i,j}}{\Delta y} = r^{i,j} - f^{i,j} \quad (8)$$

where $q_x^{i,j}$, $q_y^{i,j}$ are x and y direction discharges from a grid cell at (i, j).

By combining the equations of (6), (7) and (8), water depths and discharges are calculated at each grid cell for each time step. One important difference between the RRI model and other inundation models is that the former uses different forms of the discharge-hydraulic gradient relationship, so that it can simulate both surface and subsurface flows with the same algorithm. The RRI model replaces the equations (6) and (7) with the following equations of (9) and (10), which were originally conceptualized by Ishihara and Takasao (1962) and formulated with a single variable by Takasao and Shiiba (1976, 1988) based on kinematic wave approximations. The first equations in (9) and (10) ($h \le d$) describe the saturated subsurface flow based on the Darcy law, while the second equations ($d_a \le h$) describe the combination of the saturated subsurface flow and the surface flow. Note that for the kinematic wave model, the hydraulic gradient is assumed to be equal to the topographic slope, whereas the RRI model assumes the water surface slope as the hydraulic gradient.

$$q_x = \begin{cases} -k_a h \dfrac{\partial H}{\partial x}, & (h \le d) \\ -\dfrac{1}{n}(h - d_a)^{5/3} \sqrt{\left|\dfrac{\partial H}{\partial x}\right|} \operatorname{sgn}\left(\dfrac{\partial H}{\partial x}\right) - k_a h \dfrac{\partial H}{\partial x}, & (d_a < h) \end{cases} \quad (9)$$

$$q_y = \begin{cases} -k_a h \dfrac{\partial H}{\partial y}, & (h \le d) \\ -\dfrac{1}{n}(h - d_a)^{5/3} \sqrt{\left|\dfrac{\partial H}{\partial y}\right|} \operatorname{sgn}\left(\dfrac{\partial H}{\partial y}\right) - k_a h \dfrac{\partial H}{\partial y}, & (d_a < h) \end{cases} \quad (10)$$

where k_a is the lateral saturated hydraulic conductivity and d_a is the soil depth times the effective porosity.

Equations (11) and (12) can be also used to simulate the effect of unsaturated, saturated subsurface flow and surface flow with the single variable of h (Tachikawa et al. 2004, Sayama and McDonnell 2009 for English).

$$q_x = \begin{cases} -k_m d_m \left(\frac{h}{d_m}\right)^{\beta} \frac{\partial H}{\partial x}, & (h \le d_m) \\ -k_a (h-d_m) \frac{\partial H}{\partial x} - k_m d_m \frac{\partial H}{\partial x}, & (d_m < h \le d_a) \\ -\frac{1}{n}(h-d_a)^{5/3} \sqrt{\left|\frac{\partial H}{\partial x}\right|} \operatorname{sgn}\left(\frac{\partial H}{\partial x}\right) - k_a (h-d_m) \frac{\partial H}{\partial x} - k_m d_m \frac{\partial H}{\partial x}, & (d_a < h) \end{cases} \quad (11)$$

$$q_y = \begin{cases} -k_m d_m \left(\frac{h}{d_m}\right)^{\beta} \frac{\partial H}{\partial y}, & (h \le d_m) \\ -k_a (h-d_m) \frac{\partial H}{\partial y} - k_m d_m \frac{\partial H}{\partial y}, & (d_m < h \le d_a) \\ -\frac{1}{n}(h-d_a)^{5/3} \sqrt{\left|\frac{\partial H}{\partial y}\right|} \operatorname{sgn}\left(\frac{\partial H}{\partial y}\right) - k_a (h-d_m) \frac{\partial H}{\partial y} - k_m d_m \frac{\partial H}{\partial y}, & (d_a < h) \end{cases} \quad (12)$$

Note that to assure the continuity of the discharge change when $h = d_m$, the lateral hydraulic conductivity in unsaturated zone (k_m) can be computed by $k_m = k_a / \beta$, so that k_m is no longer the model parameter.

These stage-discharge relationship equations were originally developed to be applied to humid forest areas with a high permeable soil layer, where a lateral subsurface flow is the dominant runoff generation mechanism. On the other hand, for relatively flat areas, the vertical infiltration process during the first period of rainfall has more impact on large-scale flooding; therefore, the vertical infiltration can be treated as loss for event-based simulation. Here we calculate infiltration loss f with the Green-Ampt infiltration model (Raws et al., 1992).

$$f = k_v \left[1 + \frac{(\phi - \theta_i) S_f}{F} \right] \quad (13)$$

where k_v is the vertical saturated hydraulic conductivity, ϕ is the soil porosity, θ_i is the initial water volume content, S_f is the suction at the vertical wetting front and F is the cumulative infiltration depth.

Typically for mountainous areas where lateral subsurface flow and saturated excess overland flow dominate, the equations (9) and (10) (or (11) and (12)) can be used with setting f equals to be zero. (Note that the equations (9) and (10) (or (11) and (12)) implicitly assume that the vertical infiltration rate within the soil is infinity.) On the other hand, for plain areas where infiltration excess overland flow dominates, the surface flow equations (6) and (7) can be used with the consideration of vertical infiltration by equation (13). If the vertical infiltration f is set to be non-zero and the lateral subsurface equations are used instead of the surface flow equation, the lateral subsurface water is infiltrated to bedrock by the rate of f.

As one can see from the equations, the parameter values of k_a, k_m and k_v decide which equations to be used; i.e. (6) and (7) are used when k_a and k_m are zero, (9) and (10) are used when k_m is zero, and (13) is inactivated when k_v is zero.

1.3 One-dimensional River Routing Model

A one-dimensional diffusive wave model is applied to river grid cells. The geometry is assumed to be rectangle, whose shapes are defined by width W, depth D and embankment height H_e. When detailed geometry information is not available, the width and depth are approximated by the following function of upstream contributing area A [km^2].

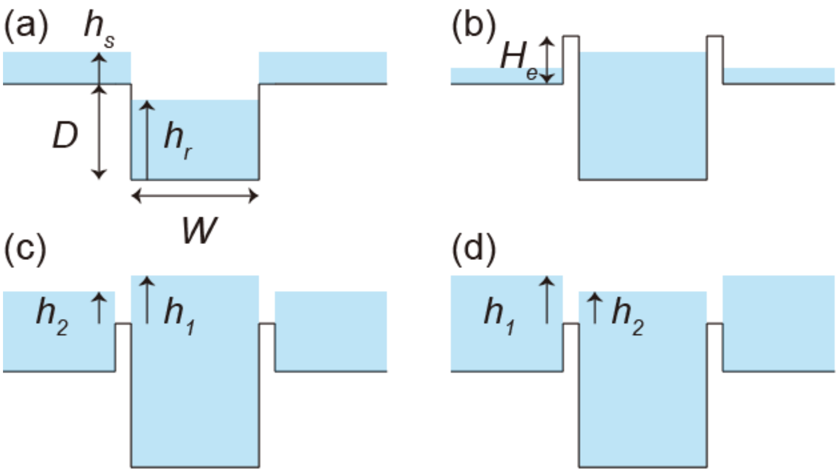
$$W = C_W A^{S_W} \quad (14)$$

$$D = C_D A^{S_D} \quad (15)$$

where C_W, S_W, C_D and S_D are geometry parameters. Here the units of W and D are meters.

1.4 River and Slope Water Exchange

Water exchange between a slope grid cell and an overlying river grid cell is calculated at each time step depending on the relationship among the levels of slope water, river water, levee crown and ground. The figure below shows four different conditions. For each condition, different overtopping formulae are applied to calculate the unit length discharge from slope to river (q_{sr}) or from river to slope (q_{rs}), which are then multiplied by the length of the river vector at each grid cell to calculate the total exchange flow rate (Iwasa and Inoue, 1982).



(a) When the river water level is lower than the ground level, q_{sr} is calculated by the following step fall formula.

$$q_{sr} = \mu_1 h_s \sqrt{g h_s} \quad (16)$$

where μ_1 is the constant coefficient ($=(2/3)^{3/2}$), and h_s is the water depth on a slope cell. As far as the river water level is lower than the ground level, the same equation is used even for the case with levees so that the slope water can flow into the river.

(b) When the river water level is higher than the ground level and both the river and slope water levels are lower than the levee height, no water exchange is assumed between the slope and river.

(c) When the river water level is higher than the levee crown and the slope water level, the following formula is used to calculate overtopping flow q_{rs} from river to slope.

$$q_{rs} = \begin{cases} \mu_2 h_1 \sqrt{2 g h_1} & h_2 / h_1 \le 2/3 \\ \mu_3 h_2 \sqrt{2 g (h_1 - h_2)} & h_2 / h_1 > 2/3 \end{cases} \quad (17)$$

where μ_2 and μ_3 are the constant coefficients (=0.35, 0.91), and h_1 is the difference between the river water level and the levee crown.

(d) When the slope water level is higher than the levee height and the river water level, the same formula as (17) is used to calculate overtopping flow q_{sr} from slope to river. In this case, h_1 is the elevation difference between the slope and the river, and h_2 is the elevation difference between the river and the levee crown.

1.5 Numerical Scheme

To solve equations (8), (9) and (10), the fifth-order Runge-Kutta method with adaptive

time-step control is applied. This method solves an ordinary differential equation by the general fifth-order Runge-Kutta formula and estimates its error by an embedded forth-order formula to control the time-step (Cash and Karp 1990, Press et al. 1992).
The general form of the fifth-order Runge-Kutta formula is

$$\begin{aligned} k_1 &= \Delta t f\left(t, h_t\right) \\ k_2 &= \Delta t f\left(t + a_2 \Delta t, h_t + b_{21} k_1\right) \\ &\cdots \\ k_6 &= \Delta t f\left(t + a_6 \Delta t, h_t + b_{61} k_1 + \cdots b_{65} k_5\right) \\ h_{t+1} &= h_t + c_1 k_1 + c_2 k_2 + c_3 k_3 + c_4 k_4 + c_5 k_5 + c_6 k_6 + O\left(\Delta t^6\right) \end{aligned} \quad (18)$$

while the embedded forth-order formula (Cash and Karp 1990) is

$$h_{t+1}^* = h_t + c_1^* k_1 + c_2^* k_2 + c_3^* k_3 + c_4^* k_4 + c_5^* k_5 + c_6^* k_6 + O\left(\Delta_t^5\right) \quad (19)$$

By subtracting h_{t+1} minus h^*_{t+1}, the error can be estimated by using k_1 to k_6 as follows,

$$\delta \equiv h_{t+1} - h_t^* = \sum_{i=1}^{6} \left(c_i - c_i^*\right) k_i \quad (20)$$

The constant values (a_i, b_{ij}, c_i, c_i^*) used in this study are the ones introduced by Cash and Karp (1990). If δ exceeds a desired accuracy δ_d, h_{t+1} is recalculated with a smaller time step (Δt_{post}).

$$\Delta t_{post} = \max\left(0.9 \Delta t \left|\frac{\delta_d}{\delta}\right|^{0\ 25}, 0.5 \Delta t\right) \quad (21)$$

As described above, the RRI model calculates slopes, rivers and slope-river interactions. Model users specify the time step for slope-river interaction Δt, which is also used as an initial time step for slope calculations. Since river calculations usually require smaller time steps because of higher water velocities and depths, the model allows river calculations to proceed independently with different time steps until the next river-slope calculation time step. The initial time step for river calculation (Δt_r) can be also specified by model users as the common divisor of Δt. In this study, $\delta_d = 0.01$, $\Delta t = 600$ sec. and $\Delta t_r = 60$ sec. were used.

References

- Sayama, T., Ozawa, G., Kawakami, T., Nabesaka, S., Fukami, K. (2012) Rainfall-Runoff-Inundation analysis of the 2010 Pakistan flood in the Kabul River basin, *Hydrological Science Journal*, 57(2), 298-312.

- Hunter, N.M., Bates, P.D., Horritt, M.S., Wilson, M.D. (2007) Simple spatially-distributed models for predicting flood inundation: A review. *Geomorphology*, 90, 208-225.

- Takasao, T., Shiiba, M. (1976), A study on the runoff system model based on the topographical framework of river basin, *Proceedings of the Japan Society of Civil Engineers*, 248, 69-82, (in Japanese with English abstract).

- Takasao, T., Shiiba, M. (1988) Incorporation of the effect of concentration of flow into the kinematic wave equations and its applications to runoff system lumping. *Journal of Hydrology*, 102, 301– 322.

- Ishihara, T., Takasao T. (1962) A study on the subsurface runoff and its effects on runoff process, *Transactions of the Japan Society of Civil Engineers*, 79, 15-23, (in Japanese with English abstract).

- Tachikawa, Y., G. Nagatani, K. Takara (2004), Development of stage discharge relationship equation incorporating saturated-unsaturated flow mechanism (in Japanese), *JSCE Annual Hydraulic Engineering*, 48, 7 – 12, (in Japanese with English Abstract).

- Sayama, T., McDonnell, J. J. (2009) A new time-space accounting scheme to predict stream water residence time and hydrograph source components at the watershed scale, *Water Resour. Res.*, 45, W07401.

- Raws, W.J., Ahuja, L.R., Brakensiek, D.L. & Shirmohammadi, A. (1992) Inflitration and soil water movement. Handbook of Hydrology, McGrow-Hill Inc., New York, 5.1-5.51.

- Iwasa, Y., Inoue, K. (1982) Mathematical simulation of channel and overland flood flows in view of flood disaster engineering, *Journal of Natural Disaster Science*, 4(1), 1-30.

- Cash, J.R. & Karp, A.H. (1990) A variable order Runge-Kutta method for initial value problems with rapidly varying right-hand sides. *ACM Trans. on Math. Software*, 16(3), 201-222.

- Press, W.H., Teukolsky, S.A., Vetterling, W.T. & Flannery, B.P. (1992) Adaptive stepsize control for Runge-Kuta. Numerical Receipes in Fortran 77, The Art of Scientific Computing Second Edition, Cambridge University Press, 708-716.

2. Getting Started

There are essentially five steps to conduct RRI Model simulation.
1. Preparing topography data (Section 3)
2. Preparing input rainfall data (Section 4)
3. Preparing model condition files with parameter settings (Section 5)
4. Executing RRI Model. (Section 6)
5. Plotting output data (Section 7)

Among the five steps, only the essence of step 4 and 5 are described here with sample data of the Solo River Basin (in 30 sec resolution) in Indonesia.

2.1 Compile

Unzip "RRI_1.4.1.zip" and save it on a working directory (e.g. C:/).

Open CUI environment where you can compile Fortran programs

(For Intel Fortran Users on Windows)
Start → Program → Intel(R) Software Development Tools → Intel(R) Fortran Compiler ** → Fortran Build Environment for Applications running on …

Move current directory to "/RRI/Model/", then type "make_1_4_1.bat" and enter to create "0_rri_1_4_1.exe". (Command "*cd directory_name*" is used to enter the directory, "cd ../" is used to move up one directory, and "dir" is used to list all files and directories in the current directory. To move D directory, just type "D:" and enter)

Also type "makePostProsess.bat" and enter to create "calcHydro.exe" and "calcPeak.exe", which are used for post processing.

Make sure if "0_rri_1_4_1.exe", "calcHydro.exe" and "calcPeak.exe" are newly created.

Note: If Intel Fortran Compiler is not available on your PC, use another Fortran compiler (e.g. gfortran) to compile. Although the performance is not guaranteed, using the precompiled executable files (0_rri_1_4_1.exe) is an option (on 64 bit windows machine). In that case, run the following redistribution patch program before running the RRI executable files:
(/etc/w_fcompxe_redist_msi_2011.10.325/w_fcompxe_redist_intel64_2011.10.325.msi)

2.2 Run RRI Model

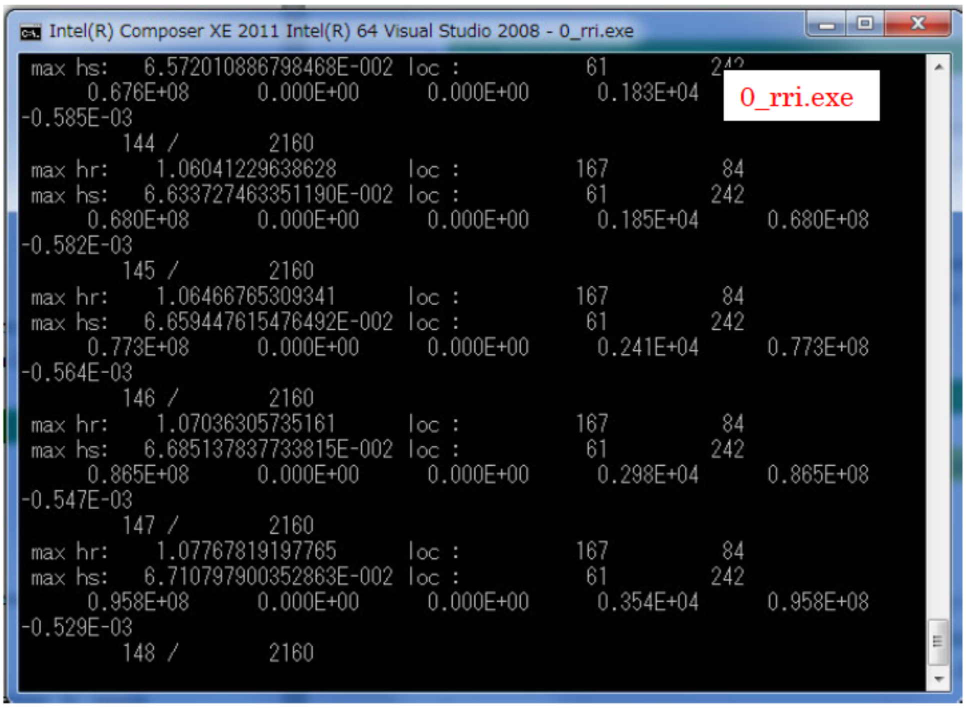
Look at "RRI_Input.txt", which is the control file of RRI Model. You can edit the control file to change the simulation settings including input files and parameters.

RRI_Input.txt

```
RRI_Input_Format_Ver1_4_1

./infile/solo30s/rain_solo_30s_gauge.dat
./infile/solo30s/adem_30s_solo.txt
./infile/solo30s/acc_30s_solo.txt
./infile/solo30s/adir_30s_solo.txt

0                   # utm(1) or latlon(0)
1                   # 4-direction (0), 8-direction(1)
360                 # lasth
600                 # dt
```

For example, L3 specifies the path to an input rainfall file and L4 – L6 specify the paths to input topography files (dem, acc, and dir). See Section 4 for more details on the RRI_Input.txt file.

Execute "0_rri_1_4_1.exe" to run RRI Model.



Confirm the output files are successfully created inside the directory of "RRI/Model/out". Note that “hr_000001.out” represents the spatial distribution of river water depths in [m] at the output time step 1. “hs_000001.out” and “qr_000001.out” represent those of slope water depths in [m] and river discharge in [m^3/s], respectively.

2.3 Post Analysis

2.3.1 Visualize Inundation Depth (./out/hs_***.out) with GNUPLOT

Run a GNUPLOT installation program “RRI/etc/gp463-win32-setup.exe” and install it onto your PC. If the installation is successful, “gnuplot” folder is appeared under All programs of windows. Choose “gnuplot 4.6” to run GNUPLOT.

Open “RRI/Model/hs.plt” with a text editor. It is a GNUPLOT script file to convert from the simulation outputs (e.g. ./out/hs_***.out) to gif files to visualize inundation depth distributions.

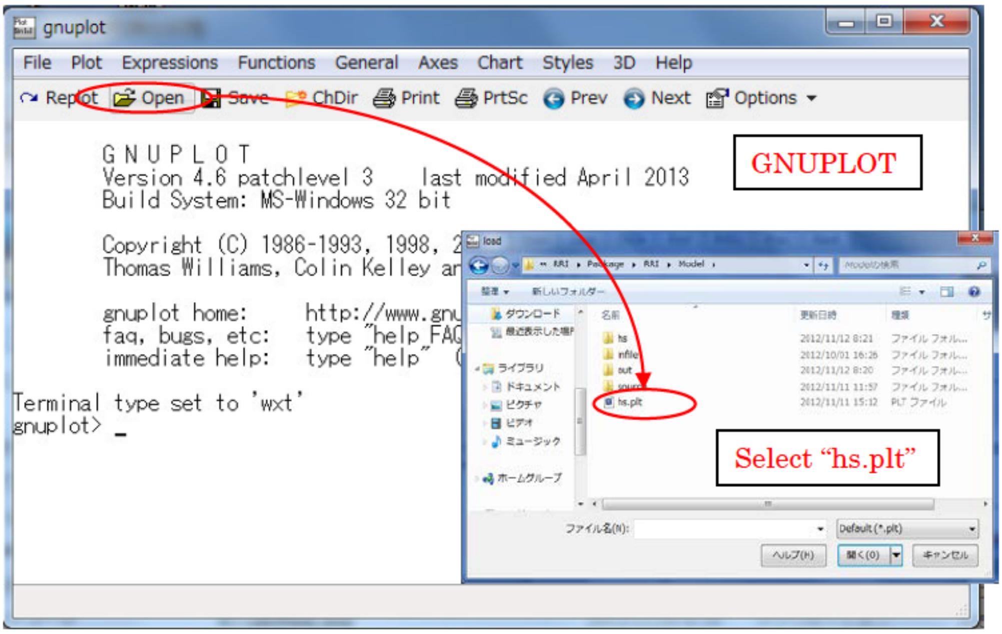
hs_plt.txt

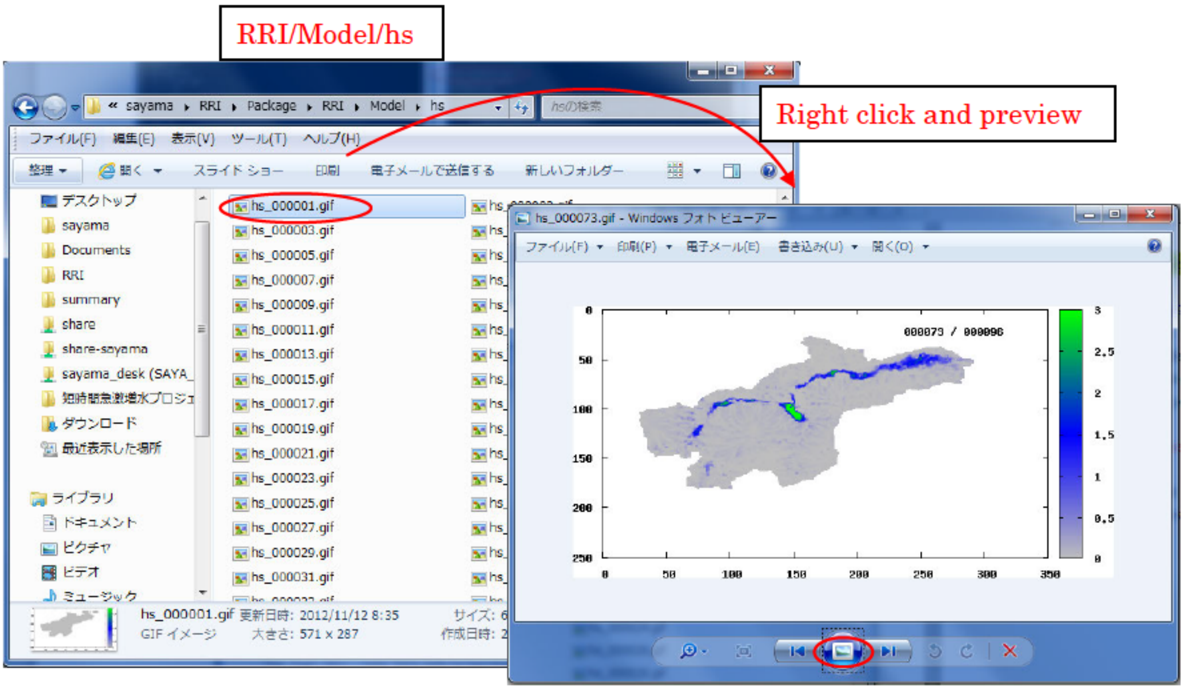
```
reset

set terminal gif medium size 672, 408 crop

set pm3d map
set palette defined (0.0 "gray", 1.5 "blue", 3 "green")

set xrange [0:]
set yrange [:] reverse
set zrange [0:] reverse

#set xrange [180:200]
#set yrange [435:455] reverse

set cbrange[0.:3]
set zrange[0.0:]

set output "./hs/hs_000001.gif"
splot "./out/hs_000001.out" matrix t "000001 / 000096"

set output "./hs/hs_000002.gif"
splot "./out/hs_000002.out" matrix t "000002 / 000096"

set output "./hs/hs_000003.gif"
splot "./out/hs_000003.out" matrix t "000003 / 000096"
```

From RRI output (hs_***.out) to gif

Select "Open" on GNUPLOT Toolbar and open "/RRI/Model/hs.plt", which is a script file to create gif files from the RRI output (see above figure).

Look at "RRI/Model/hs" directory, where gif files are newly created. Check the created gif files by preview.





2.3.2 Compute hydrograph

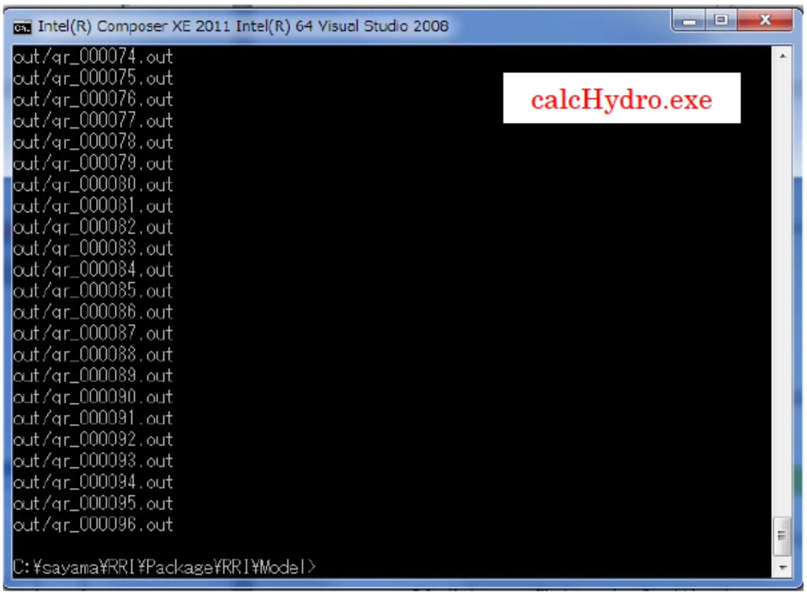
Look at "RRI/Model/calcHydro.txt" (see more details "RRI/etc/calcHydro/00_readme.txt")

L1 : [In] location file (e.g. ./infile/solo30s/location_30s_solo.txt)

L2 : [In] RRI output file (e.g. ./out/qr_***.txt)

L3 : [Out] hydrograph file (e.g. ./infile/solo30s/disc_Cepu.txt)

Execute "calcHydro.exe".



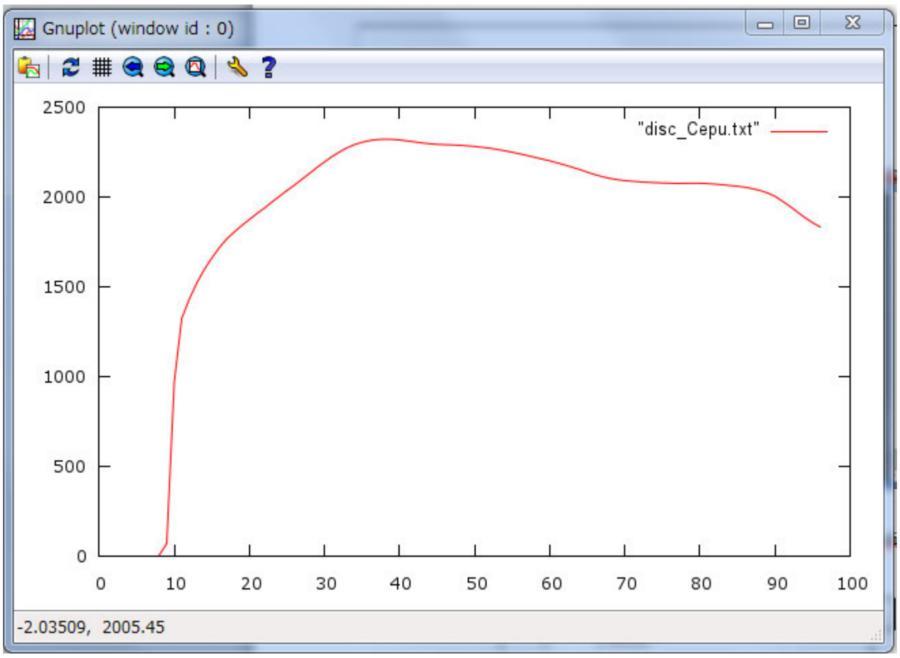
A hydrograph file named "disc_Cepu.txt" is created inside "RRI/Model/infile/solo30s".

./infile/solo30s/disc_cepu.txt

1	0.00789
2	0.04591
3	0.08256
4	0.10557
5	0.12529
6	0.14543
7	0.24838
8	0.56375
9	69.88281
10	967.36834
11	1322.37727
12	1429.53330
13	1518.85970
…..	……

Visualize the created hydrograph file (e.g. "./infile/solo30s/disc_Cepu.txt") by GNUPLOT.

From GNUPLOT screen, open and select "hydrograph.plt", which is a GNUPLOT script file to plot hydrograph from the "disc_Cepu.txt".



2.3.3 Compute and visualize peak inundation depths

Look at "RRI/Model/calcPeak.txt" (see more details "RRI/etc/calcPeak/00_readme.txt") and edit the file if necessary.

L1 : [in] dem file

L2 : [in] output file (e.g. ./out/hs_***.out)

L3 : [in] the number of output files

L4 : [out] output peak inundation depth file (e.g. ./infile/solo30s/hpeak_30s_solo.txt)

Execute "calcPeak.exe". Then the file "hpeak_30s_solo.txt" is created under /infile/solo30s/

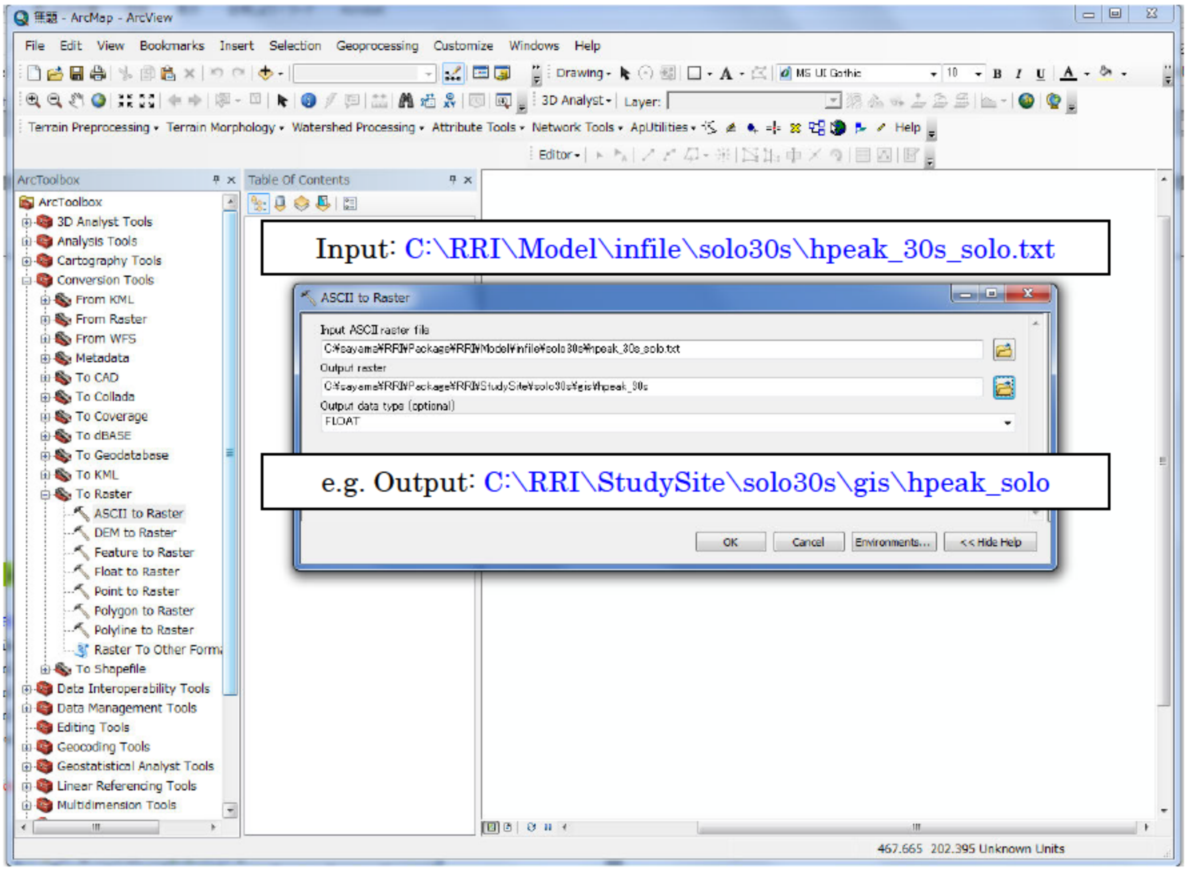
hpeak_30s_solo.txt

```
ncols          336
nrows           204
xllcorner      110.2
yllcorner      -8.3
cellsize       0.00833333333333
NODATA_value   -9999
 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
```

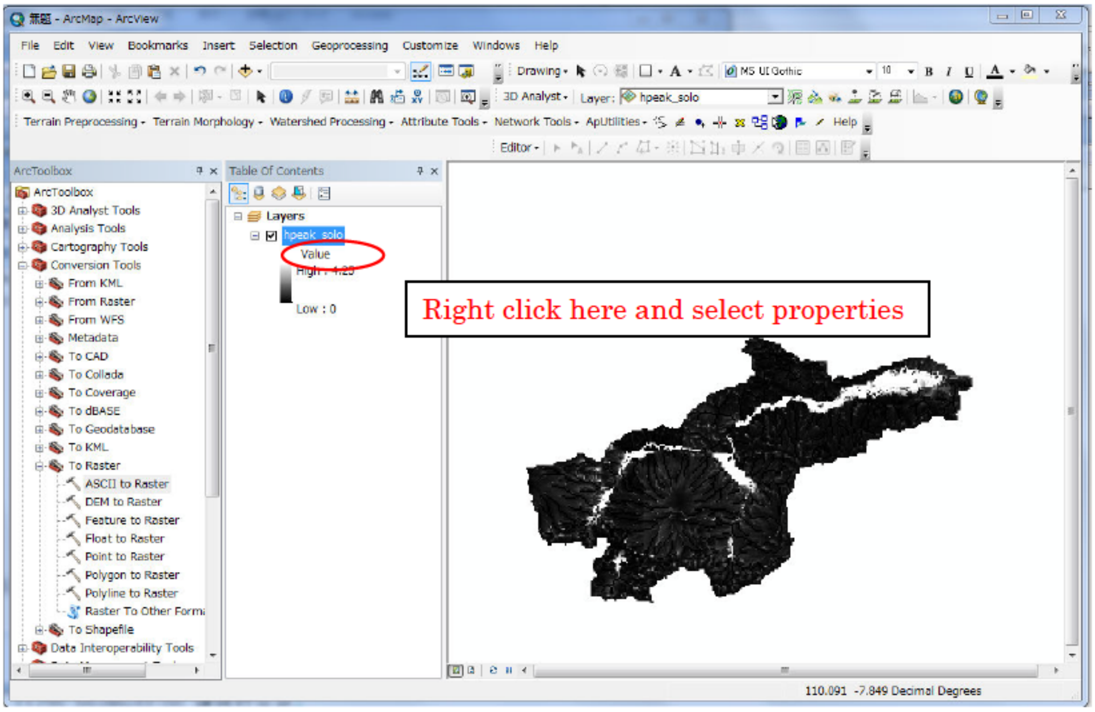
Visualize the created “hpeak_30s_solo.txt” on ArcGIS by converting it from ASCII to Raster.

1) Start ArcGIS (Skip the following procedure if ArcGIS softwre is inaccessible. Consider the use of GRASS GIS by following the instruction in 3.3)

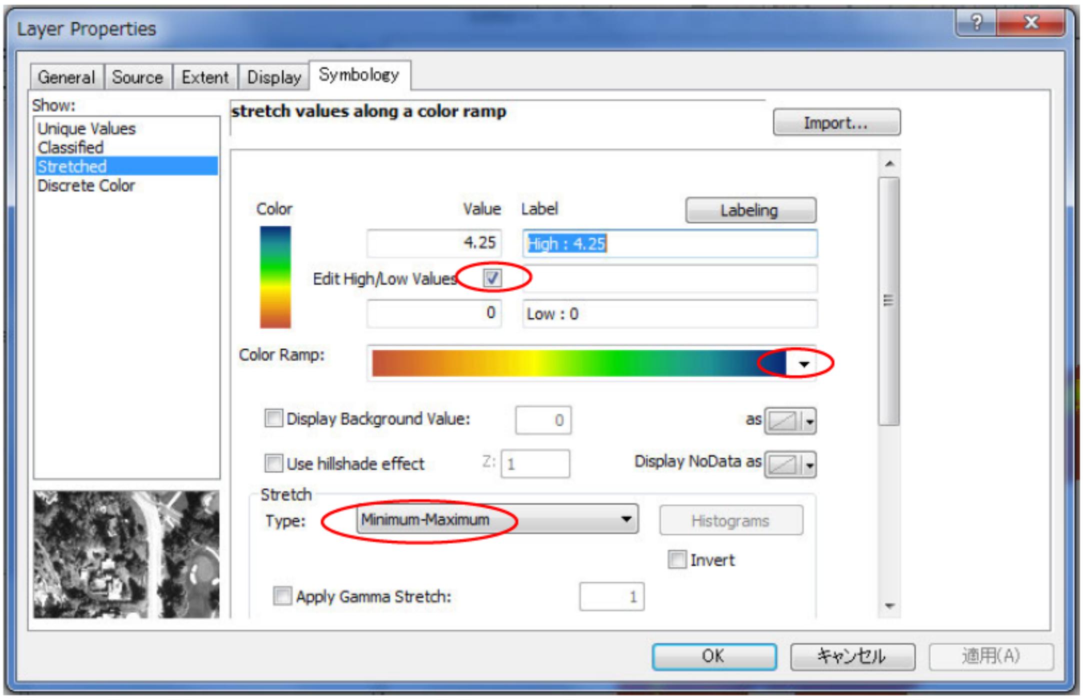
2) From ArcToolbox → [Conversion Tools] → [To Raster] → [ASCII to Raster]



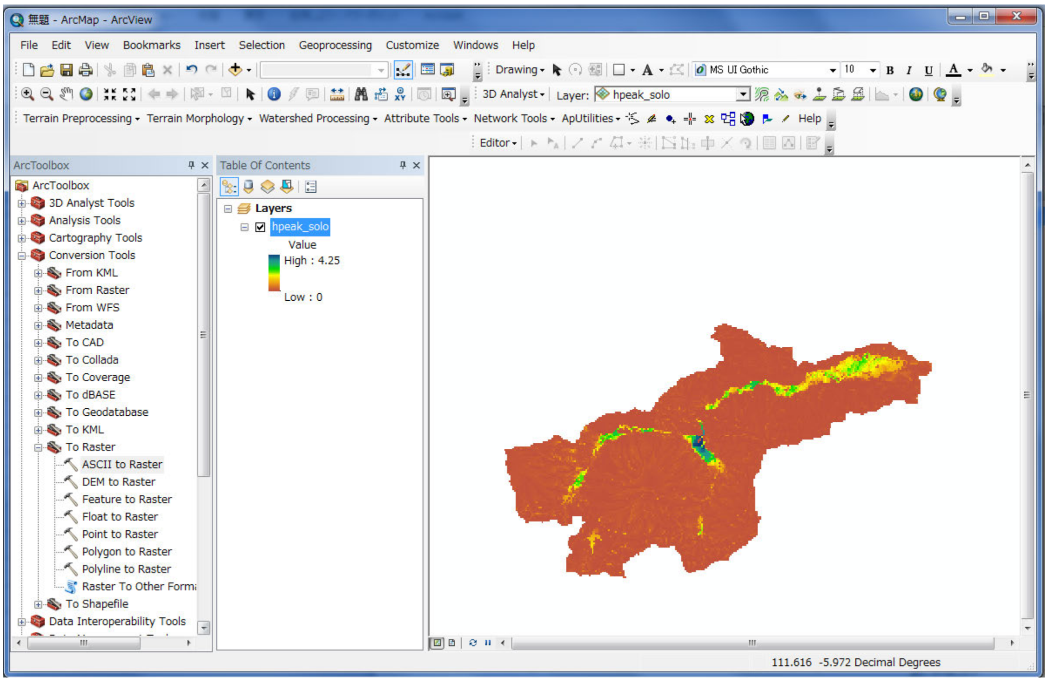
3) For the input data, select “hpeak_30s_solo.txt”. For the output raster, a user may use “RRI/StudySite/solo30s/gis/hpeak_solo”.



4) Right click “hpeak_30s” and select properties to change the layer color setting.



5) On the layer property, change the stretch type to “Minimum-Maximum” and change Color Ramp if necessary. By checking “Edit High/Low Values”, you can change the max and min value range of the stretching.

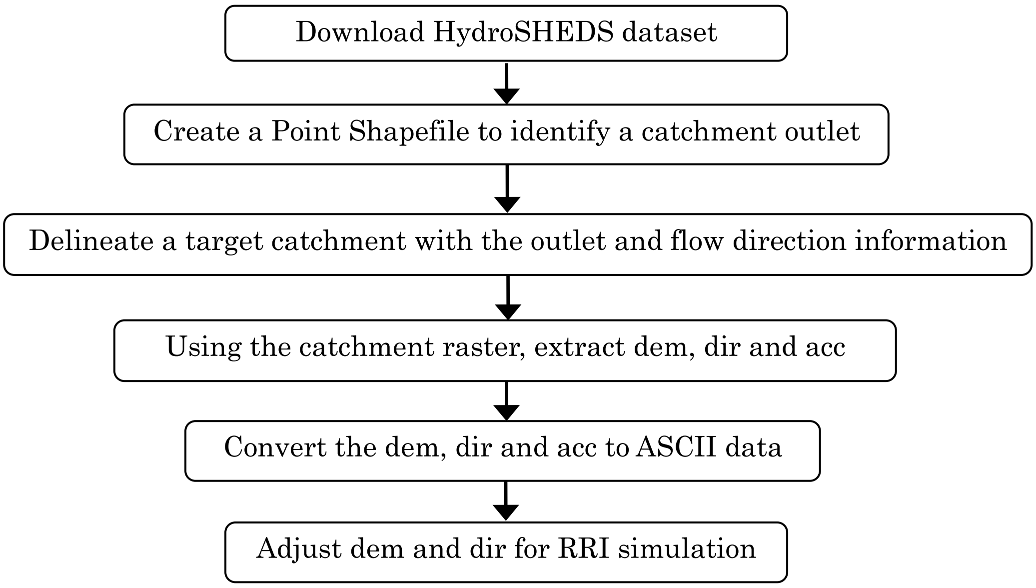

無題 - ArcMap - ArcView
File Edit View Bookmarks Insert Selection Geoprocessing Customize Windows Help
Drawing
MS UI Gothic
3D Analyst
Layer: hpeak_solo
Terrain Preprocessing Terrain Morphology Watershed Processing Attribute Tools Network Tools ApUtilities Help
Editor
ArcToolbox
ArcToolbox
3D Analyst Tools
Analysis Tools
Cartography Tools
Conversion Tools
From KML
From Raster
From WFS
Metadata
To CAD
To Collada
To Coverage
To dBASE
To Geodatabase
To KML
To Raster
ASCII to Raster
DEM to Raster
Feature to Raster
Float to Raster
Point to Raster
Polygon to Raster
Polyline to Raster
Raster To Other Form
To Shapefile
Data Interoperability Tools
Table Of Contents
Layers
hpeak_solo
Value
High : 4.25
Low : 0
111.616 -5.972 Decimal Degrees

3. Preparing Input Topography Data

This section shows the method to prepare topography data input to the RRI Model. The topography data can be prepared by a user or downloaded from the website of USGS HydroSHEDS, which is a global scale dataset offered by the United States Geological Survey (USGS). The dataset includes elevation, flow direction and flow accumulation.

From the downloaded topographic dataset, a user must clip out the target river basin and save them as ESRI/ASCII format files. Then using a program included in RRI Model package, one adjusts the original DEM and flow direction data to be suitable for the RRI simulation. The following chart shows the procedure descried in this section. In the previous section, the 30 second resolution of the Solo River Basin data was used, whereas this section presents how to prepare the topographic data in 15 second.

The flow of the procedure is as follows.



3.1 Downloading HydroSHEDS Data

The following three types of topography data must be downloaded from HydroSHEDS website for RRI simulation.

1) Elevation data

3 arc-second (about 90 m), 15 arc-second (about 500 m), and 30 arc-second (about 1,000 m) are available.

2) Flow direction data

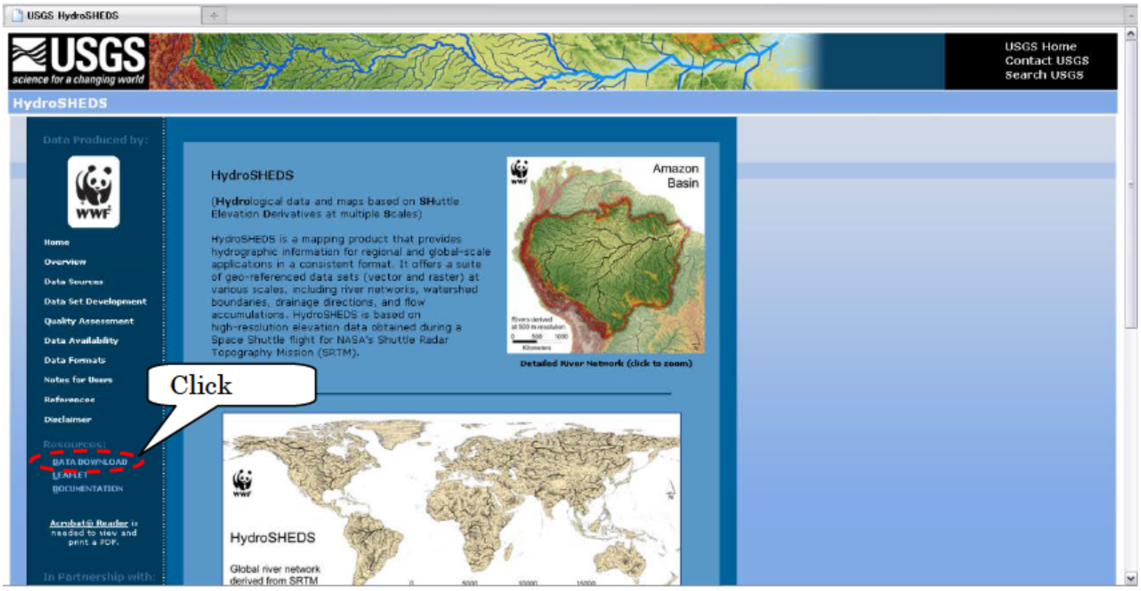
3 arc-second, 15 arc-second, and 30 arc-second are available.

3) Flow accumulation data

Only 15 arc-second and 30 arc-second are available. For 3 arc-second resolution, a user must prepare a flow accumulation by using a GIS function [Spatial Analyst] → [Hydrology] → [Flow Accumulation].

※ For detailed specifications of HydroSHEDS, refer to HydroSHEDS Technical Documentation packaged with the downloaded data.

① Access USGS HydroSHEDS website (http://hydrosheds.cr.usgs.gov/index.php) from a web browser and then select and click the DATADOWNLOAD button on the lower left.



② Select "15sec GRID: Conditioned DEM" and download "as_dem_15s_grid.zip" (207 MB) for Asian region with 15 sec grid-size. NOTE that for 3 sec, recommend to choose "Void-filled DEM". For 15 sec and 30 sec, only "Conditioned DEM" is available, and they are previously named as "Void-filled DEM".

③ Select also "15 sec GRID: Flow Accumulation" and "15 sec Flow Direction" to download "as_acc_15s_grid.zip" (132 MB) and "as_dir_15s_grid.zip" (64 MB) as well.

④ Unzip the three types of topography data downloaded.

※Folder naming rule

"Continental range" _"Data type"_ "resolution"

e.g.) as_acc_15s → Asia catchment area data 15 sec

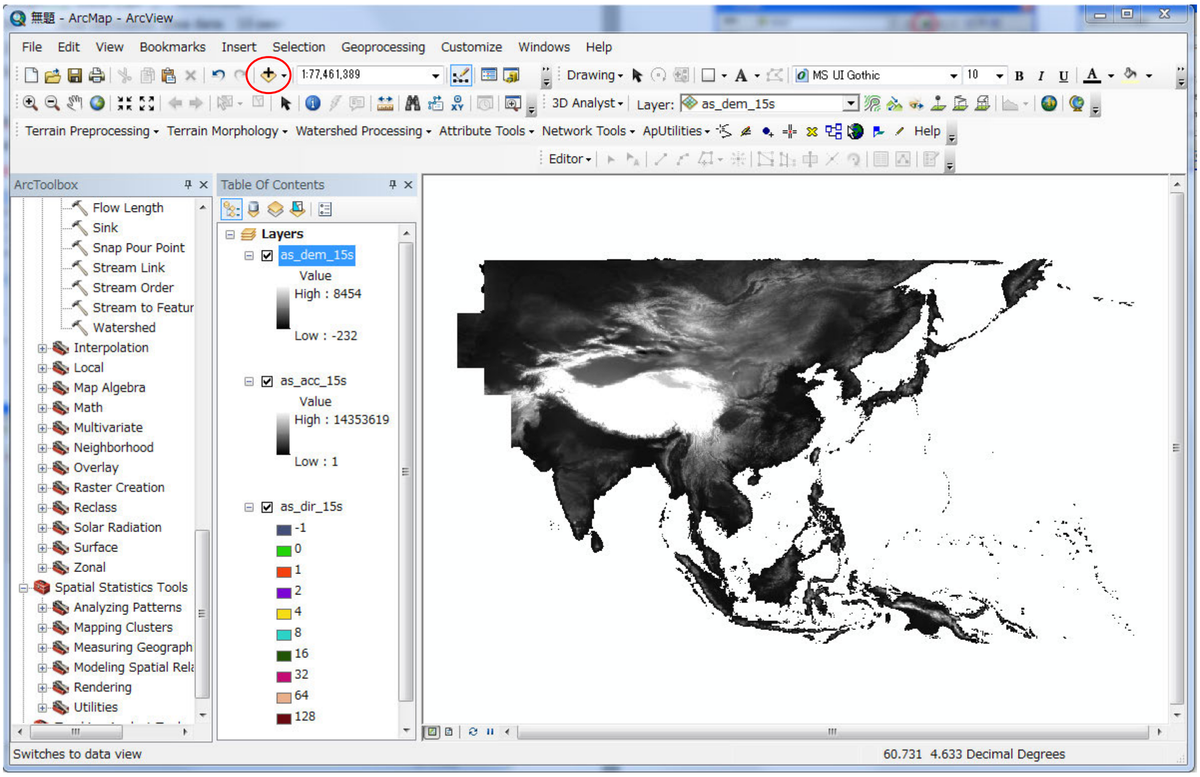
as_dem_15s → Asia digital elevation data 15 sec

as_dir_15s → Asia flow direction data 15 sec

3.2　Delineating HydroSHEDS Data using ArcGIS

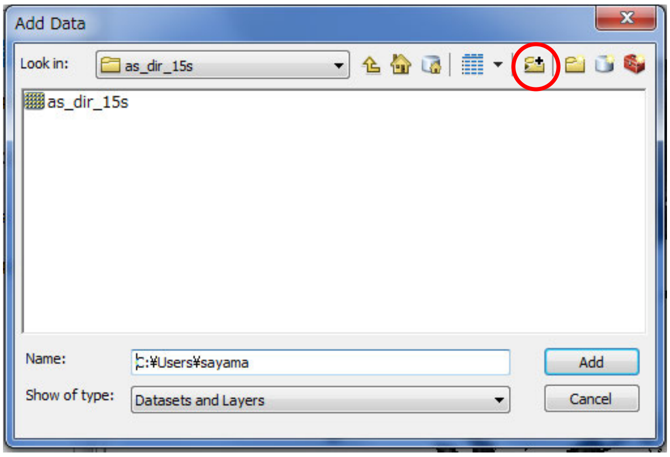
(If ArcGIS is inaccessible, skip this section and go to 3.3 to use free GLASS GIS)

① Start ArcMap, and read in the unzipped files by selecting [File]＞[Add Data]. (Or use icon of "Add Data" on the standard tool bar). Perform the same operation for all the three types (dem, dir, acc) of topography data.

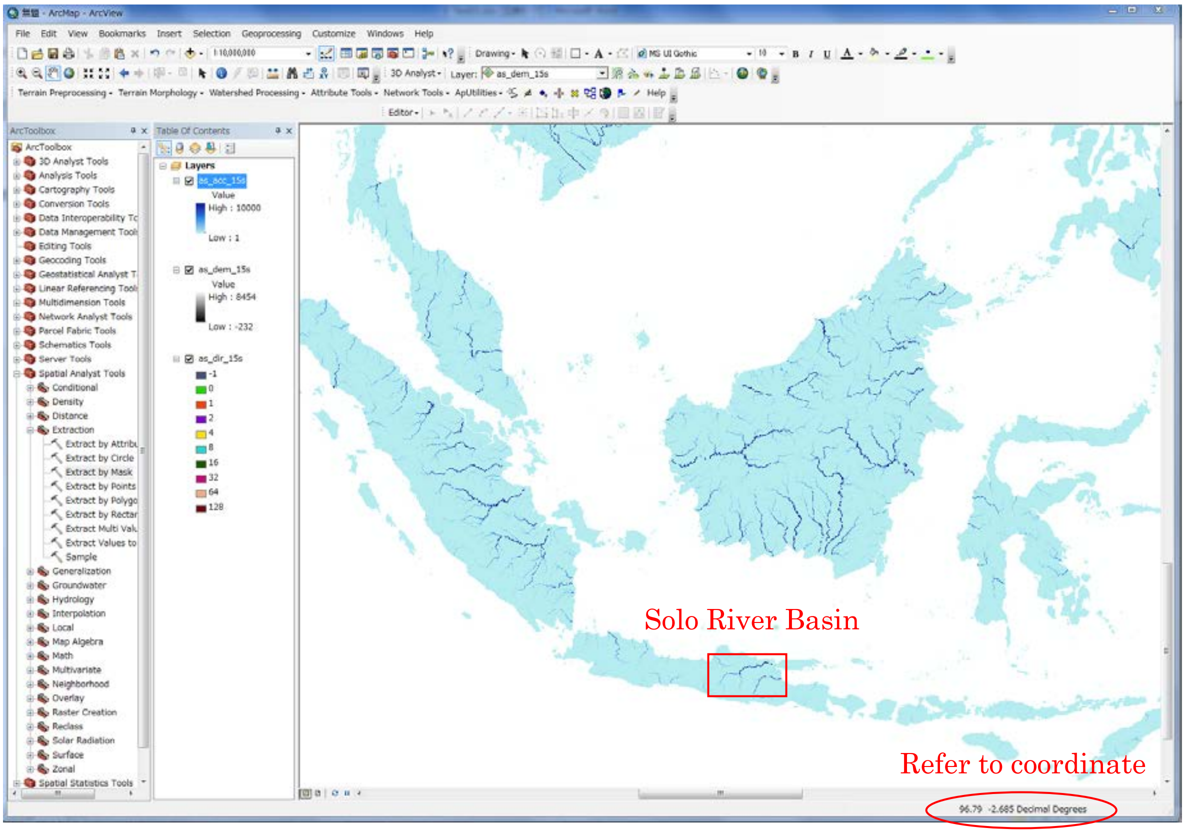


※Selecting the folder to connect

If the folder you need to connect is not displayed in the window, click "Connect to Folder" to connect to the working folder.



③ Display the flow accumulation data (i.e. as_acc_15s) on top screen (change the color range to show river network clearly). Then find your target river and decide the rectangular range, which covers all upstream contributing area. (At this stage, the following rectangle range should be just written down on your notebook and no operation is necessary with GIS.)



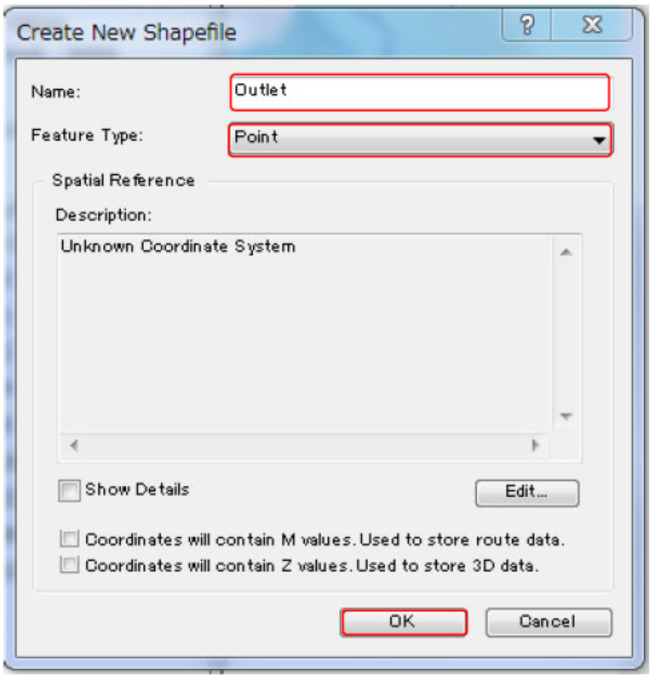
-6.6

110.2 e.g.) Solo River 113.0

-8.3

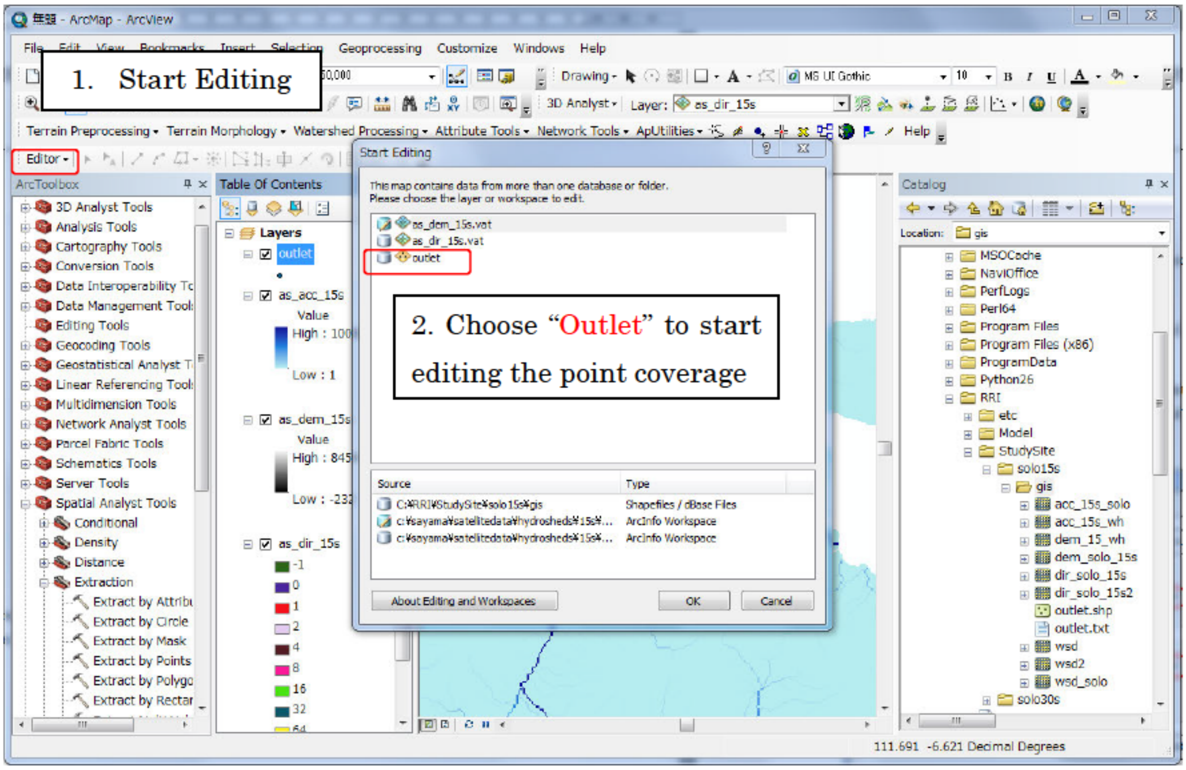
(The range should be written down on your notebook.)

④ Show arc catalog (from the main menu, [Windows] → [Catalog]).
On the arc catalog, "Folder Connections" to a working folder (e.g. /RRI/StudySite/Solo15s/gis/) and right click to choose New → Shapefile to create a point Shapefile (e.g. Outlet).



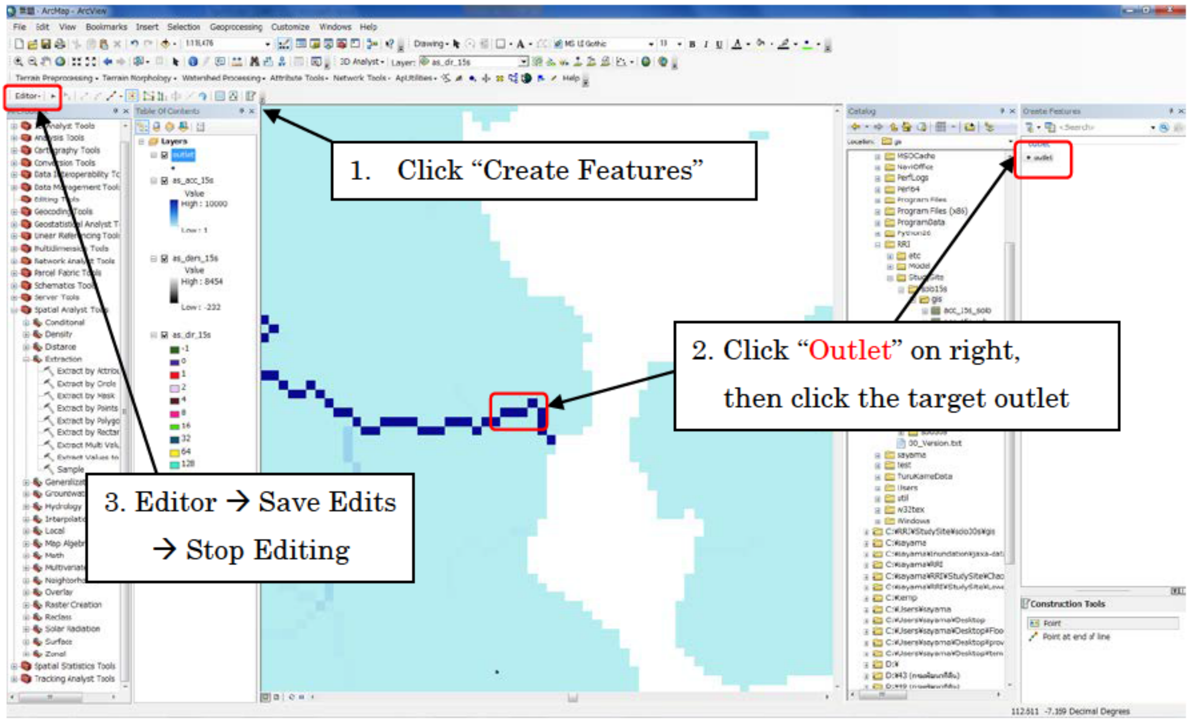
⑤ From the main menu [Customize] → [Toolbars] → [Editor]

On the Editor, choose [Start Editing], then Choose "Outlet" (the new Shapefile) to start editting.



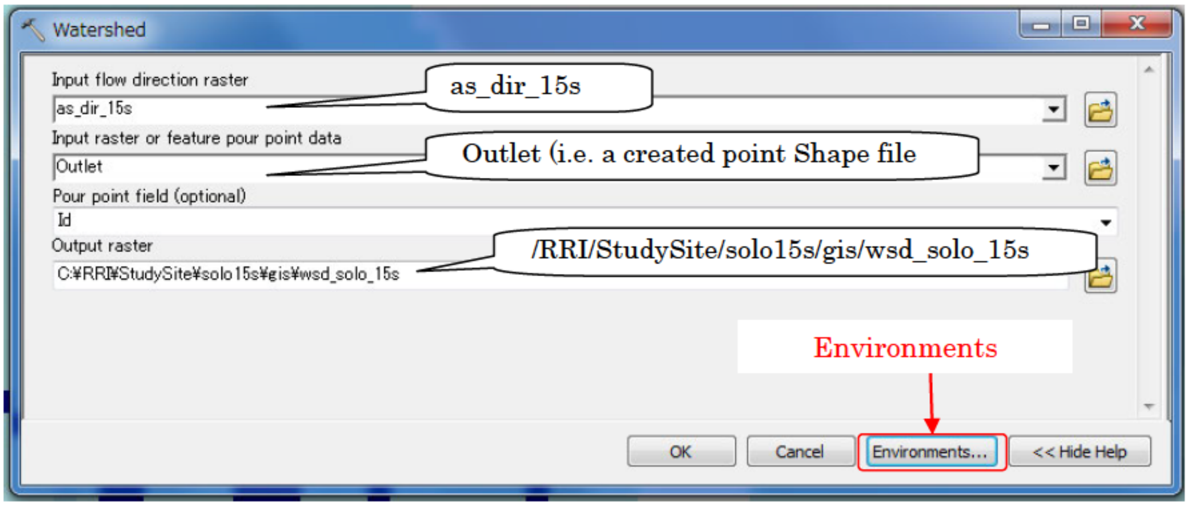
Clicking "Outlet", so that you can bring a point to indicate the target outlet.

After editting the outlet point, go to the editor menu to save and stop editing.

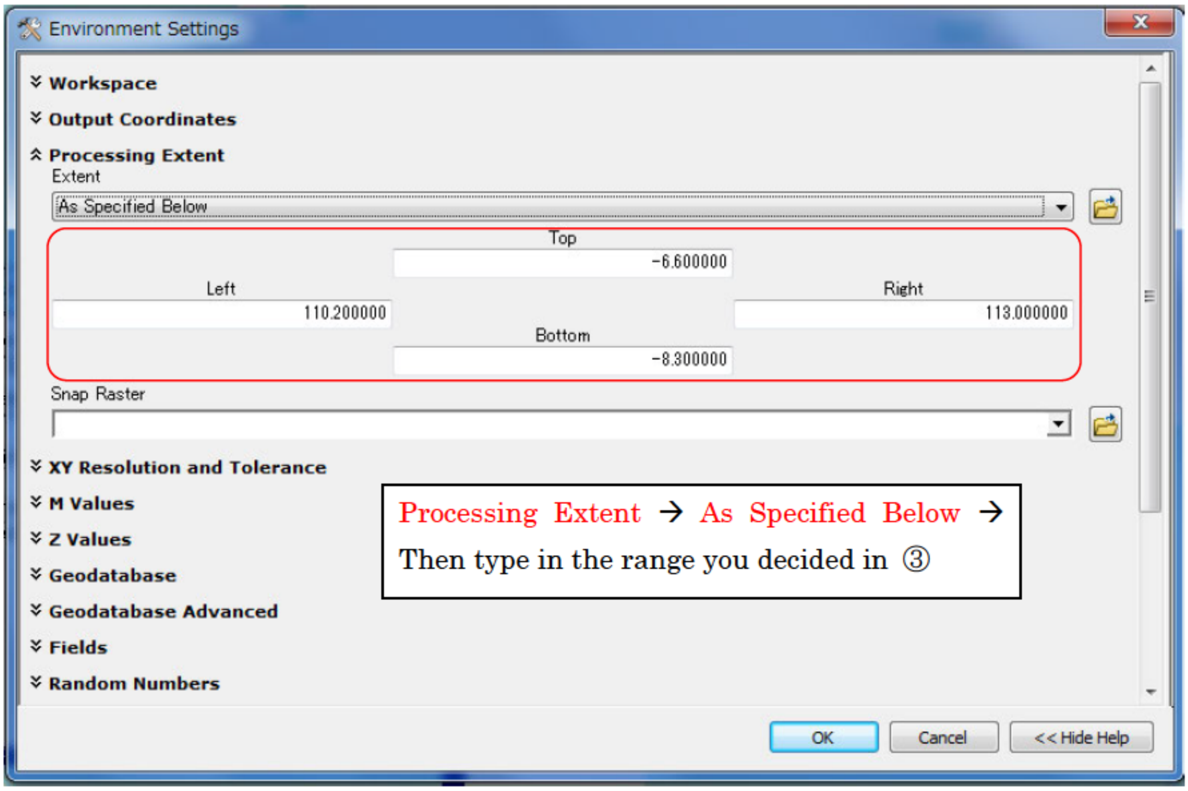


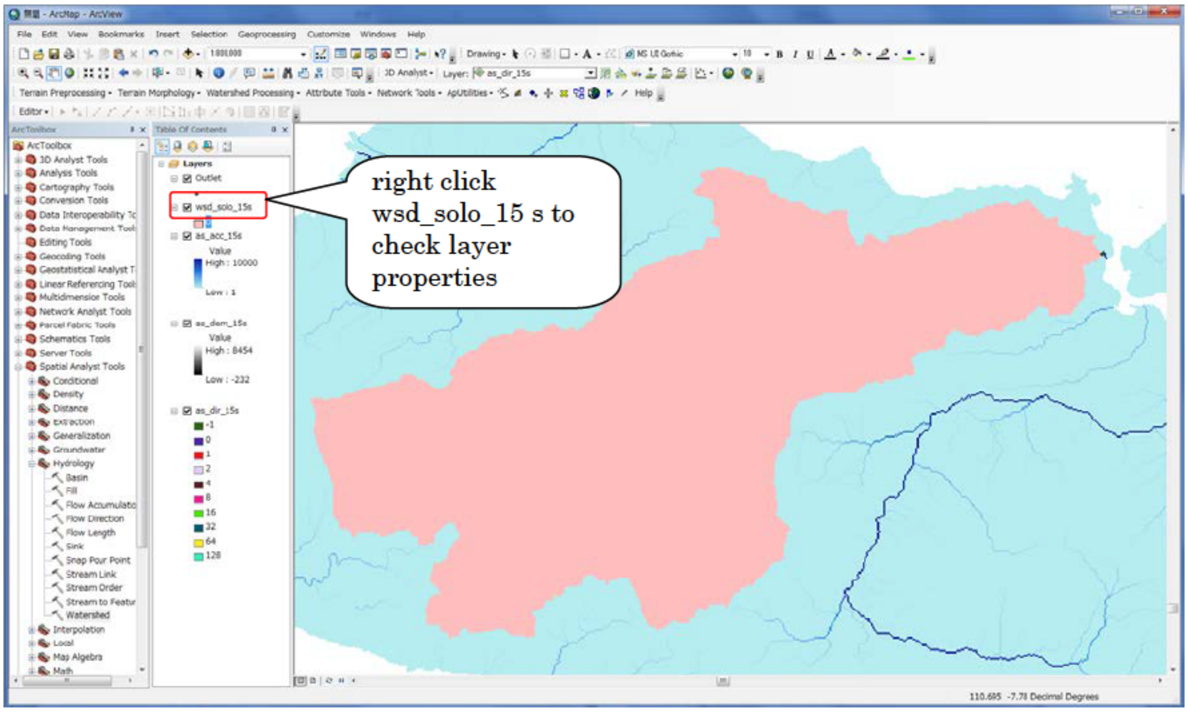
⑥ Using [ArcToolbox] → [Spatial Analyst Tools] → [Hydrology] → [Watershed], delineate a watershed with the defined outlet.

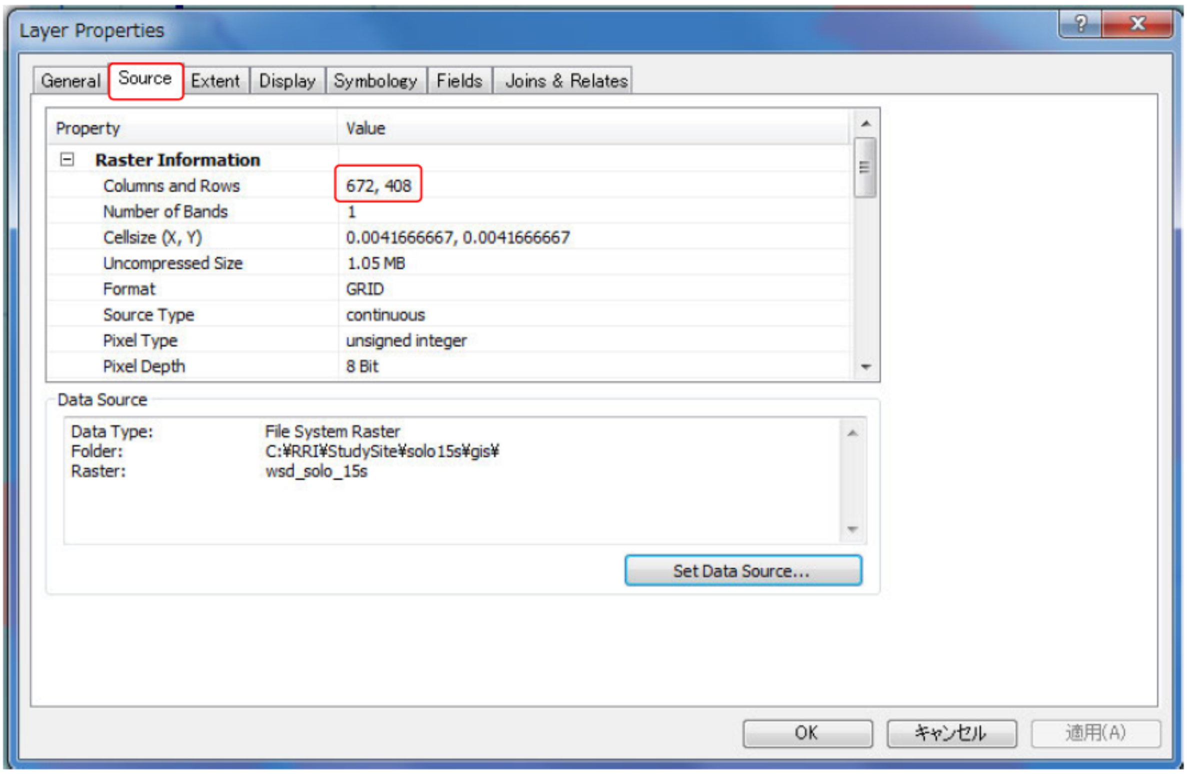
(IMPORTANT) To use [Spatial Analyst Tools] on ArcGIS, you must have the extension and activate it by choosing [Cusomize] → [Extentions] → add a check for [Spatial Analyst].



(IMPORTANT) Analysis range must be specified from the "environment" as below;

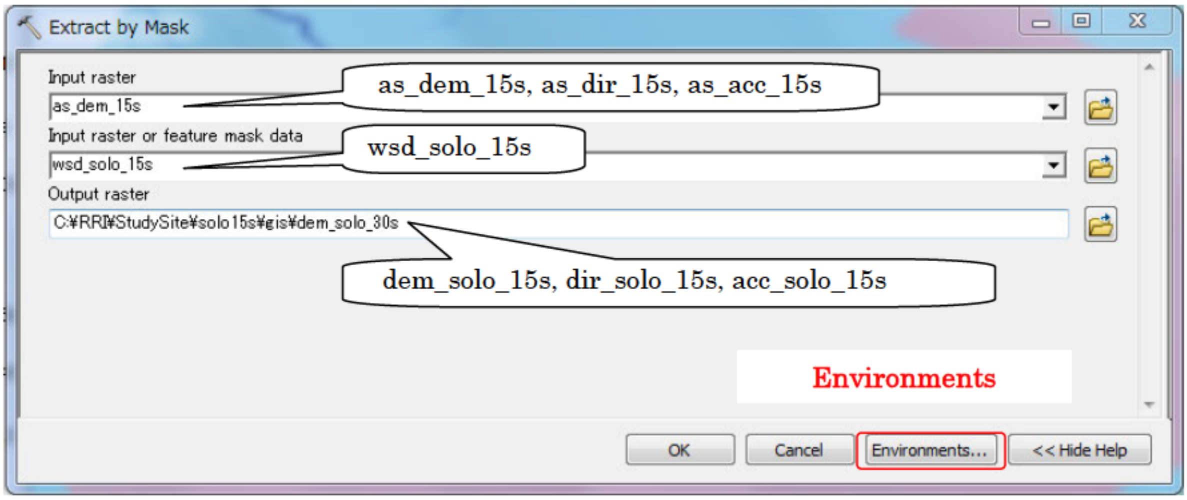

Environment Settings
Workspace
Output Coordinates
Processing Extent
Extent
As Specified Below
Top
-6.600000
Left
110.200000
Right
113.000000
Bottom
-8.300000
Snap Raster
XY Resolution and Tolerance
M Values
Z Values
Geodatabase
Geodatabase Advanced
Fields
Random Numbers
Processing Extent → As Specified Below →
Then type in the range you decided in ③
OK
Cancel
<< Hide Help


right click
wsd_solo_15 s to
check layer
properties

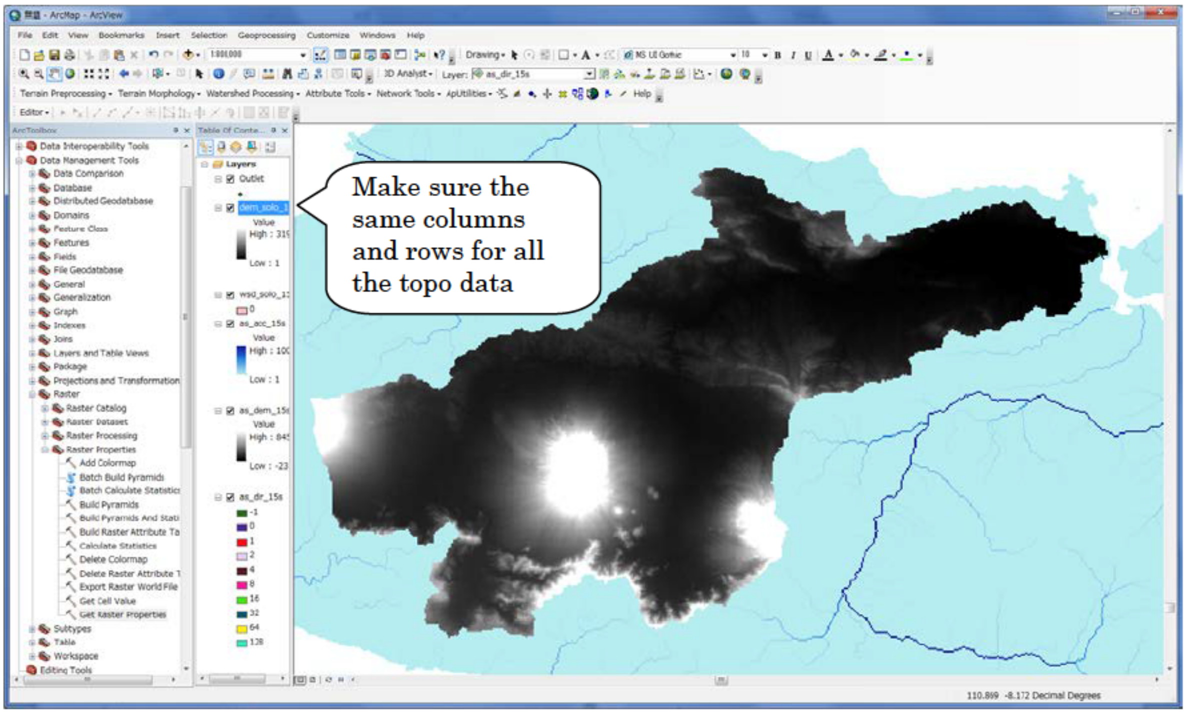


⑦ Right click the created watershed raster (e.g. wsd_solo_15s) and check layer properties. Under the "Source" tab, you can check "Columns and Rows". This will be the number of columns and rows for the topographic data used by RRI Model. If it exceeds more than 1000, using coarser resolution data is recommended to use.

⑧ Using [Spatial Analyst Tools] → [Extract] → [Extract by Mask], prepare dem (elevation), acc (flow accumulation) and dir (flow direction) masked by the delineated watershed.

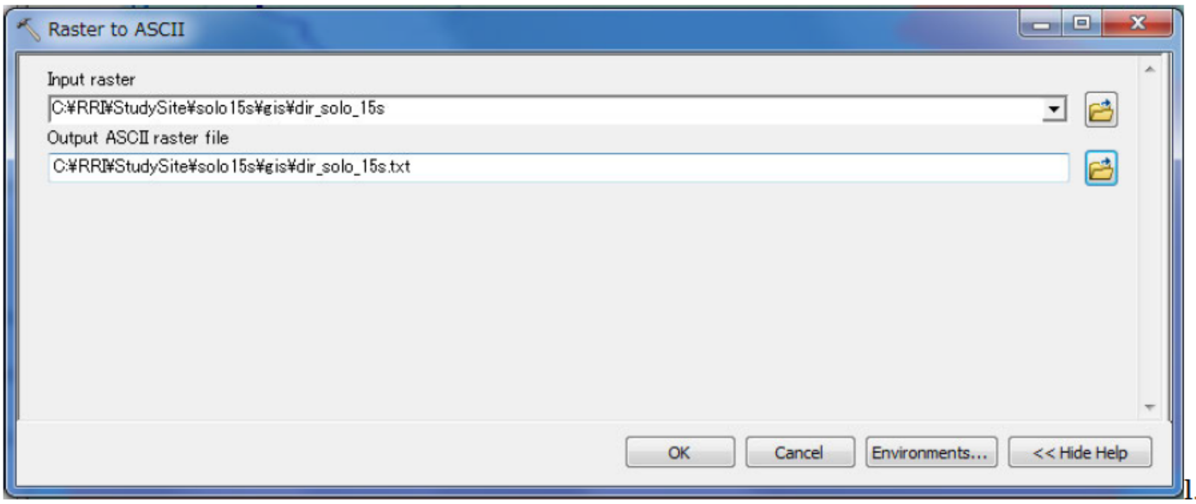


(IMPORTANT) Analysis range must be specified from the "environment" the same as above.



The above figure is the example of dem. The dir and acc must be also extracted in a same way.

⑨ Convert all the processed data (i.e. dem, dir, and acc) from ArcGIS Raster to ASCII, which are input data files for RRI Model. Using [Conversion tool] → [Conversion from Raster] → [Raster to ASCII], perform conversion from raster to ASCII for all the three types of topography data.



l.

The created ASCII data have the following format. Make sure once again all the three datasets have the same numbers in “ncols” and “nrows”.

```
ncols          673
nrows           409
xllcorner       110.2
yllcorner       -8.3
cellsize        0.004166666667
NODATA_value   -9999
 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
```

dir_solo_15s.txt

In the RRI model, the following three data must be prepared on the ASCII data format.

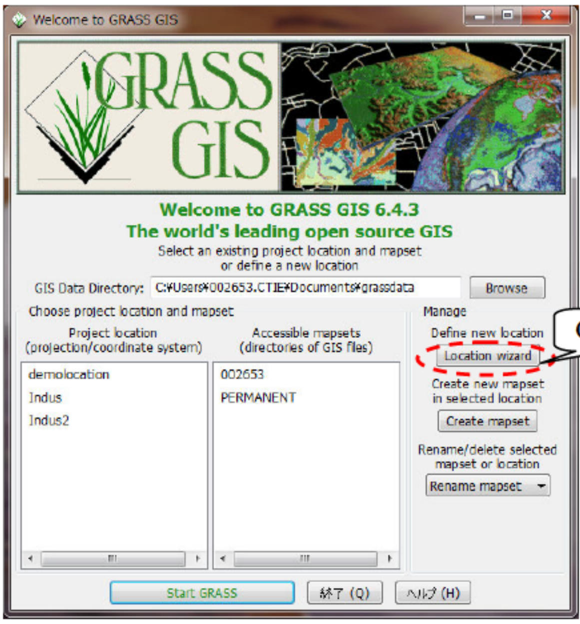
- DEM data (dem)
- Flow accumulation data (acc)
- Flow direction data (dir)

3.3 Delineating HydroSHEDS Data using GLASS GIS (optional)

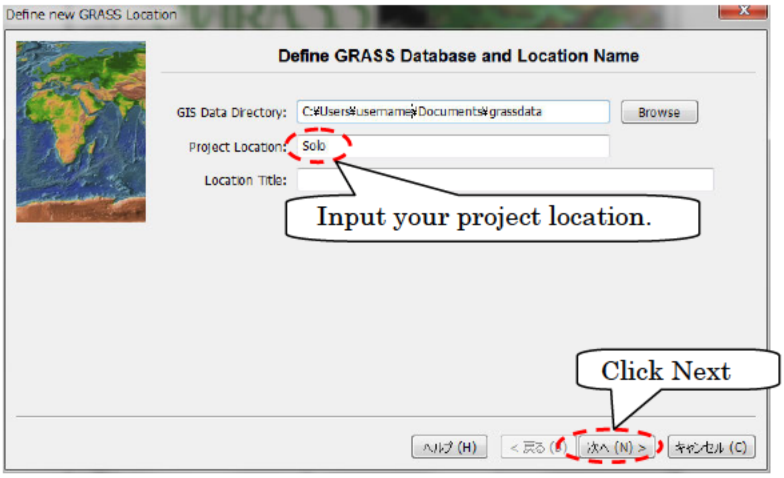
(If the HydroSHEDS data delineation is completed with ArcGIS, skip this section.)

① Install the latest GRASS GIS (Latest GRASS in December 2013 is ver 6.4.3.) (GRASS website: http://grass.osgeo.org/).

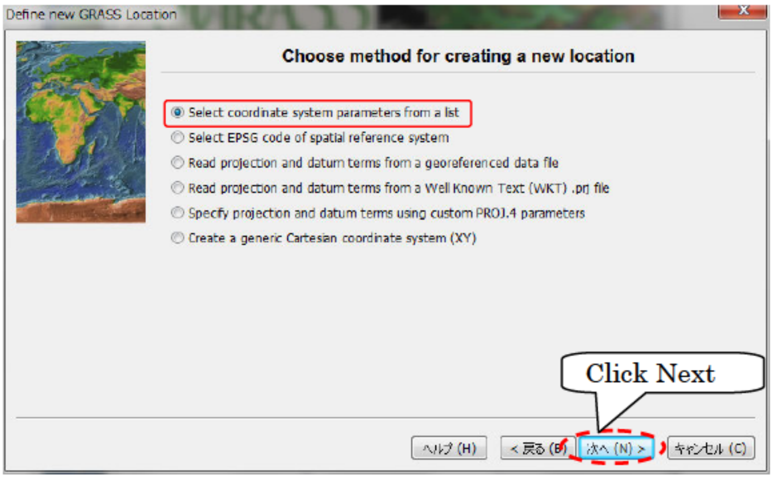
② Start GRASS GIS GUI, and click "Location wizard".

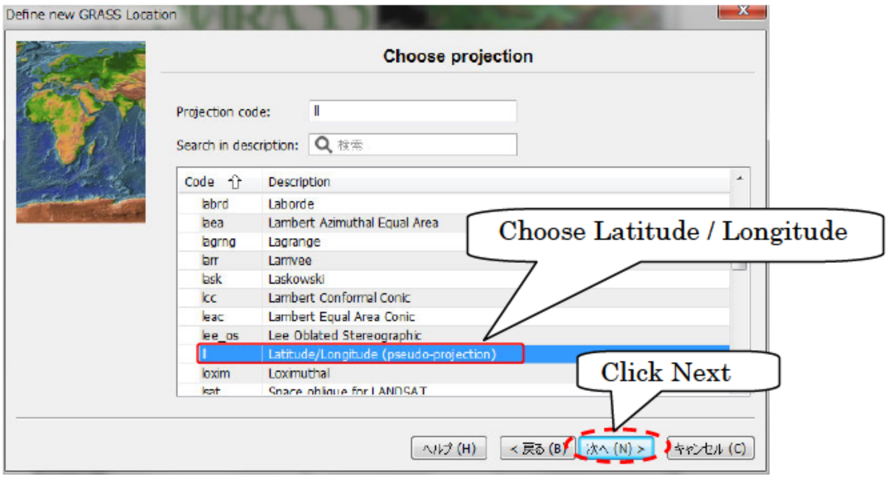


③ Input your location name (e.g. Solo) and Click next.

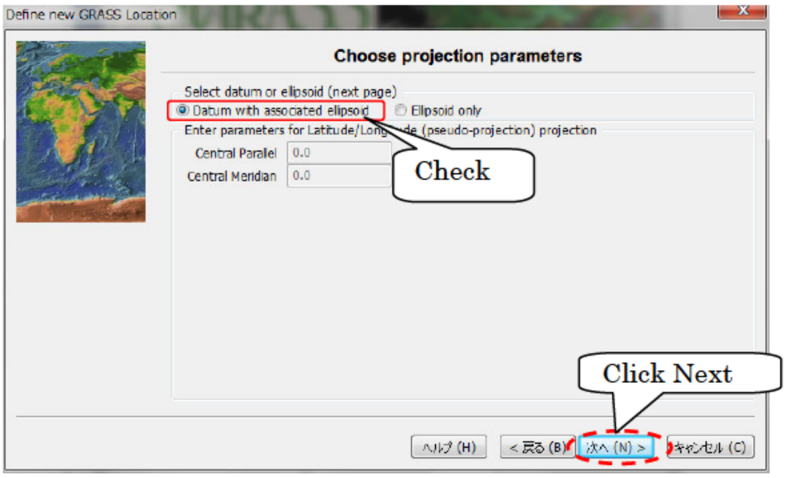


④ Select "Select coordinate system parameters from a list" and "Latitude/longitude (Pseudo-projection)" as a projection.

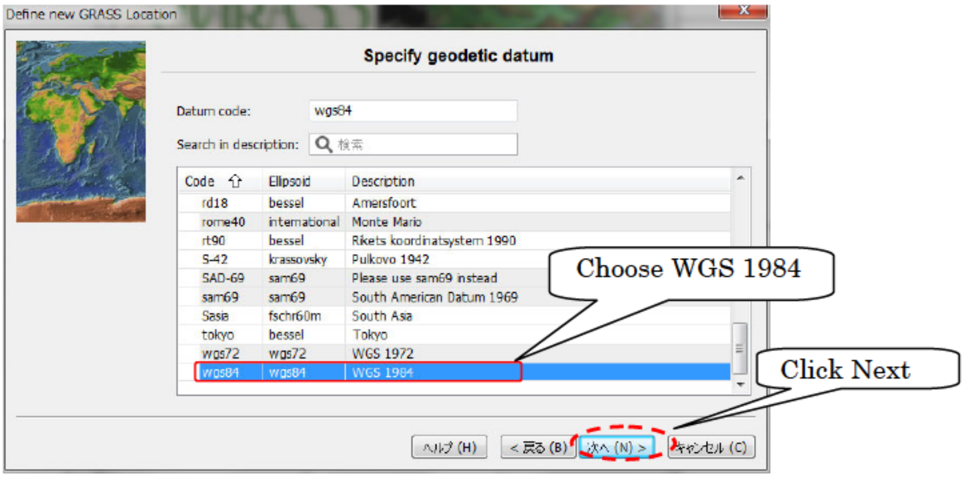




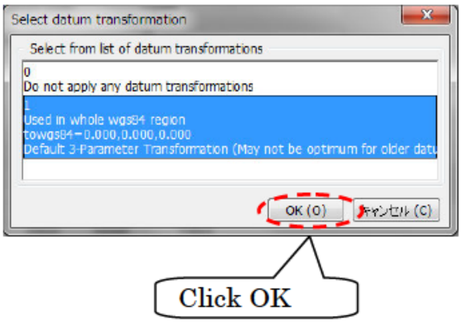
⑤ Check “Datum with associated ellipsoid” and click “NEXT“.

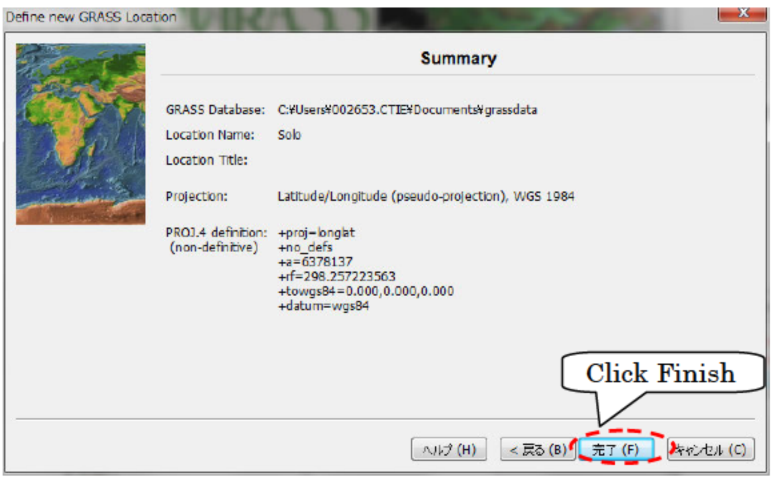


⑥ Select “WGS 1984” and as a geodetic datum and click “NEXT”.

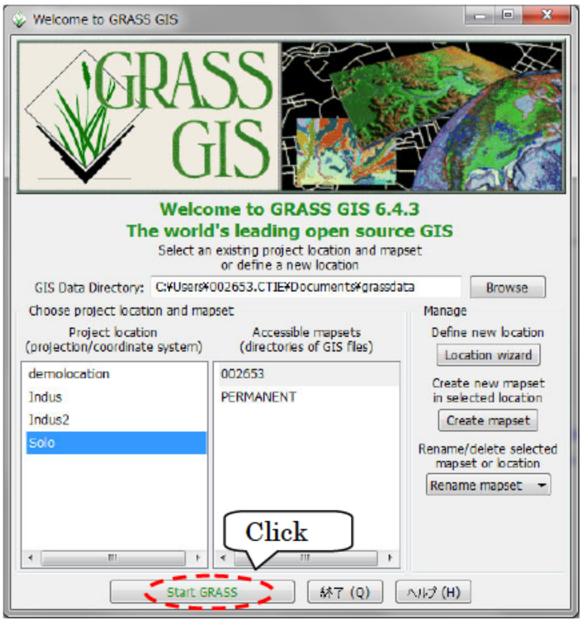


⑦ Click “OK” on “Select datum transformation” window and click “FINISH” on Summary window. (Select “Cancel” for default resolution setting).

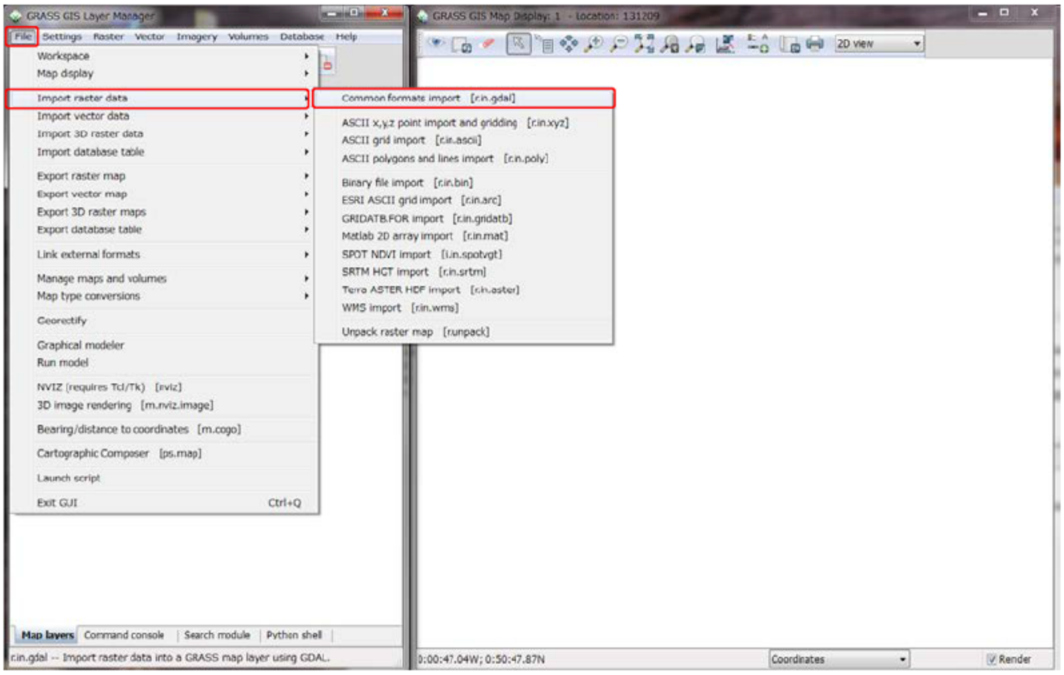




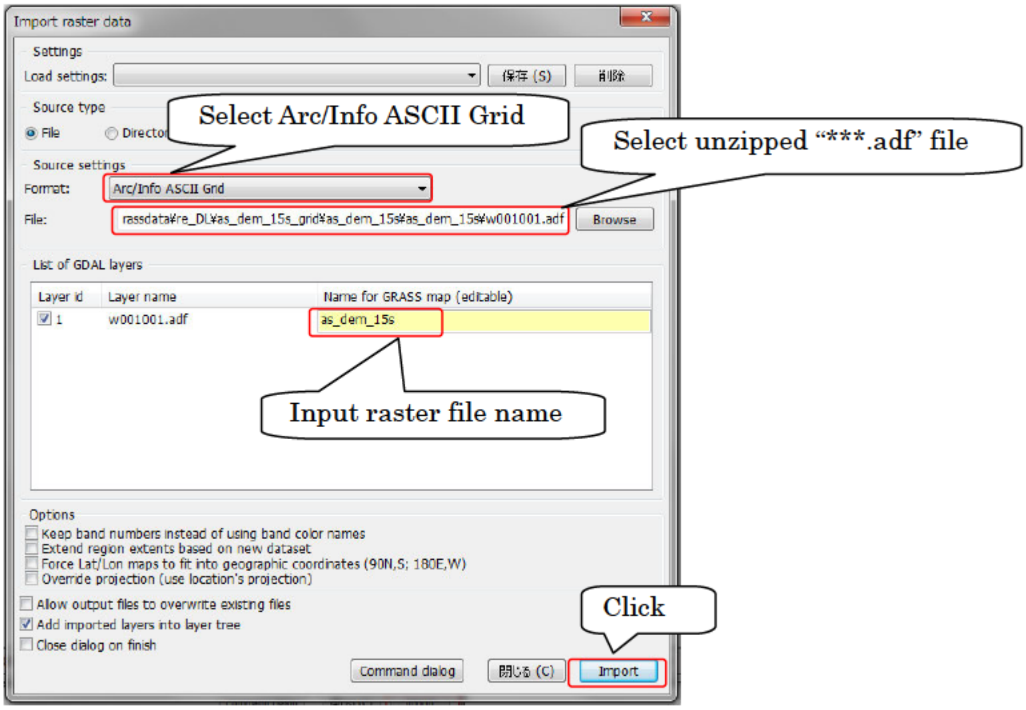
⑧ Click “Start GRASS” to start GRASS GIS.



⑨ Read in the unzipped files by selecting [File]＞[Import raster data] ＞[Common formats import].

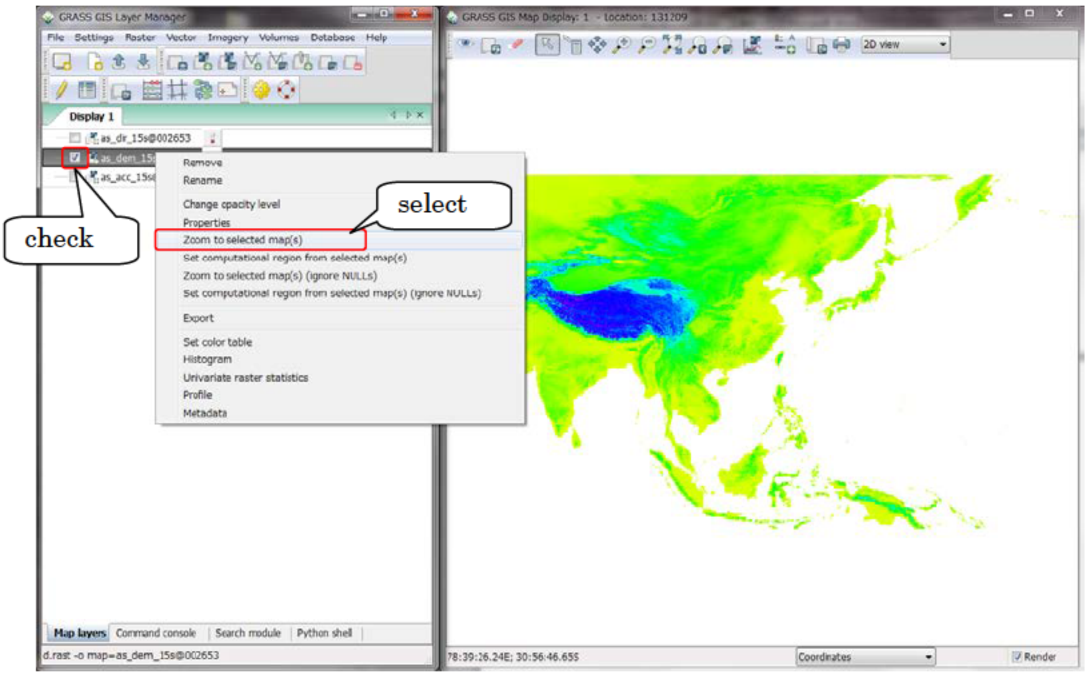


⑩ Select “Arc/Info ASCII Grid” from the “Format” list and select unzipped HydroSHEDS raster file name (e.g. w001001.adf for dem). Input “Name for GRASS map (editable)” as “as_dem_15s” for example and click Import.

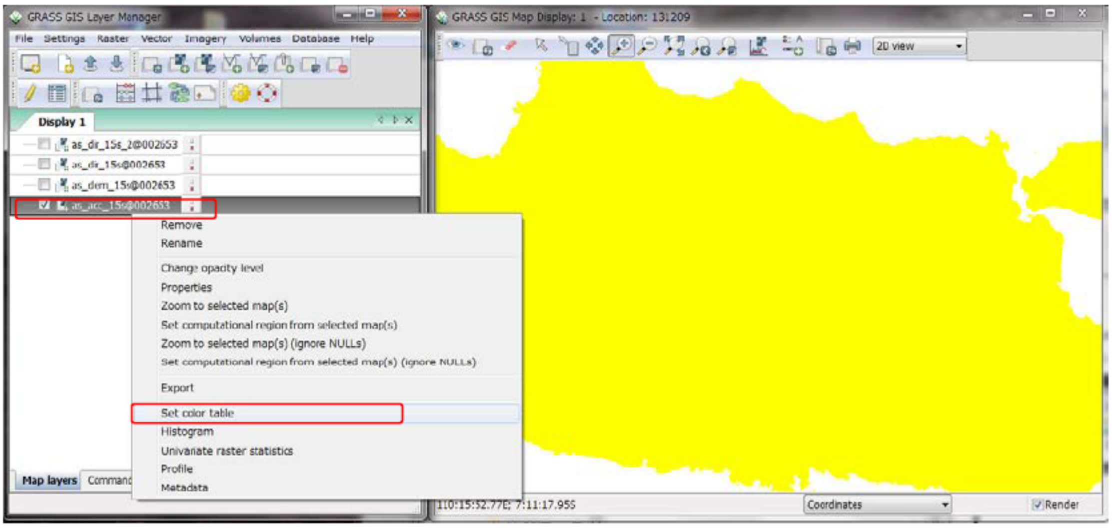


⑪ Perform the same operation for all the three types (dem, dir, acc) of topography data.

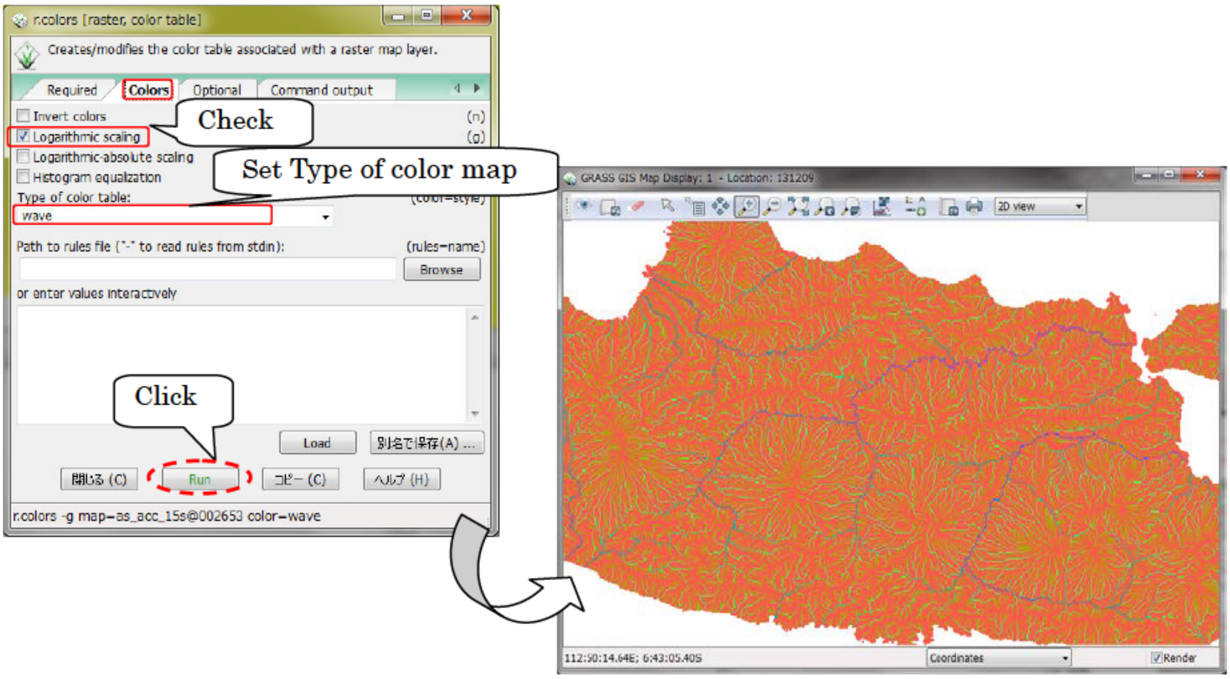
⑫ After importing three types of topography data, check the layer and right click on it and "select zoom to selected map(s)", then the raster file will be displayed in the window. (the following figure shows the example of "dem" display)



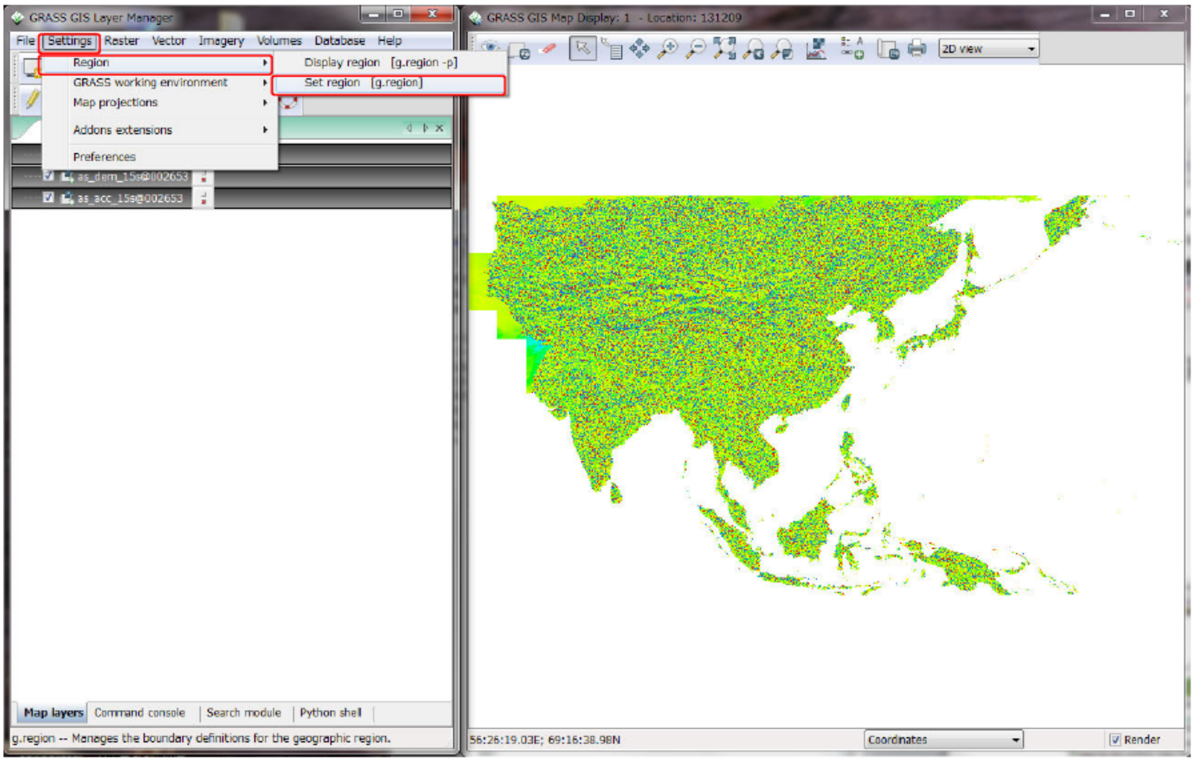
⑬ To show the flow accumulation (acc) clearly, right-click the filename of "acc" and select "Set color table".



⑭ Check "Logarithmic scaling" on "Colors" tab and select "Type of color map". User can select color table from several color tables. Following figure shows the example selecting "wave" as "Type of color table".

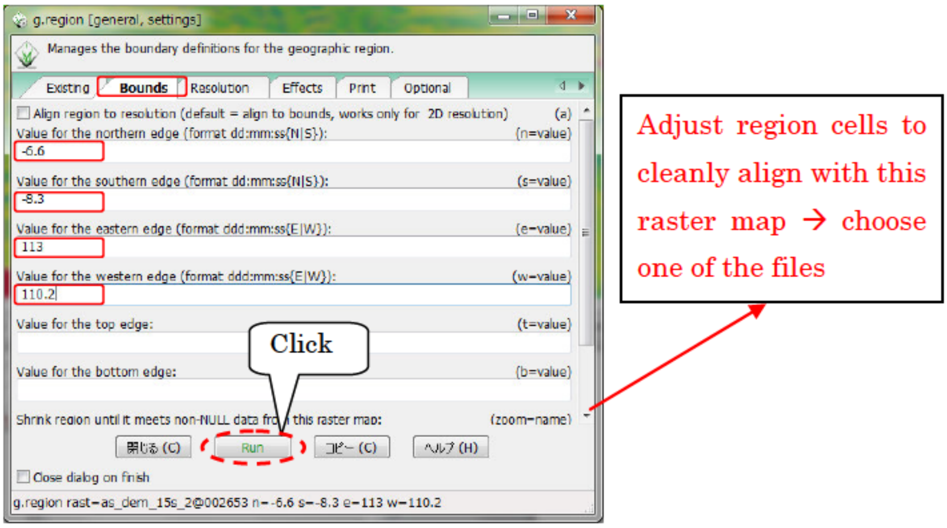


⑮ To set the delineation range, select [Settings] > [Region] > [Set Region].

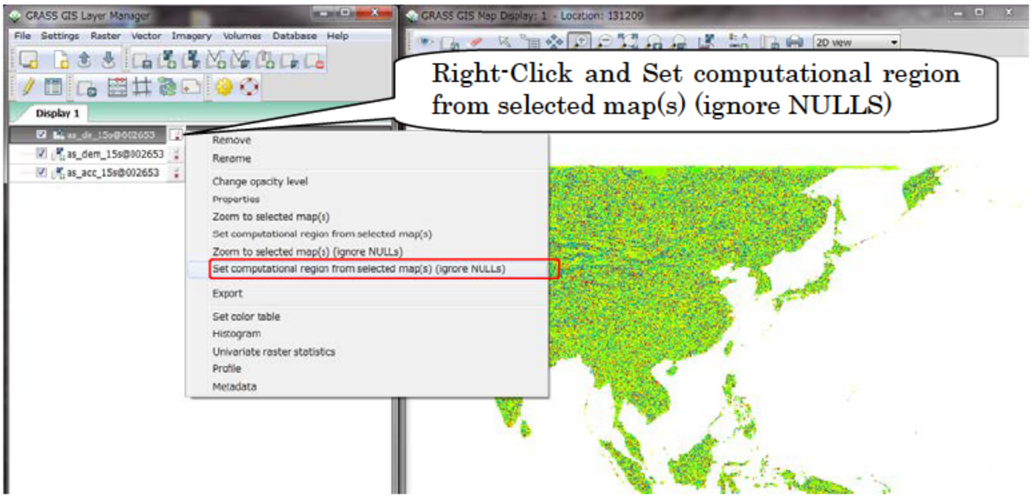


⑯ Input values for edges of the target area (coordinates) and set a file for adjusting region cells to cleanly align with a raster map, then click "Run".

(To decide your target area, display the flow accumulation data (i.e. as_acc_15s) on top screen to find your target river. The set rectangle range must cover all upstream contributing area.)

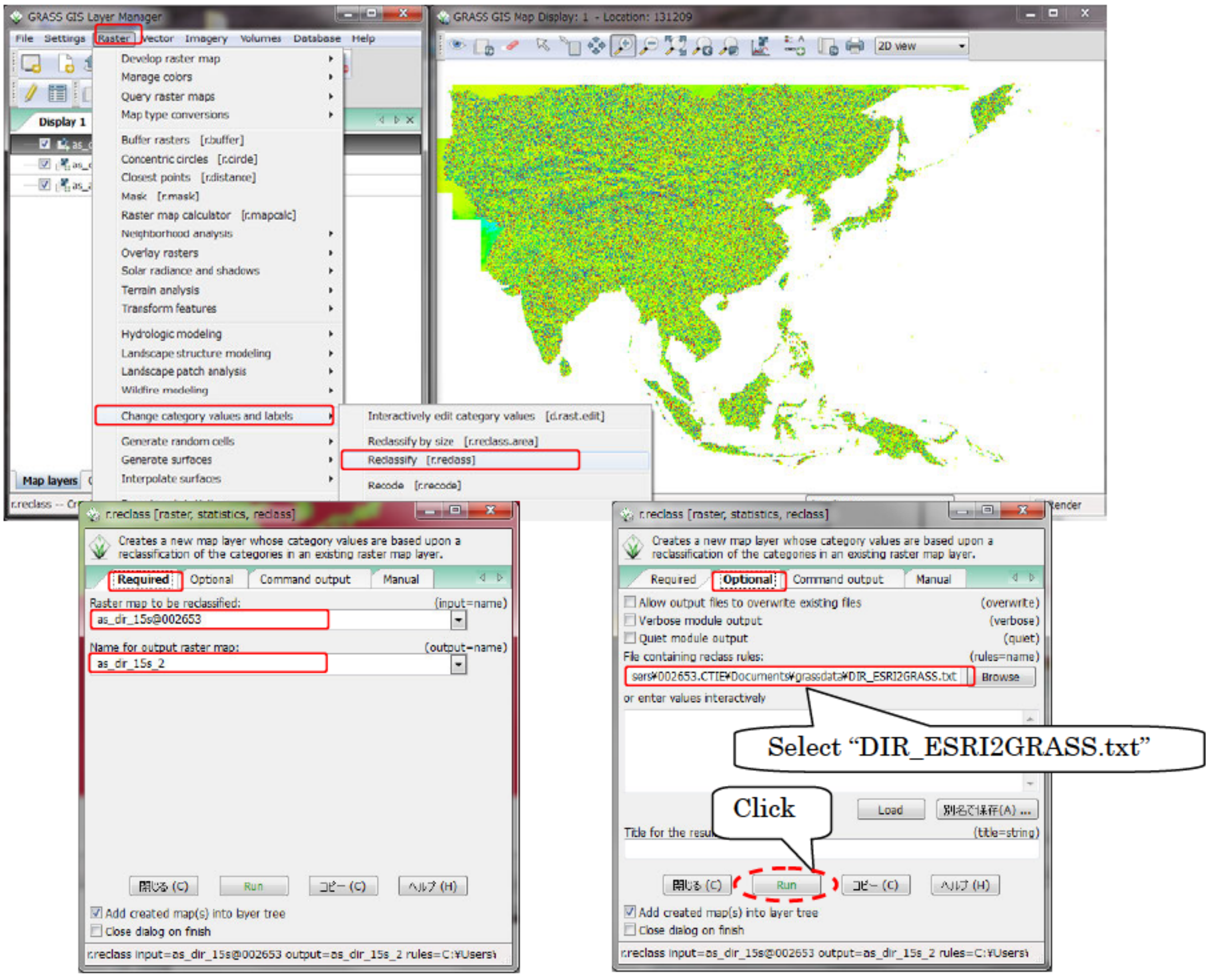


Right-click the filename of “dir” file and select “Set computational region from selected map(s) (ignore NULLS)”. Perform the same operation for all the three types (dem, acc and dir) of topography data.

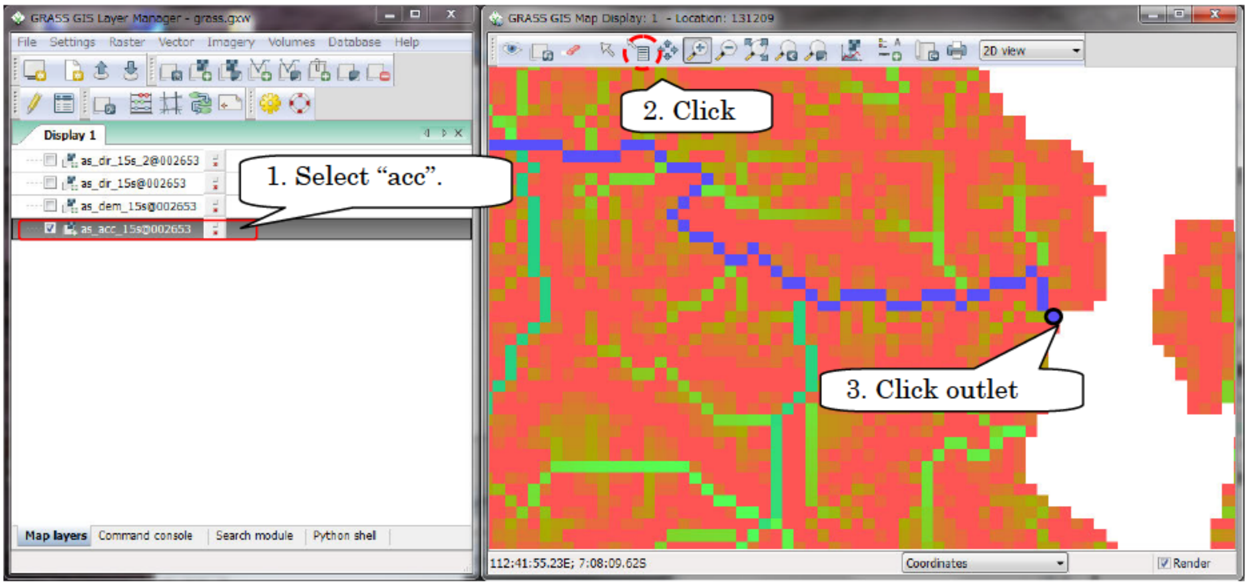


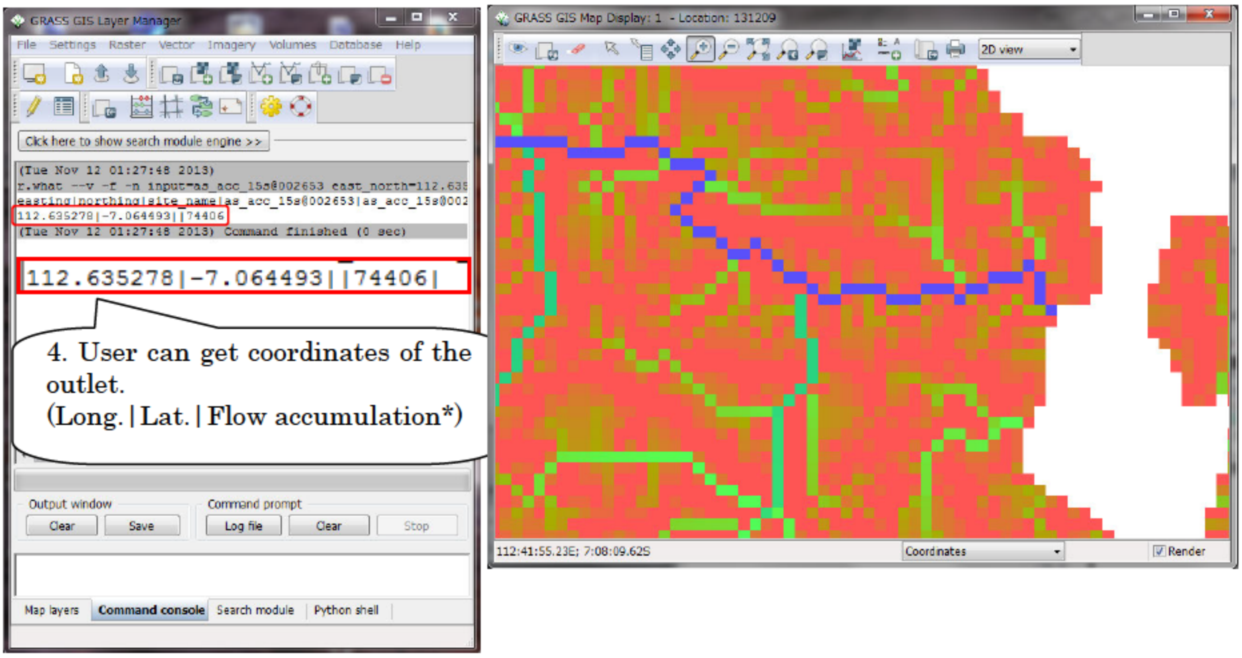
⑰ Only for flow direction, change category values (definition of river flow direction in DIR file), from ESRI type (1, 2, 4, 8, 16, 32, 64, 128) to GRASS type (1, 2, 3, 4, 5, 6, 7, 8).

Select [Raster] > [Change category values and labels] > [reclassify]: Select “DIR_ESRI2GRASS.txt”, prepared in package (/RRI/etc), as “File containing reclass rules”

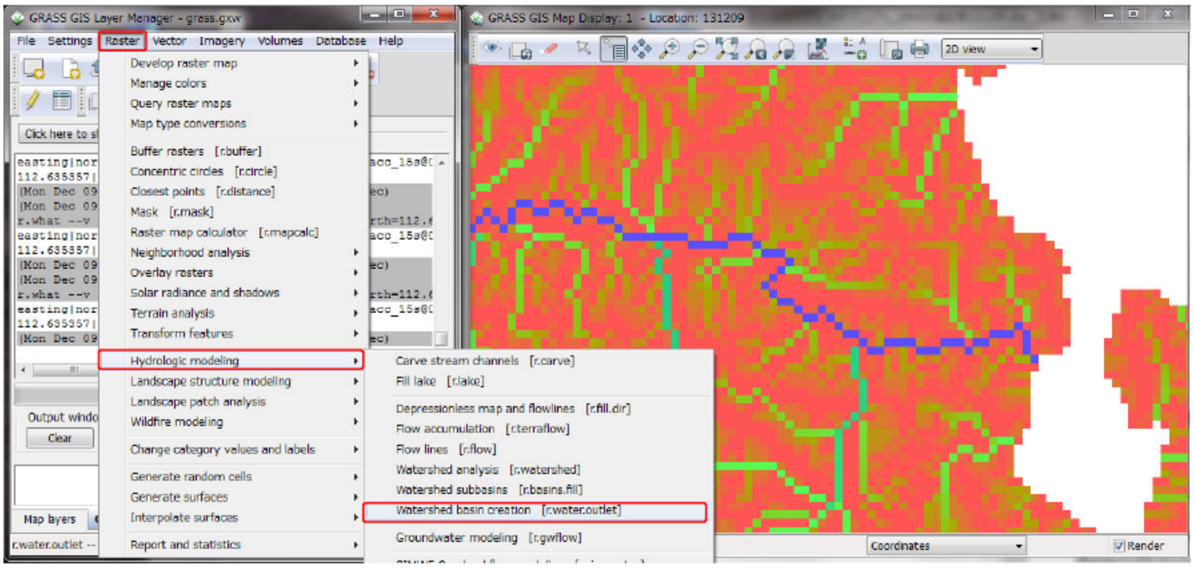


⑱ User needs to know the coordinates of the outlet (long./lat.) of target river basin to clip. Select “acc” file and perform 1, 2, 3 and 4 as shown in following figures.



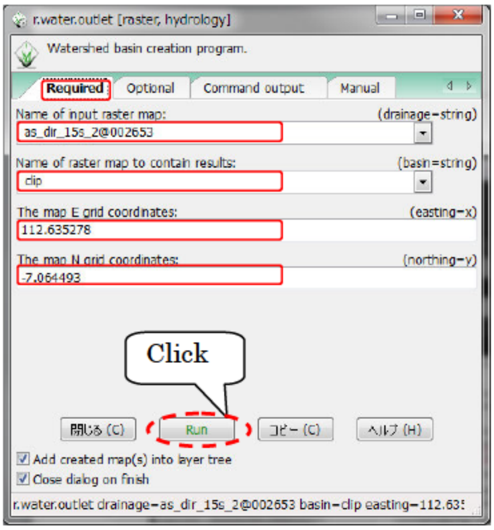


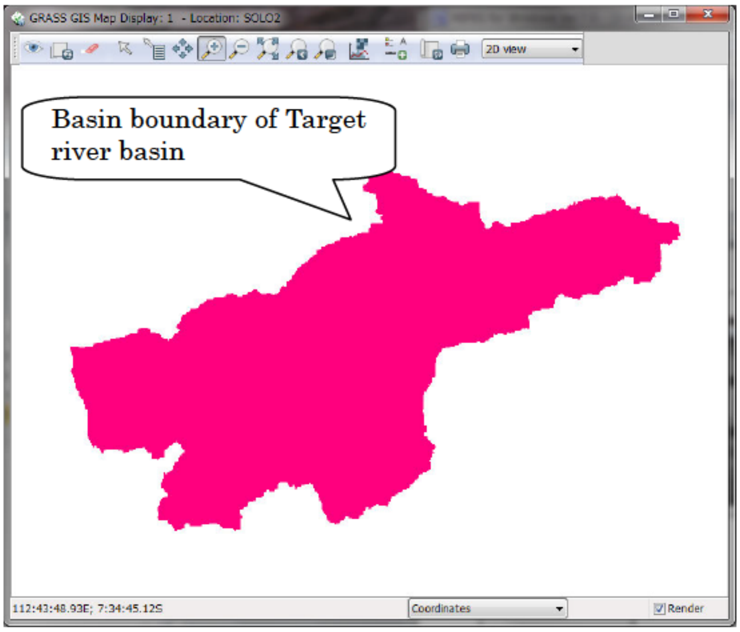
⑲ To create basin boundary, select the outlet of target river basin.
[Raster]>[Hydrologic modeling]>[Watershed basin creation (r.water.outlet)]



⑳ Select layer of "dir" file as "Name of input raster map" and input layer name of basin boundary data in "Name of raster map to contain results".

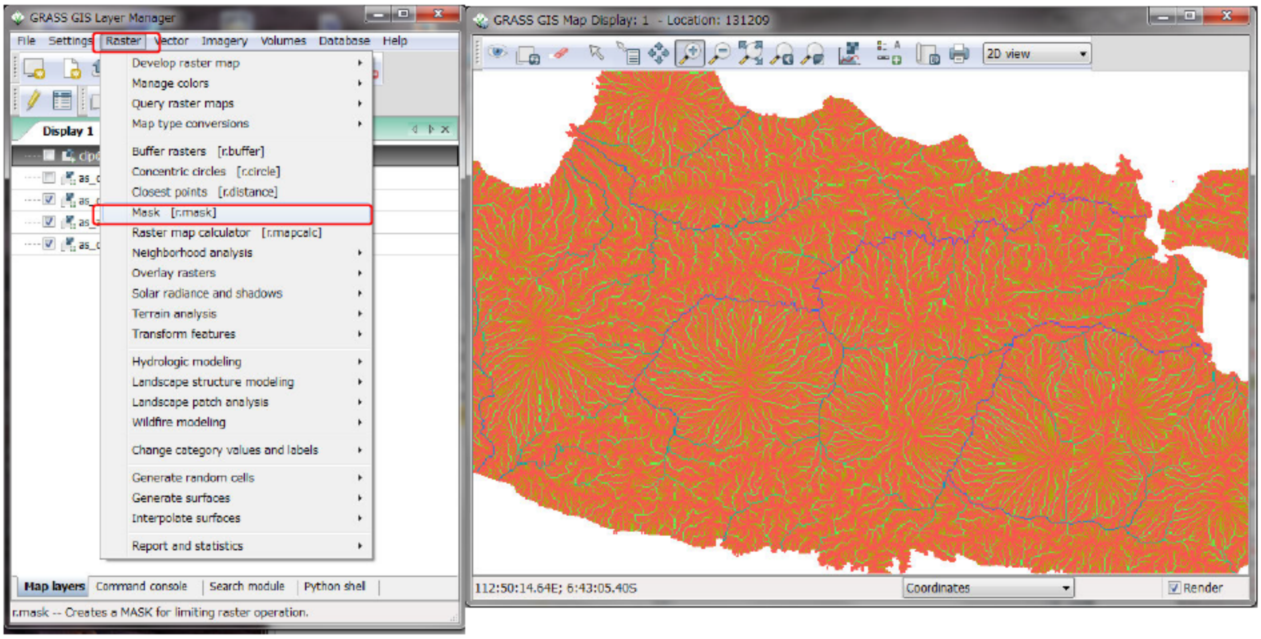
㉑ Input x-coordinate(long.) of the outlet in "The map E gird coordinates" and input y-coordinate(lat.) of the outlet in "The map N grid coordinates" and click "Run". Then, basin boundary layer of target river basin will be shown.



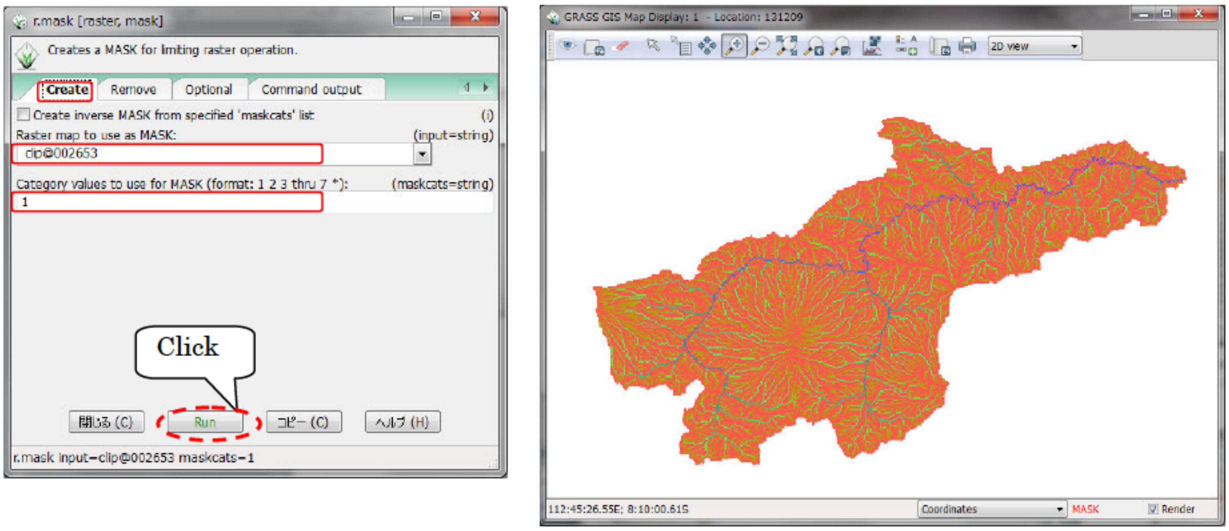


㉒ Clip target river basin by using basin boundary layer.

[Raster]>[Mask(r.mask)]

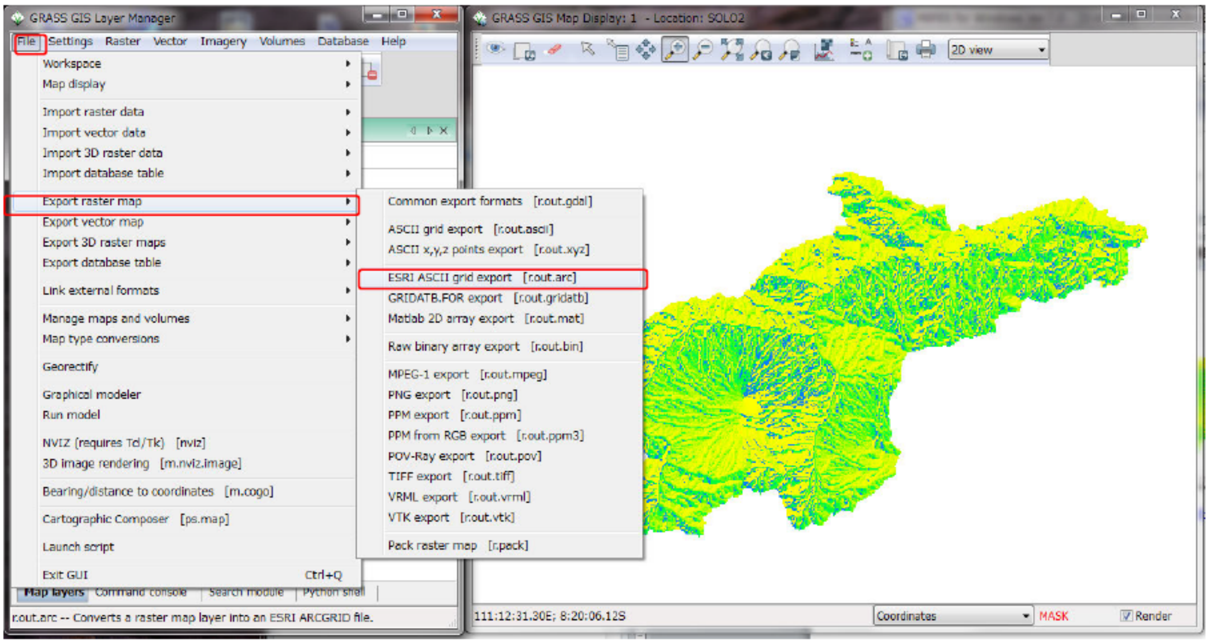


㉓ Select basin boundary layer as “Raster map to use as MASK” and input “1” in “Category values to use for MASK” on “Create” tab and click “Run”. Then, clipped target river basin will be shown.



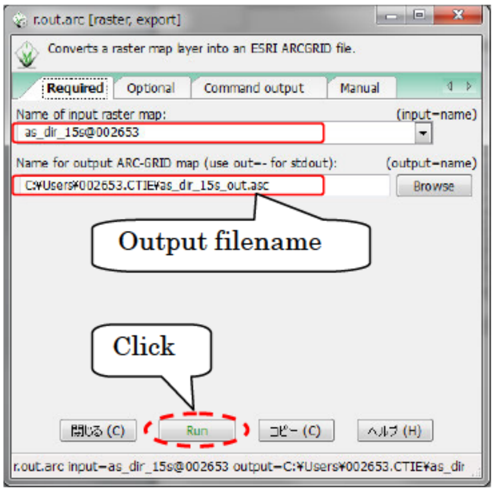
㉔ Export the three layer data (dem, dir, acc).

[File]>[Export raster map]> >[ESRI ASCII grid export]



㉕ Select three layer data (dem, dir, acc) and input output file name in "Name for output ARC-GRID map" and click "Run".

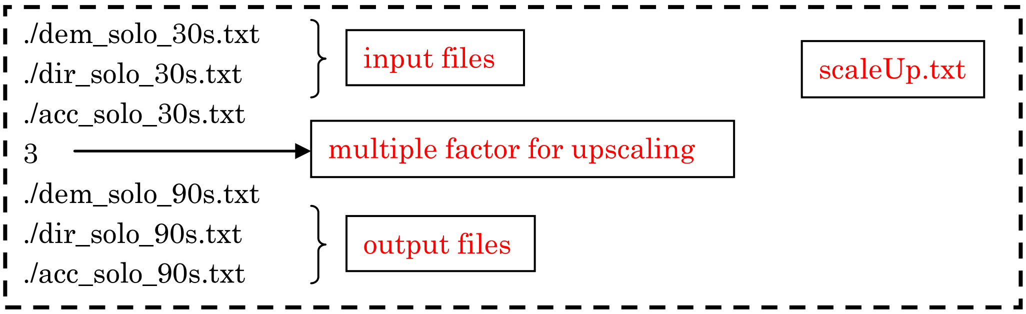
㉖ Perform the same operation for all the three layers (dem, dir, acc).



3.4 Upscaling the spatial resolutions of DEM, DIR and ACC (optional)

If a user needs to upscale the resolutions of the topography files (dem, dir and acc), one can use a program called "./etc/scaleUp/scaleUp.f90". By specifying a multiple factor for upscaling the resolution, the program outputs new dem, dir and acc based on the original topography files. For example, if the spatial resolutions of the topography files are 30 sec and the specified multiple factor is 3, the program creates the topography files having 90 sec (30s x 3). The following shows the procedure to use the program.

① Prepare "./etc/scaleUp/scaleUp.txt" file:



② Execute "./etc/scaleUp/scaleUp.exe" and find the created three sets of the topographic data indicated in L5, L6 and L7 in scaleUp.txt

3.5 DEM Data Adjustment

There are some hollows in the original HydroSHEDS elevation data. Some of them represent actual topographic features, while some of them are caused due to the intrinsic characteristics of DEM. For example, deep and narrow valley, in which a river flows, may be blocked by surrounding topography because of the DEM resolution. In that case, the simulated water depths and river discharges with the original DEM are unrealistic.

Therefore, the following DEM adjustment is always recommended to avoid the unrealistic hollows in the original DEM. The provided program called demAdjust2 (/RRI/etc/demAdjust2) follows the flow direction of HydroSHEDS and remove all the negative slope along the flow direction by carving and lifting the original DEM.

The algorithm of demAdjust2 is as follows;

1. Based on the flow direction, demAdjust2 finds upstream cells (i.e. cells with no inflow).
2. Among the detected upstream cells, searching order is determined from the total length of the flow paths from each upstream cell to its most downstream cell.
3. Following the above decided order, demAdjust2 adjusts elevations based on the following procedures.

1) The negative elevation is set to be zero.

2) Lifting: If a single cell is extremely low (likely as a noise error) compared to its upstream and downstream cells, the cell's elevation will be replaced by the same elevation as the upstream cell. The parameter "lift" is used as the threshold to detect sudden drop and its default value is set to be 500 m.

3) Carving: If the elevation suddenly increases along the flow direction, the cell's elevation will be replaced by the same elevation as the upstream cell. The parameter "carve" is used as the threshold to detect the sudden increase and its default value is 5 m.

4) Lifting and Carving: By searching from the most upstream, it finds a cell whose downstream elevation is higher than that cell (point L). By searching from point L toward downstream, it finds a cell whose downstream is lower than that cell (point H). The point L is lifted and point H is carved by the parameter "increment", whose default is 0.01 m.

The demAdjust2 program conducts each of the above procedure repeatedly for each flow path ways from all the detected upstream cells until all negative slopes are removed. Note that the above procedure does not change flow direction.

Run demAdjustment2 program in /RRI/etc/demAdjust2.

The process is necessary even if a user would like to use original dem data. "demAdjust2" program modifies not only "dem" data but also flow direction data "dir". The modified "dir" (named as "adir") has flow direction equals to zero at outlet cells. This operation must be done and "adir" always must be used for RRI simulation. Also note that there is no correction for "acc", so use the original "acc" regardless the demAdjust2 procedure.

Read the adjusted dem and dir data to ArcGIS to visualize the data.

Select [ArcToolBox] > [Conversion tool] > [Conversion from raster] > [ASCII→Raster].

"adem", "adir", "acc" are the three important topography data for the RRI simulation.

4. Preparing Input Rainfall Data

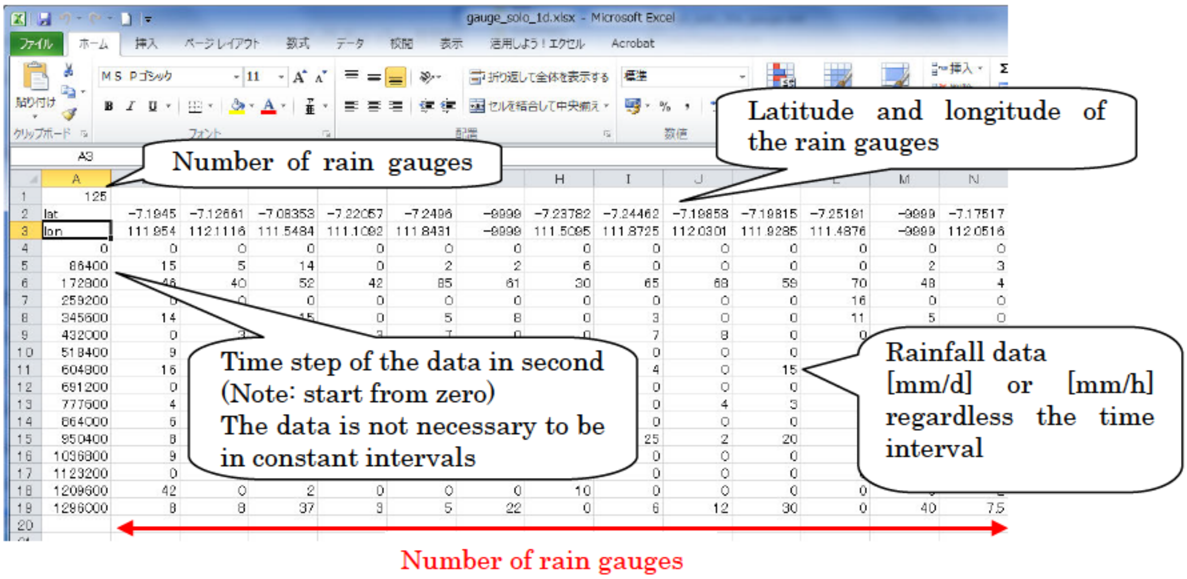
This section explains the method to prepare rainfall data for RRI Model. A user can prepare the data by any method as far as it follows a specified data format. Currently three program sets are prepared for processing:

1) gauged rainfall with Thiessen polygon interpolation (/etc/rainThiessen),
2) GSMaP satellite based rainfall (/etc/GSMaP) and
3) 3B42RT (/etc/3B42RT) satellite based rainfall.

4.1 Prepare Input Rainfall Data from Gauged Rainfall Records

To use ground gauged data for creating input rainfall for the RRI simulation, one can use /RRI/etc/rainThiessen/rainThiessen.f90 program.

① First, prepare rain gauge data in Excel (e.g. gauge_solo_1d.xlsx).



Set any negative value (e.g. -999) for missing data, not to be used for the interpolation.

② Select all cells having values, and copy and paste on a text editor. Then save it as txt file (e.g. gauge_solo_1d.txt)

③ Edit the input file "rainThiessen.txt" as follows.

```
./solo/gauge_solo_1d.txt
24
./solo/rain_solo_gauge_30s.dat
./solo/gauge_map_solo_30s.txt
ncols    336
nrows 204
xll 110.2
yll -8.3
cellsize 0.0083333333333333
```

1D rainfall data prepared above

Divide parameter
set 1 if the input rain data is in [mm/h],
set 24 if it is in [mm/d]

Output file names
L3 : output rainfall (i.e. input for RRI) [mm/h]
L4 : output map file (to check the spatial distribution of rain gauges)

Coordinate specifications of the output rainfall file. One can copy and paste the header of a topographic file (i.e. dem).

However any coordinates values and cellsize can be specified as far as the same coordinate is set in “RRI_Input.txt” and the range covers an entire domain. See the detail in Section 8.2.

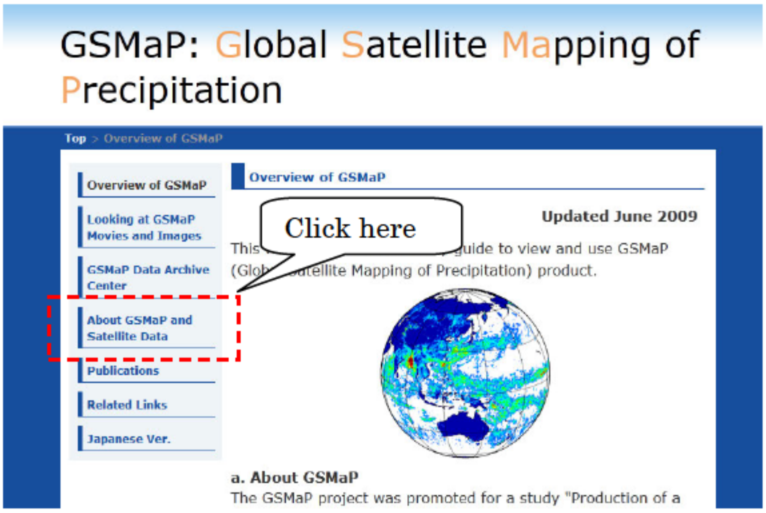
④ Run rainThiessen program.

⑤ The created output file (e.g. “rain_solo_30s_gauge.dat”) can be copied to /RRI/Model/input/ and used as the input rainfall.

4.2 Prepare Input Rainfall Data from GSMaP

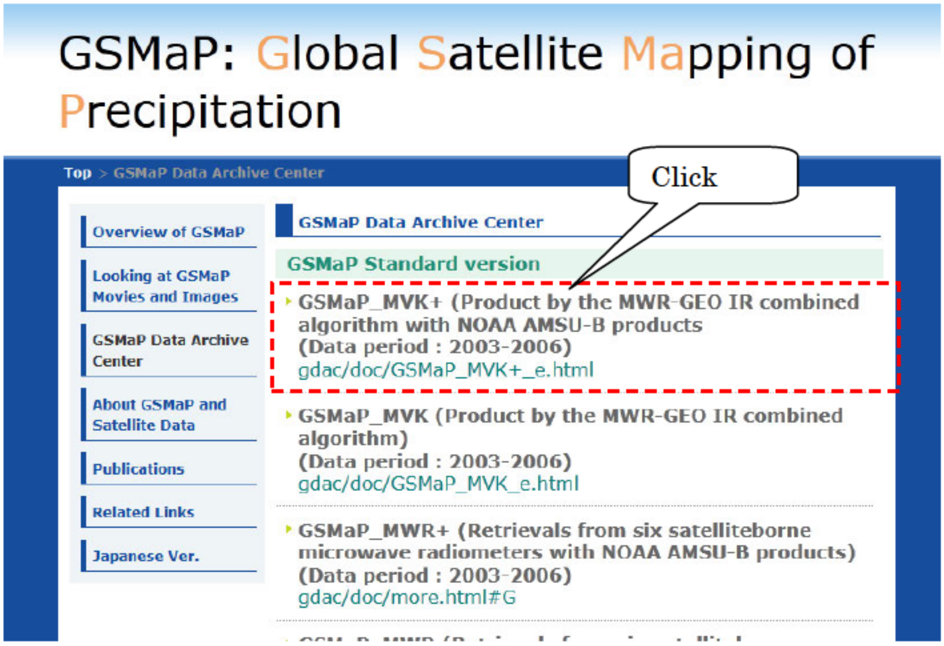
4.2.1 Download GSMaP Data

① Open the following GSMaP web site with a web browser, and click on the link to the GSMaP Data Archive Center.



(http://sharaku.eorc.jaxa.jp/GSMaP_crest/index.html)

② Click on “GSMaP_MVK+” as the data to be downloaded.



Then choose hourly or daily rainfall data to download. Note that this link allows you to download the data only between 2003 and 2006. For the data after 2006, you must register through the same website to obtain the password and download GSMaP NRT product.

4.2.2 Calculate Rainfall Data Range for a Target Catchment

To calculate the suitable range for the delineation, /etc/GSMaP/calc_area/calc_area.f90 program can be used. Before using /etc/GSMaP/calc_area.f90 program, the following "ncols" to "cellsize" must be specified in the "calc_area.txt" suitable for the target catchment. These parameters can be found from the headers of topographic files of "dem", "acc" or"dir" prepared for the target catchment.

Note: The spatial resolutions of original GSMaP are different between hourly (0.1 deg) and daily (0.25) data. Please choose appropriate "cell_size_rain" specified in "calc_area.f90" before running the program. Remember to compile the program if it is changed.

After running the program, the following information can be obtained and also output to "out_by_calc_area.txt".

out_by_calc_area.txt

```
xll       :        110.2000
yll       :        -8.300000
xur       :        113.0000
yur       :        -6.600000

xll_rain          :     110.1250
yll_rain          :    -8.375000
xur_rain          :     113.1250
yur_rain          :    -6.375000

jleft     :        440
ibottom   :        273
jright    :        452
itop      :        265

xllcorner_rain (raster)   :       110.0000
yllcorner_rain (raster)   :       -8.500000
cellsize_rain             :       0.2500000
```

Information that is necessary to delineate GSMaP (jleft, ibottom, iright, itop)

Rainfall location information to be specified in the RRI_Input.txt (xllcorner, yllcorner, cellsize)

4.2.3 Delineating GSMaP Data for Target Area

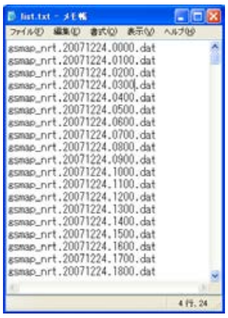
① First, save GSMaP rainfall data in a folder (e.g. infile).

② Execute the bat file "/etc/GSMaP/makeList.bat" to prepare a file list named as "list.txt"

```
dir /b infile > list.bat
```

Revise the pass, then save and execute.

When executed, [list.txt] is created



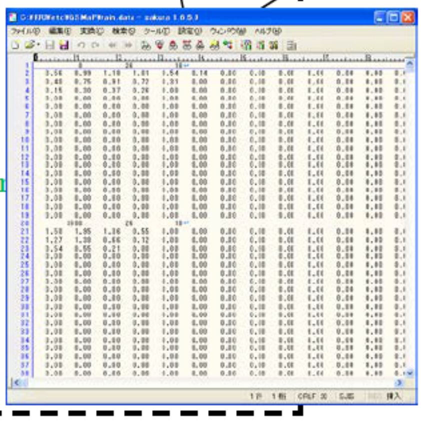
③ After revising the following, compile and execute [read_GSMaP_0.1deg.f90.]

read_GSMaP_0.1deg.f90

```
! Read GSMaP binary data and convert it to ASCII format,
! which can be read by RRI Model
!
! list.txt file must be prepared in advance by using make.bat
!
character*256 listfile
parameter( listfile = "list.txt" )

! input folder
character*256 infile1, infile
parameter( infile1 = "./infile/" )
character*256 cutfile1, cutfile
parameter( cutfile1 = "./cutfile/" )
character*256 rainfile
parameter( rainfile = "rain.data" )
character*256 sumfile
parameter( sumfile = "sumRain.data" )

! Target Area
integer jleft, ibottom, jright, itop

! The range to delineate
! The values can be calculated by calc_area.f90 program
! for a target catchment
parameter( jleft    = 440 )
parameter( ibottom = 273 )
parameter( jright   = 452 )
parameter( itop     = 265 )
```

Designate the path of the rainfall data folder

Running the program creates "rain.data", which is rainfall input file for RRI simulation



Designate the delineation range from calcArea.exe

4.3 Prepare Input Rainfall Data from 3B42RT

4.3.1 Download 3B42RT Data

① Access the following FTP site for downloading 3B42RT data

ftp://trmmopen.gsfc.nasa.gov/pub/merged

② Download "3B42RT.20*****.7R2.bin.gz" files from the FTP site and save them under ./etc/3B42RT/read/infile/

4.3.2 Calculate Rainfall Data Range for a Target Catchment

To calculate the suitable range for the delineation, /etc/3B42RT/calc_area.f90 program can be used. See details in 4.2.2, the same process is used for GSMaP data extraction.

4.3.3 Delineating 3B42RT Data for Target Area

① The following process uses “bash script”. Windows users may install “clink” program to run bash scripts (*.sh) on windows command prompt. The “clink” program can be downloaded from: http://code.google.com/p/clink/

② To calculate the suitable range for the delineation, /etc/3B42RT/calc_area.f90 program can be used. See details in 4.2.2 because the same process is applied also to GSMaP data extraction.

③ Execute "bash unzip.sh" in /etc/3B42RT/read/ to unzip the downloaded files under ./etc/3B42RT/read/infile.

④ Edit "read_rt_file.sh" file to set extraction range in L4 to L7 (jleft, ibottom, jright, itop) suggested by calc_area.f90.

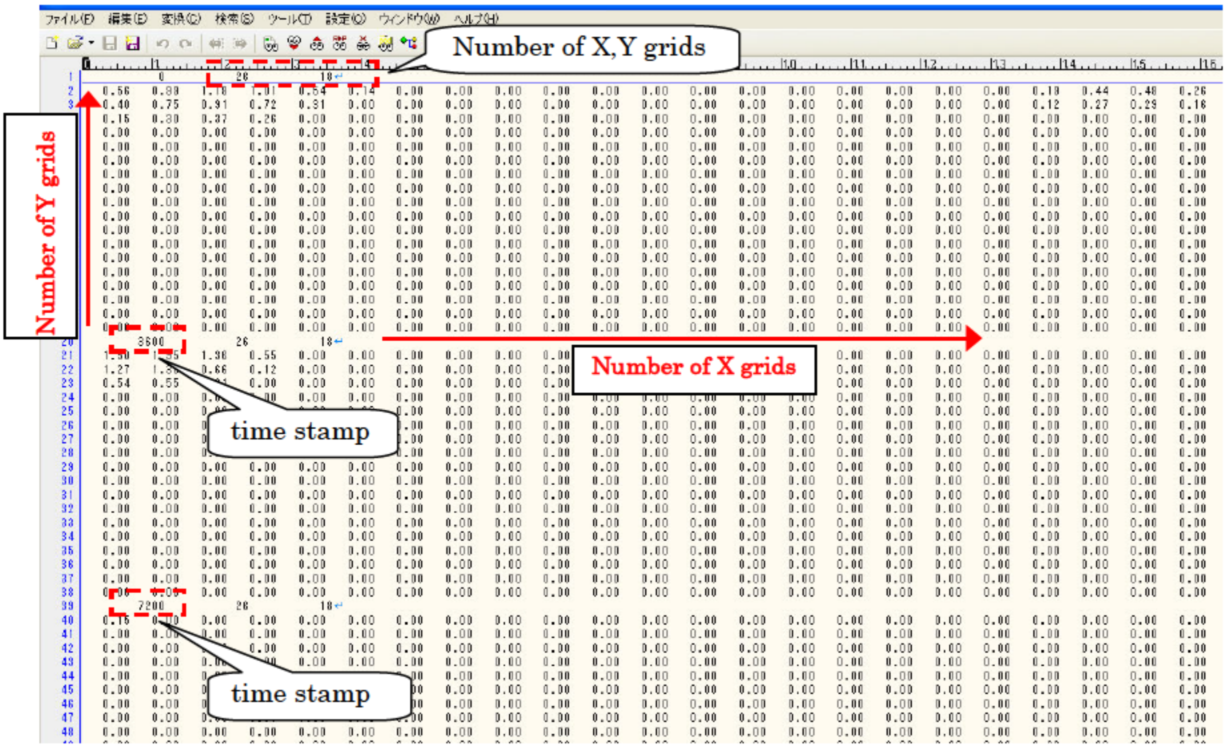
⑤ Execute "bash read_rt_file.sh" to extract data
Note: the extract does not run if the same output files already exist

⑥ Edit "combine.sh" by setting the extraction range in L4 to L7 (jleft, ibottom, jright, itop) suggested by "out_by_calc_area.txt", and set output file name on L9. Also edit L14, L17 and L20 to indicate which year, month and day of the data should be processed.

⑦ Execute "bash ./combine.sh" to combine all rainfall files to create the RRI input, so that the rainfall file, which can be read by the RRI program, will be created.

4.4 Format of Input Rainfall Data for RRI Model

Here is the format of the input rainfall data used for RRI Model. By specifying the cell size, xll_corner and yll_corner of the rainfall data into a control file of RRI model (i.e. “RRI_Input.txt”), the model can overlay the rainfall distribution even if the ranges and the resolutions are different from topographic data as far as the rainfall data covers all the simulation extent.



※ The input unit of rainfall must be always **mm/hr** regardless the data interval.
※ The time interval is not necessary to be constant.
※ Rainfall between 3600 and 7200 is written under the time stamp of 7200 (just like rain gauge data).

【RRI_Input.txt】・・・Control file of the RRI Model

RRI_Input.txt

```
RRI_Input_Format_Ver1_4_1

./infile/solo30s/rain_solo_30s_gauge.dat
./infile/solo30s/adem_solo_30s.txt
./infile/solo30s/acc_solo_30s.txt
./infile/solo30s/adir_solo_30s.txt

0                # utm(1) or latlon(0)
1                # 4-direction (0), 8-direction(1)
360              # lasth
600              # dt
60               # dt_riv
96               # outnum
110.2d0          # xllcorner_rain
-8.3d0           # yllcorner_rain
0.00833333d0 0.00833333d0    # cellsize_rain
```

Coordinates and grid size of the south-west end of the rainfall data range

4.5 Calculation of Catchment Average Rainfall

To calculate catchment average rainfall from the input rainfall data, a user can use "./etc/rainBasin.f90" program. To run the program, "rainBasin.txt" must be prepared in the following way.

rainBasin.txt

```
../../Model/infile/solo30s/rain_solo_30s_gauge.dat
../../Model/infile/solo30s/dem_solo_30s.txt
110.2d0
-8.3d0
0.00833333d0 0.00833333d0
rain_hyeto.txt
rain_dist.txt
rain_cum.txt
```

L1 : [in] rainfall file (RRI format) [mm/h]

L2 : [in] catchment mask file (e.g. dem file)

L3 : [in] rainfall xll corner

L4 : [in] rainfall yll corner

L5 : [in] rainfall cellsize (x, y)

L6 : [out] hyetograph [mm/h]

L7 : [out] total rainfall distribution map [mm]

L8 : [out] cumulative rainfall [mm]

Executing "rainBasin.exe" creates three output files identified in L6, L7 and L8 to show hyetograph, total rainfall distribution map and cumulative rainfall, respectively.

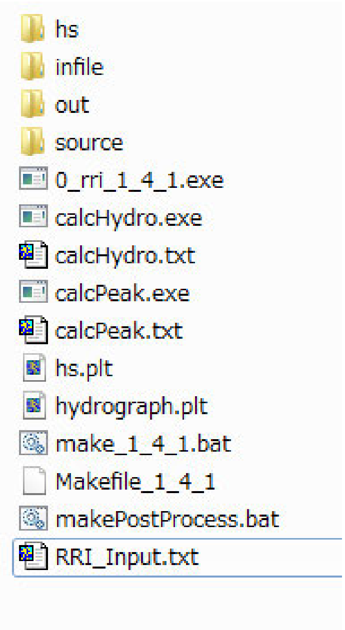
Note: To calculate average rainfall over a sub-catchment, one can replace the file indicated in L2. First, one can use GIS to delineate the sub-catchment and convert the mask into ASCII GIS format. For areas having pixel values greater than -10 will be considered as a sub-catchment area.

5. Conditions Setting for RRI Simulation

5.1 Folder Configuration

The following shows the folder configuration and roles of each folder in /RRI/Model

[/RRI/Model]



-Folders-

infile : Stores following sets of input data for each project

- Adjusted topography data (adem)
- Flow accumulation data (acc)
- Adjusted flow direction data (adir)
- Rainfall data

(optional)

- Land use data
- Initial and boundary conditions

out : Stores simulation results for each output time step

- hr_ : River water depth [m]
- hs_ : Slope water depth [m]
- qr_ : River discharge [m^3/s]
- qu_ : Slope discharge for x direction [m^3/s]
- qv_ : Slope discharge for y direction [m^3/s]
- gampt_ff : Green-Ampt cumulative water depth [m]
- storage.dat : water balance checking file

source : Fortran source programs of RRI Model

hs : Stores figures of inundation depths (hs) by gnuplot

【Important Files】

- make_1_4_1.bat : Compiling RRI model source (use Makefile_1_4_1 for UNIX/LINUX)
- 0_rri_1_4_1.exe: RRI model execution file
- RRI_Input.txt : RRI model control file

【Other Programs and Files】

- makePostProcess.bat : Compiling calcHydro.exe and calcPeak.exe
- calcHydro.exe : hydrograph calculation program (post processing)
- calcPeak.exe : peak inundation depth calculation program (post processing)
- hs.plt : gnuplot script to create inundation depths figures (prepared by /etc/prepHsPlt)

5.2 RRI Model Control File (RRI_Input.txt)

RRI_Input.txt

```
L1   RRI_Input_Format_Ver1_4_1
L2
L3   ./infile/solo30s/rain_solo_30s_gauge.dat
L4   ./infile/solo30s/adem2_solo_30s.txt
L5   ./infile/solo30s/acc_solo_30s.txt
L6   ./infile/solo30s/adir_solo_30s.txt
L7
L8   0                      # utm(1) or latlon(0)
L9   1                      # 4-direction (0), 8-direction(1)
L10  360                    # lasth [hour]
L11  600                    # dt [sec]
L12  60                     # dt_riv [sec]
L13  96                     # outnum [-]
L14  110.2d0                # xllcorner_rain
L15  -8.3d0                 # yllcorner_rain
L16  0.00833333d0 0.00833333d0     # cellsize_rain
L17
```

Note that #comment is allowed only for lines with numbers like L8 to L16, but it is not allowed for lines with characters like L3 to L6.

L1 : Version of the control file format.

This version has to be compatible with the RRI program version. When RRI Model version is updated, user may be requested to modify this control file to be suitable for the updated version.

L3 – L6 : Paths of the input files (rainfall, dem, acc, dir)

Note that adjusted direction file having zero at the outlet must be read in the flow direction column. This adjustment (for dem and dir) can be implemented through the process of demAdjust2.

L8 : Topographic and rainfall data coordinate system (UTM (1) or Lat Lon(0))

L9 : Simulating with 4- (0) or 8-direction (1) by the two dimensional model [default : 1]

L10 : Simulation period [hour]

L11 : Simulation time step [sec], [default : 600 sec]

L12 : Simulation time step for river [sec], [default : 60 sec]

The above time steps are just initial setting. The adaptive Runge-Kutta algorithm used for RRI simulation may shorten the time steps if necessary.

L13 : Number of output files

Simulation period specified above is equally divided for simulation output.

L14 – L16 : South west coordinate and resolution of rainfall data

Number of col and row are written in the rainfall data.

RRI_Input.txt

```
L18 | 0.03d0      # ns_river
L19 | 1           # num_of_landuse
L20 | 1           # diffusion(1) or kinematic(0)
L21 | 0.4d0       # ns_slope
L22 | 0.0d0       # soildepth
L23 | 0.475d0     # gammaa
L24 |
L25 | 0.d0        # ksv
L26 | 0.3163d0    # faif
L27 |
L28 | 0.0d0       # ka
L29 | 0.0d0       # gammam
L30 | 0.0d0       # beta
L31 |
```

L18 : Manning's roughness in river channel

L19 : Number of landuse

Parameter sets specified below should correspond to the number of landuse specified here. For example, if there are three landuse types in a catchment, write three different parameter sets. Prepare also the landuse map which has numbers from one to three, so that the parameter sets described below will be assigned to each landuse grid cell. First column parameters are assigned to landuse type "1" in the landuse map.

L20 : diffusion (1) or kinematic (0) [default : 1]

The default mode of RRI model uses diffusion wave equations. However, by setting zero here, RRI model can use kinematic wave approximation.

L21 : Manning's roughness on slope cells

L22 : Soil depths [m]

L23 : Effective porosity [-]

L25, L26 : Green-Ampt infiltration model parameters

Set ksv = 0 for inactivating Green-Ampt infiltration model.

“ksv” : vertical saturated hydraulic conductivity [m/s], ”faif” is the suction at the wetting front defined by S_f.

Note: In the previous versions of RRI Model, “delta” and “infilt_limit” parameters were used. The parameter “delta” is now replaced by “gamma” to represent soil porosity minus initial water volume content ($\phi - \theta_i$). The “infilt_limit” parameter is computed within the RRI program by multiplying “soildepth” and “gamma” to estimate the maximum cumulative infiltration depths in meter. Once the cumulative infiltration depths reaches to this maximum depths, no more infiltration happens at the grid-cells.

L28 – L30 : lateral subsurface and surface model parameters

L28 and L30 are options to consider unsaturated and saturated subsurface flow and surface flow in lateral direction. “ksv” is lateral saturated hydraulic conductivity (which is typically two or three orders high compared with the vertical hydraulic conductivity set for Green-Ampt model. To start with, set zero for “dm” to inactivate the option to consider unsaturated subsurface flow. Setting zero makes no saturated subsurface flow consideration. See 8.7 for the details of the parameter settings.

Note: In the previous version of RRI Model, a parameter “da” was used to represent maximum water depth in saturated subsurface flow. Now this is calculated as “soil depth” times “gamma” within the program.

L32 – L36

Set “kgv = 0.d0” to avoid groundwater computation, whose algorithm is under development and not completed at RRI ver1.4.1β. L33-L36 become inactive with kgv = 0.d0.

L38 – L44 : River channel geometry setting by equations

$$width = c_w A^{S_w}$$

$$depth = c_d A^{S_d}$$

The above equations are used as default settings for river channel widths and depths.
Note that A in the equations is the upstream catchment area [km^2] for each river grid-cell.

L46 – L49 : River channel geometry setting by files (optional)

If one would like to set width, depth and embankment height from files instead of the above equations, set 1 in L46 and prepare the files in ESRI/ASCII format.

```
L38 20               # riv_thresh                  RRI_Input.txt
L39 5.0d0            # width_param_c
L40 0.35d0           # width_param_s
L41 0.95d0           # depth_param_c
L42 0.20d0           # depth_param_s
L43 0.d0             # height_param
L44 20               # height_limit_param
L45
L46 0
L47 ./infile/solo/width_solo.txt
L48 ./infile/solo/depth_solo.txt
L49 ./infile/solo/height_solo.txt
L50
```

L51 – L55 : Initial water depth on slope, river, groundwater and GA Model cumulative by files (optional)

If one would like to set initial water depths on slope and river for each grid cell, set 1 in L51 and prepare the initial condition distribution files specified in L52, L53, L54 and L55. Note that the format of the files is the same as RRI model output.

```
L51 0 0 0 0                                            RRI_Input.txt
L52 ./infile/solo30s/hs_init_dummy.out
L53 ./infile/solo30s/hr_init_dummy.out
L54 ./infile/solo30s/hg_init_dummy.out
L55 ./infile/solo30s/gampt_ff_init_dummy.out
L56
L57 0 0
L58 ./infile/hs_wlev_bound_dummy.txt
L59 ./infile/hr_wlev_bound_dummy.txt
L60
L61 0 0
L62 ./infile/hs_disc_bound_dummy.txt
L63 ./infile/hr_disc_bound_dummy.txt
L64
```

L57 – L59 : Water depths boundary conditions (optional)

L57 : Slope water depths boundary conditions, L58 : River water depths boundary conditions

See Section 8 for the format of the boundary condition files. Use flag 1 for one-dimensional data format (i.e. time series data at specific boundary condition locations). Use flag 2 in case the boundary condition files are prepared in two-dimensional data format, whose number of grid-cells must be the same as the topographic data including dem, dir, and acc. In both cases, time stamps in the boundary condition can vary within the file.

L61 – L63 : Water discharge boundary conditions (optional)
(Same as L57 – L59)

RRI_Input.txt

L65	0
L66	./infile/solo/landuse_solo.txt
L67	
L68	0
L69	./infile/damcnt_ALL.txt
L70	
L71	0
L72	./infile/div_dummy.txt
L73	
L74	0
L75	./infile/potentialET.txt
L76	110.2d0 # xllcorner_evp
L77	-8.3d0 # yllcorner_evp
L78	0.00833333d0 0.00833333d0 # cellsize_rain
L79	
L80	0
L81	./infile/solo30s/section.txt
L82	
L83	0
L84	./infile/solo30s/sec_map.txt
L85	./infile/solo30s/section/sec_
L86	

L65 – L66 : Landuse setting (optional)
If one would like to use multiple parameter sets for different grid-cells, set 1 in L65 and read landuse file specified in L66.

L68 – L69 : Dam condition setting (optional)
RRI model simulates the effect of dam reservoir operations based on simple rule. Refer to the source code "RRI_Dam.f90" for details. (See also 8.11)

L71 – L72 : River diversion setting (optional)
River channel diversion setting (See also 8.10)

L74 – L78 : Evapotranspiration setting (optional)

Prepare ET file and specify the path on L74. The format of ET file is the same as rainfall. The resolution and xll and yll corners can be different from the rainfall file as far as it covers all the simulation domain.

L80 – L81 : River length setting (optional) : newly added option to set arbitrary the length of river channel for each river grid cell (under preparation for more detail on this option).

L83 – L85 : River cross section settings (optional) : newly added option to set arbitrary cross section information for each river grid cell (under preparation for more detail on this option).

RRI_Input.txt

L87	1 1 0 1 0 0 0 0 0 1
L88	./out/hs_
L89	./out/hr_
L90	./out/hg_
L91	./out/qr_
L92	./out/qu_
L93	./out/qv_
L94	./out/gu_
L95	./out/gv_
L96	./out/gampt_ff_
L97	./out/storage.dat
L98	
L99	0
L100	./tecout.dat

L87 – L97 : Output file settings

Change the settings of L77 to "1" to output different sets of simulation results listed in the same order between L78 and L84

L99 – L100 : Output simulation results in Tecplot format (Optional)

Set 1 in L86 if one wants to get the input input file for Tecplot to visualize simulation results.

6. Running RRI Model

① Compiling RRI Model using make.bat

② Prepare "RRI_Input.txt" in "/RRI/Model/" folder (One can copy the sample of input file from /RRI/Model/infile/).

③ Execute "0_rri.exe", which reads the RRI_Input.txt stored in the same folder.

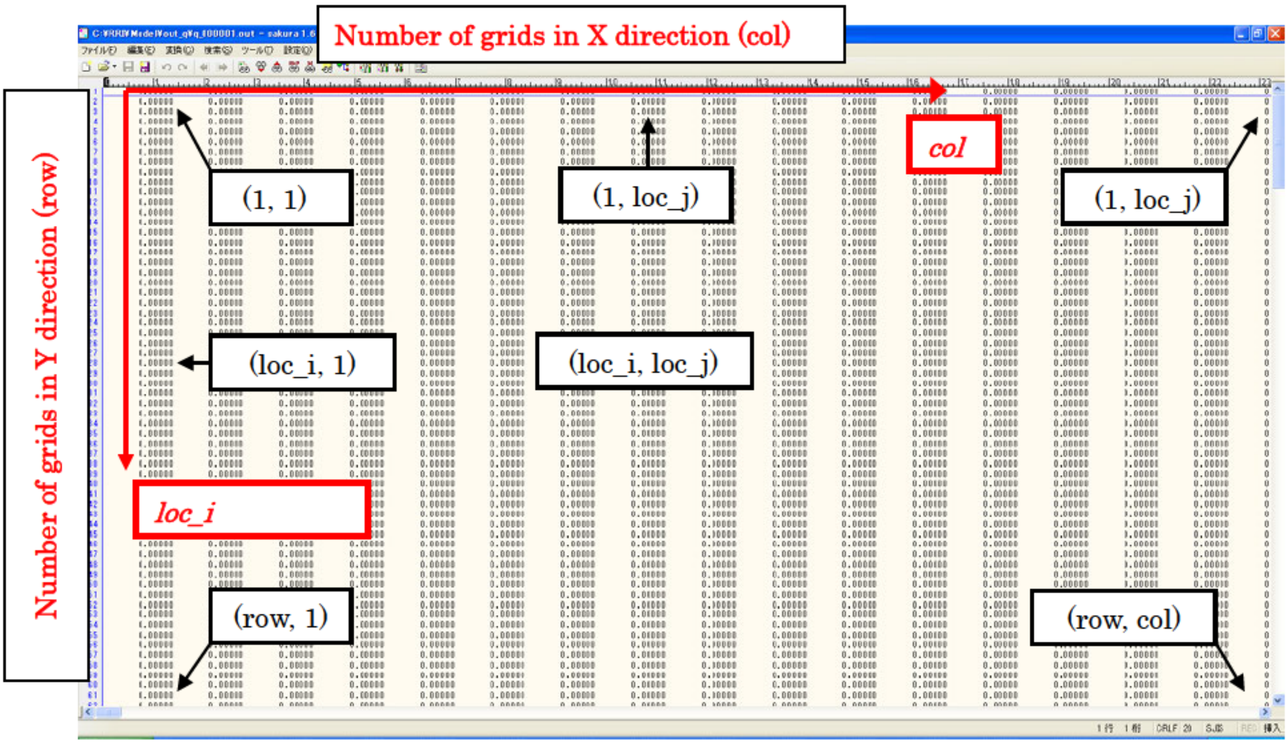
※For the folder configuration and roles of each folder of the RRI model, see Section 5

7. Visualize Output Data

This section explains how to visualize RRI Model output.

7.1 Format of the Output Files

Each output file contains water depths on slope (hs_) and on river (hr_) and river discharges (qr_) on river at a particular time step. The units of the output are [m] for water depths and [m³/s] for discharge.



※The numbers of rows and columns are the same as those of the topographic data.

Note that for each type of model output, the number of the files is defined in RRI_Input.txt (L13 : outnum). The simulation period is equally divided by "outnum" and the number assigned to each output file represents the output time stamp.

7.2 Visualize Inundation Depth with GNUPLOT

GNUPLOT can be used to illustrate flood inundation depth distributions. In RRI/Model, the GNUPLOT script named "hs.plt" is included. To change the settings, one can edit "hs.plt" directly or create another "hs.plt" by using a Fortran program named "prepHsPlt.f90" saved in "RRI/etc/prepHsPlt".

① Edit “hs.plt” file to change the configurations.

hs_plt.txt

```
reset

set terminal gif medium size 672, 408 crop

set pm3d map
set palette defined (0.0 "gray", 1.5 "blue", 3 "green")

set xrange [0:]
set yrange [:] reverse
set zrange [0:] reverse

#set xrange [180:200]
#set yrange [435:455] reverse

set cbrange[0.:3]
set zrange[0.0:]

set output "./hs/hs_000001.gif"
splot "./out/hs_000001.out" matrix t "000001 / 000096"

set output "./hs/hs_000002.gif"
splot "./out/hs_000002.out" matrix t "000002 / 000096"

set output "./hs/hs_000003.gif"
splot "./out/hs_000003.out" matrix t "000003 / 000096"
```

The size of output GIF file, X and Y direction. Use the same X and Y ratio as DEM’s col and row.

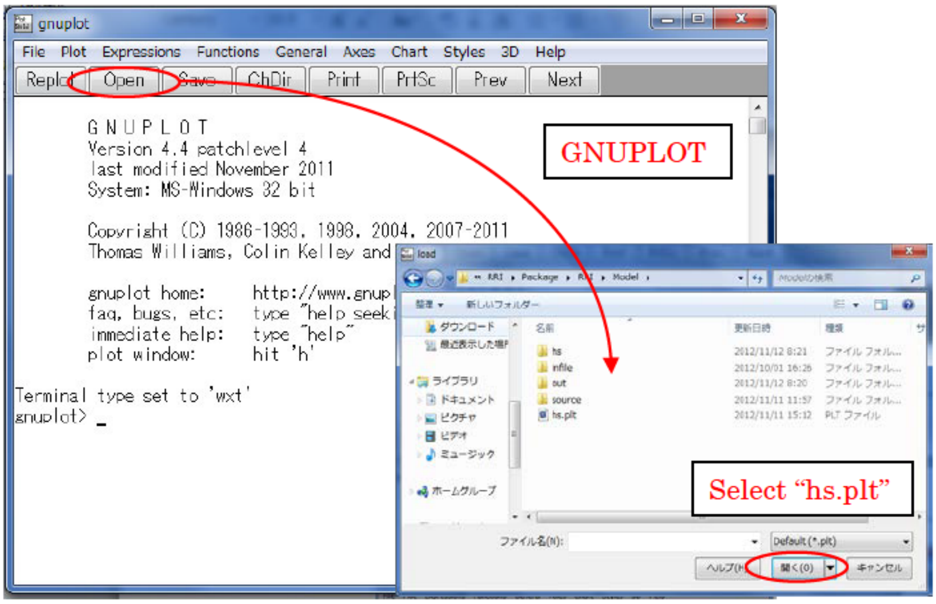
Color pattern settings

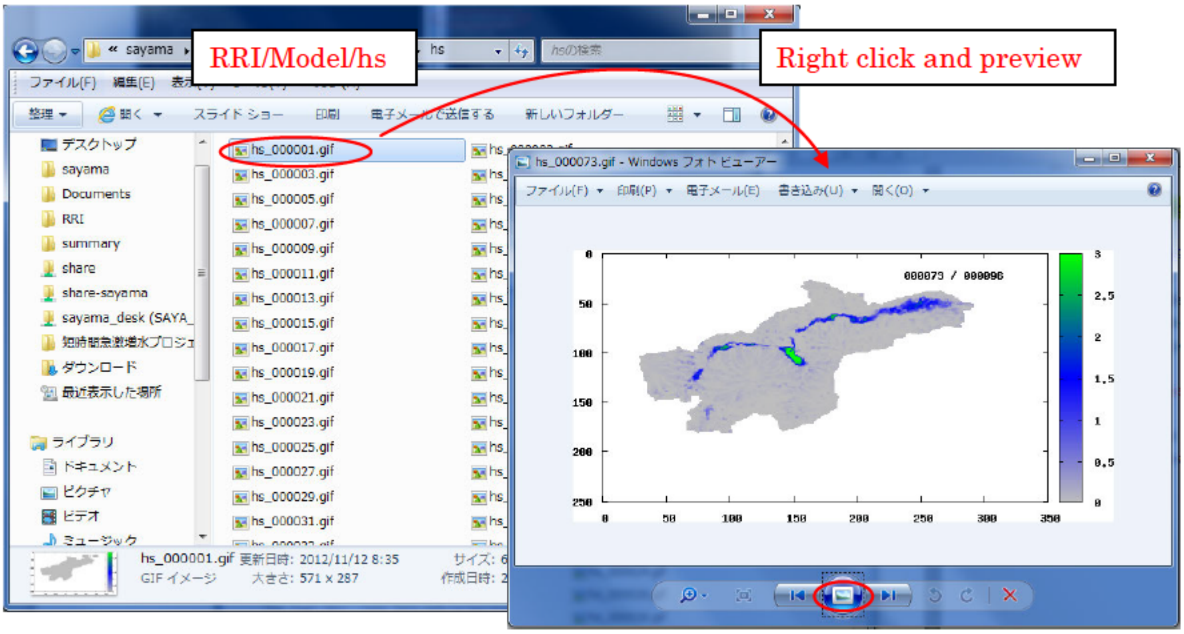
Color range

From RRI output (hs_***.out) to gif

② Start GNUPLOT program by clicking “/RRI/etc/gnuplot/binary/wgnuplot.exe”

Then open and select “hs.plt” script file.





7.3 Hydrographs at Specific Locations

A Fortran program named "calcHydro.exe" can be used to generate hydrographs by picking up values from "out/qr_***.txt" at specified locations.

① Edit "RRI/Model/calcHydro.txt" (see more details "RRI/etc/calcHydro/00_readme.txt")

L1 : [In] location file (e.g. ./infile/solo30s/location_solo_30s.txt)

L2 : [In] RRI output file (e.g. ./out/qr_)

L3 : [Out] hydrograph file (e.g. ./infile/solo30s/disc_)

calcHydro.txt

```
./infile/solo30s/location_solo_30s.txt
out/qr_
./infile/solo30s/disc_
```

location_30s_solo.txt

```
Cepu 68 167
(list all target locations)
```

② Run "calcHydro.exe". (Execute "makePostProcess.bat" in advance to compile.)

③ Check the created files specified in L3 of "calcHydro.txt". (e.g. ./infile/solo30s/disc_)

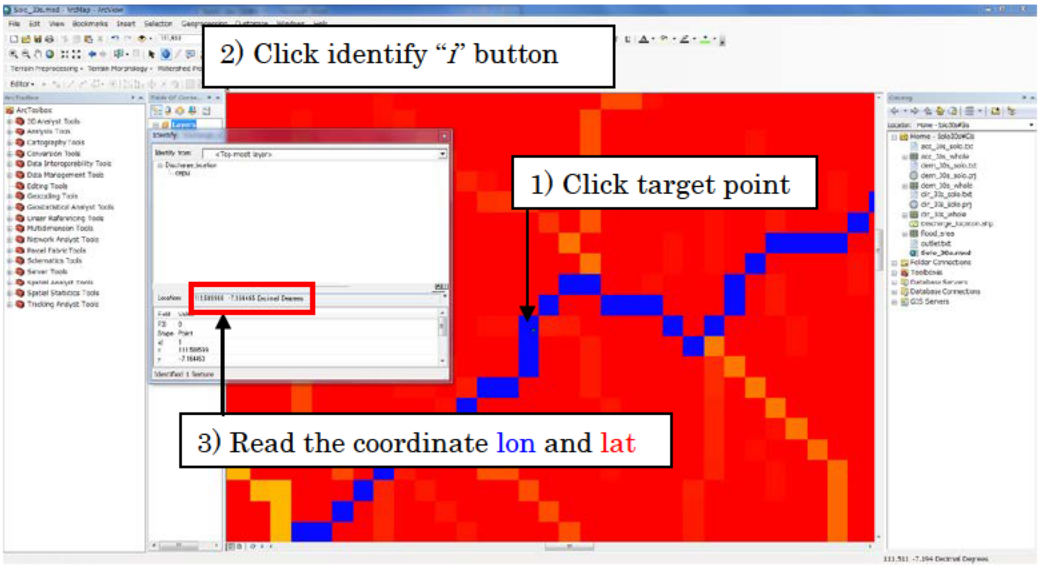
④ From GNUPLOT screen, open and select "hydrograph.plt", which is a GNUPLOT script file to plot hydrographs. Any other plotting software, such as Excel, can be also used to draw hydrographs from created files (e.g. ./infile/solo30s/disc_Cepu.txt).

In the location file (e.g. ./infile/solo30s/location_solo_30s.txt), one can list all target points, which you want to calculate hydrographs. Write the "*name of location*" and "loc_i" (y-direction) and "loc_j" (x-direction)

Note that "loc_i" is the row (y-direction from top) and "loc_j" is the col (x-direction from left).

To identify the observation points in mesh coordinate (loc_i, loc_j), one can use "/RRI/etc/coordinate.xlsx" to calculate based on the coordinate in latitude(y) and longitude(x).

① Find the latitude (y) and longitude (x) of the observation point using ArcGIS.



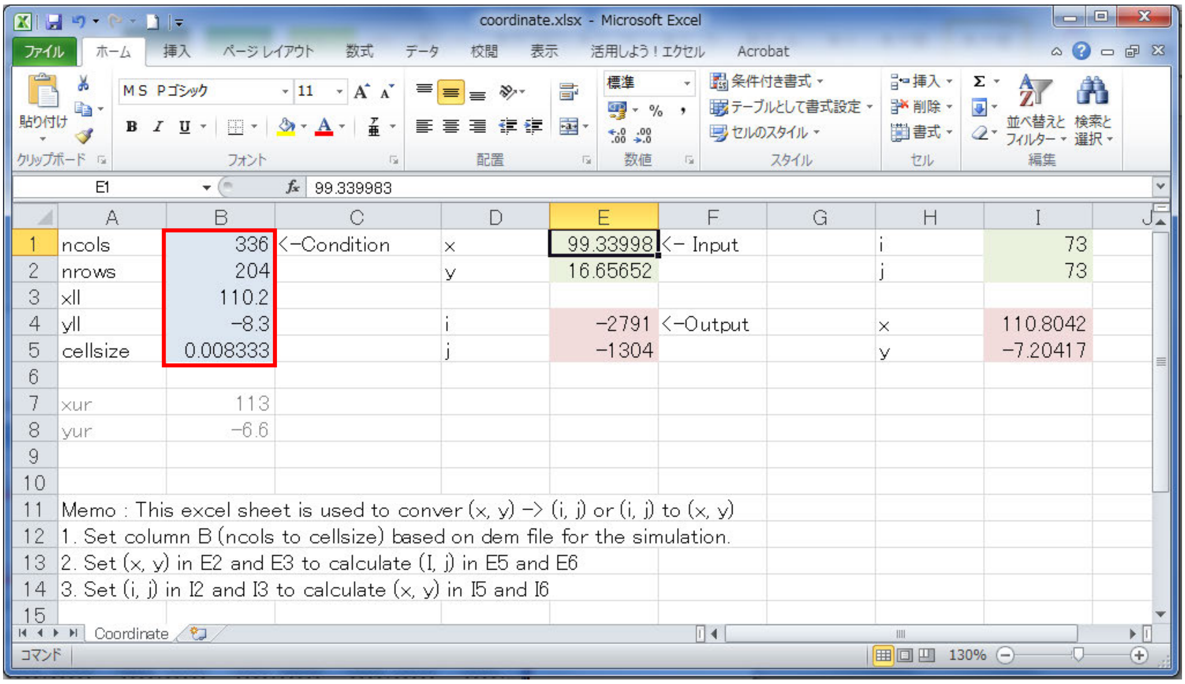
(Displaying "acc" on top to make sure the selected point is on a river grid cell.)

② Open one of the topographic data (i.e. dem, dir, or acc)

acc_solo_30s.txt

```
ncols           336
nrows            204
xllcorner       110.2
yllcorner       -8.3
cellsize        0.00833333333333
NODATA_value   -9999
 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
```

③ Read the header part (red box in the above figure) of the topographic data and copy the same information in the Excel file (i.e. /RRI/etc/coordinate.xlsx).



④ Type x and y (or lon and lat) coordinate of the target point, then the calculated mesh coordinate (loc_i, loc_j) appears in (E4, E5).

("coordinate.xlsx" can be used also to convert from (loc_i, loc_j) to (lon, lat).

7.4 Visualize Peak Inundation Depths

Fortran program named "calcPeak.exe" can be used to compute the maximum flood depths based on RRI Model output ("out/hs_*.out"). See 2.2.3 the procedure more in detail.

① Edit "RRI/Model/calcPeak.txt" file after RRI model execution.
In "calcPeak.txt", L1 sets the path of dem file, L2 sets the RRI model output file to calculate the peak, and L3 sets the number of output files. L4 defines the output file of calcPeak program. See details the readme file of "/etc/calcPeak".

② Execute "calcPeak.exe". (Execute "makePostProcess.bat" if the executable file does not exist.)

③ Check the created files specified in L4 of "calcPeak.txt".

④ The obtained peak water data follows ESRI/ASCII format that can be visualized with ArcGIS.

7.5 Visualize Inundation Depths with Google Earth (Optional)

7.5.1 Preparing KML File

By using "RRI/etc/makeKML.f90", a kml file (e.g. "runoff.kml") can be prepared.

User needs to edit "RRI/etc/Kml_input.txt".

Kml_input.txt

```
2007   12   24    0    0                : Start time (Year Month Day Hour Min ( UTC ))
  96                                    : Number of gif files( = "outnum" of RRI_Input.txt )
 3.75                                   : Timestep (hourly) ( ≒ "lasth / outnum" of RRI_Input.txt )
./infile/solo/adem_30s_solo.txt         : Dem file name (for lat,lon)
./runoff.kml                            : Output file name
```

"Time step" needs to be input as "hourly" data.
This "Time step" should be "lasth" / "outnum" input in "RRI_Input.dat".

When it is executed, "runoff.kml" is output.

runoff.kml

```
<Folder>
 <GroundOverlay>
  <TimeSpan>
   <begin>2007-12-24T00:00Z</begin>
   <end>2007-12-24T03:45Z</end>
  </TimeSpan>
  <Icon>
    <href>hs_kml/hs_000001.gif</href>
  </Icon>
  <LatLonBox>
   <north>   -6.60000</north>
   <south>   -8.30000</south>
   <east> 113.00000</east>
   <west> 110.20000</west>
  </LatLonBox>
 </GroundOverlay>
…
```

※ The output of "runoff.kml" reads gif files created in the folder of "hs_kml".

7.5.2 Preparing GIF Files with GNUPLOT

The method of plotting “hs_kml.plt” using “gnuplot” is shown below.

① Prepare a gnuplot file (e.g. “RRI/Model/hs_kml.plt”), which can be essentially the same as hs.plt explained above. However, the gnuplot script file used here (i.e. hs_kml.plt) must have some additional statements in the blue box in the following figure. The statements delete unnecessary axis and legends to be appropriately overlay on Google Earth.

hs_kml.plt

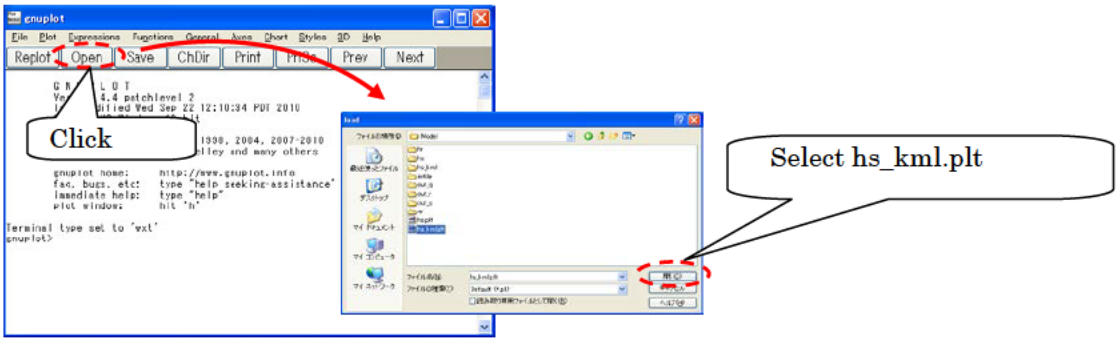
```
reset

set terminal gif medium size 672, 408 crop

set lmargin 0
set bmargin 0
set rmargin 0
set tmargin 0
set notics
set nokey
unset colorbox

set pm3d map
set palette defined (0.0 "gray", 1.5 "blue", 3 "green")

set xrange [0:]
set yrange [:] reverse
set zrange [0:] reverse

#set xrange [180:200]
#set yrange [435:455] reverse

set cbrange[0.:3]
set zrange[0.0:]

set output "./hs_kml/hs_000001.gif"
splot "./out/hs_000001.out" matrix t "000001 / 000096"
…
```

modify: 672, 408 — Designate size so that the aspect ratio of size and ratio of number of meshes match.

add: set lmargin 0 … unset colorbox — This part must be added to the original hs.plt file.

modify: /hs_kml/ — The folder name, “hs_kml” should be input here.

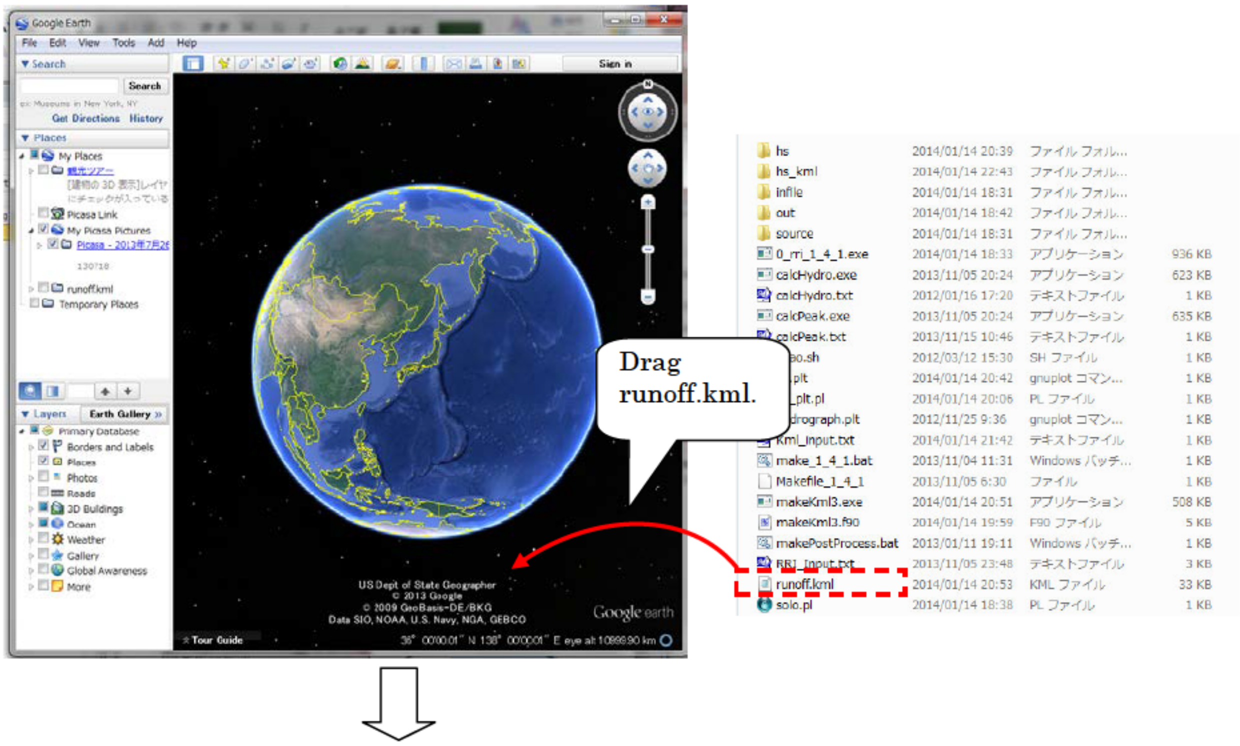
② Start “GNUPLOT” and run “RRI/Model/ hs_kml.plt”.

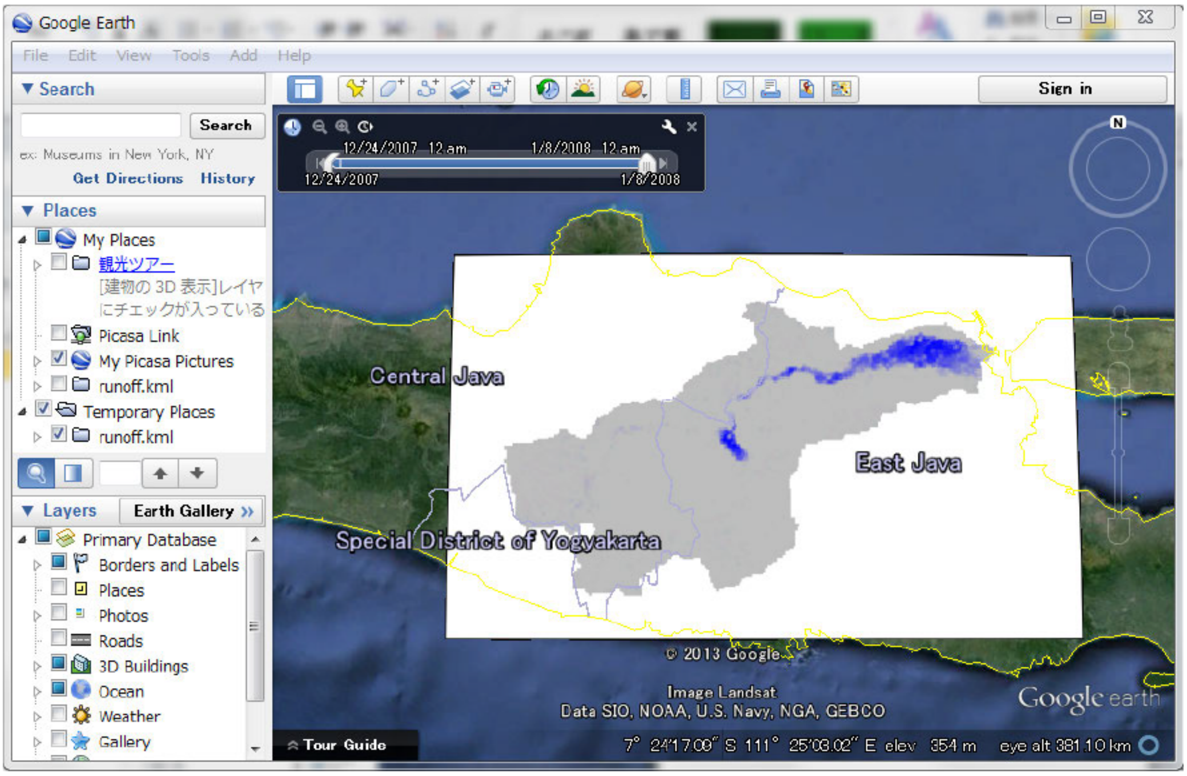


③ An image file is prepared in the “RRI/Model/hs_kml” folder. (Note that a new folder hs_kml must be created in advance.)

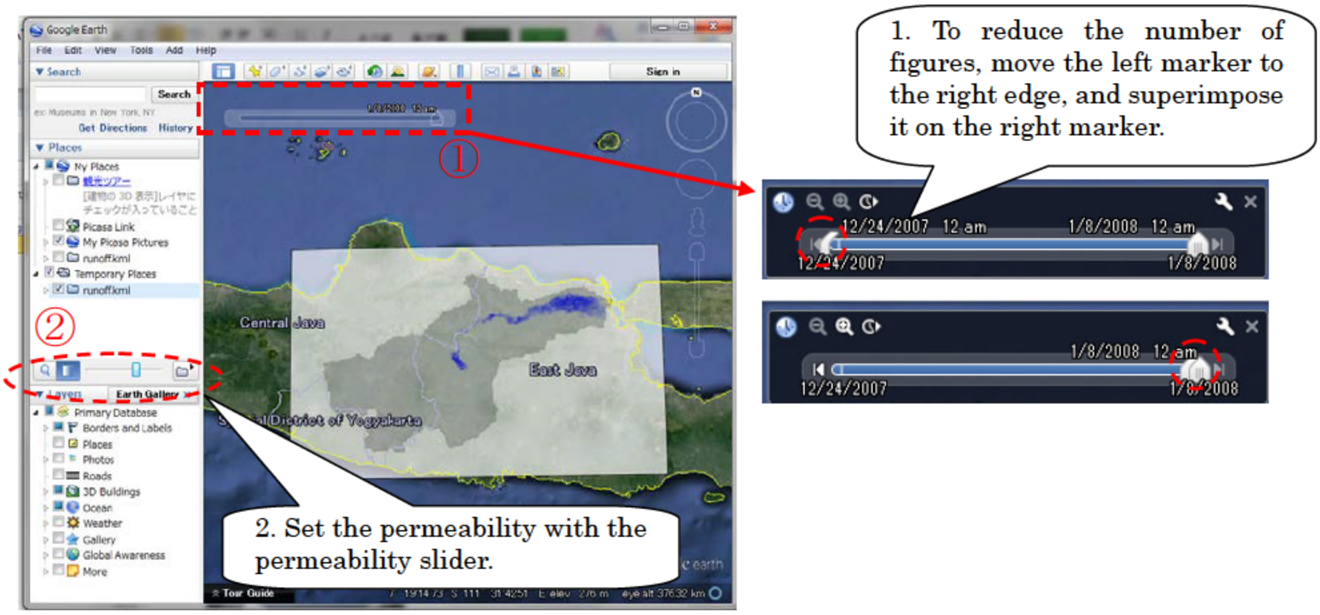
7.5.3 Visualize GIF files with Google Earth

① Start Google Earth and drag “/RRI/Model/runoff.kml”.



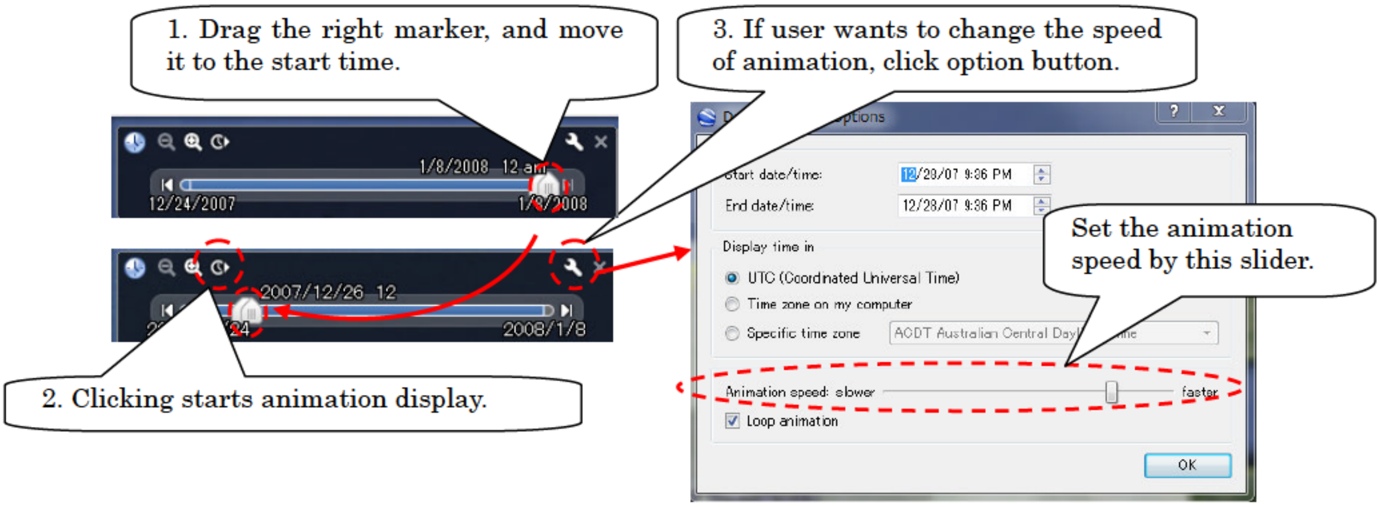


② Designate the number of figures to display at once and their transparency.

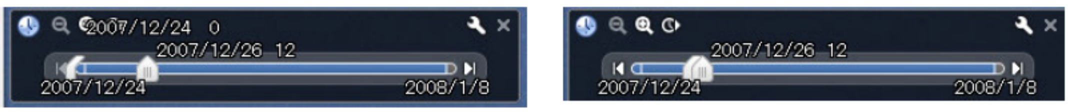


※ On time slider: The right marker represents the present time, while the left marker is used for the number of figures to overlay. Figures in the period between two markers are displayed.

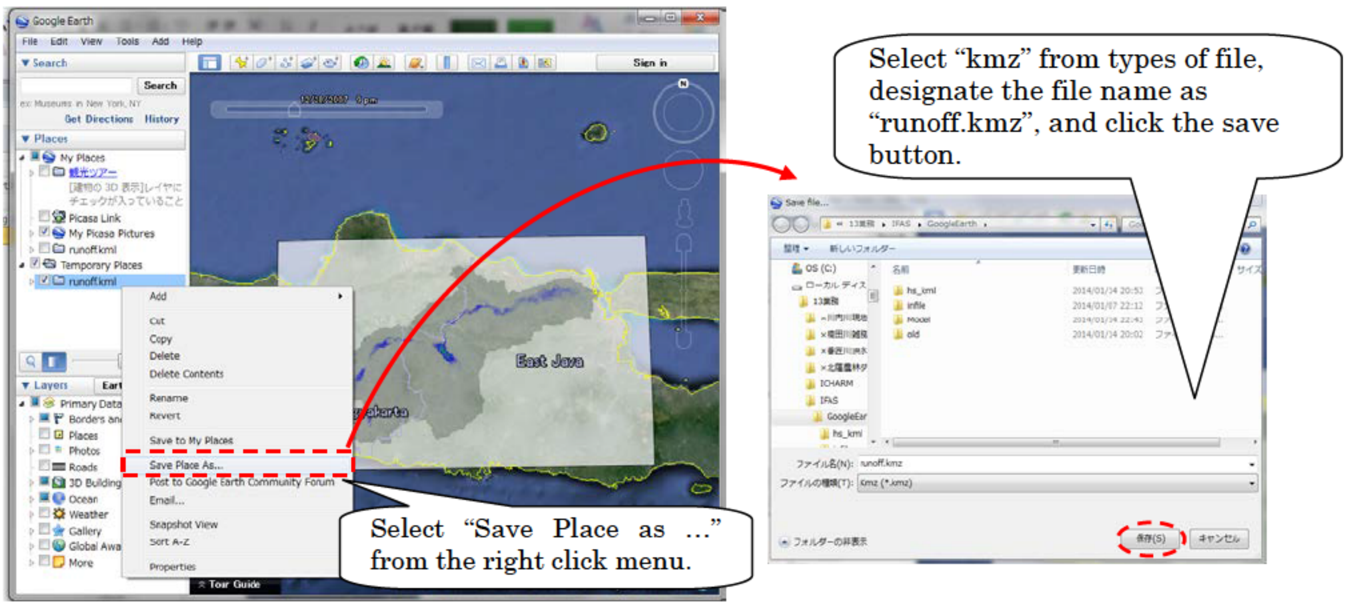
③ Execute animation.



※ <note>. During the animation, two markers should be moved at the same time. If user can't move the left marker, stop the animation and fit the left marker to the position of right marker and restart the animation.



④ Save the results with kmz file, so that it can be distributed to other users without gif files.



7.6 Visualize Results with Tecplot (Optional)

7.6.1 Preparing input File of Tecplot

calcTecplot.txt

```
2007 12 24 00                  # start time
360                             # lasth [hour]
96                              # outnum [-]

./Model/infile/solo30s/rain_solo_30s_gauge.dat
./Model/infile/solo30s/adem_30s_solo.txt

110.2d0             # xllcorner_rain
-8.3d0              # yllcorner_rain
0.00833333d0 0.00833333d0     # cellsize_rain

1 1 0 1 1 1 0 0 0 1
./Model/out/hs_
./Model/out/hr_
./Model/out/hg_
./Model/out/qr_
./Model/out/qu_
./Model/out/qv_
./Model/out/gu_
./Model/out/gv_
./Model/out/gampt_ff_

./calcTecplot_out.dat

TITLE        = "Internally created data set"
VARIABLES = "X"
"Y"
"DEM (m)"
"Rainfall (mm/h)"
"Water depth hs (m)"
"Water depth hr (m)"
"Water depth hg (m)"
"River discharge qr (m3/s)"
"Slope discharge qu (m3/s)"
"Slope discharge qv (m3/s)"
"Ground discharge gu (m3/s) "
"Ground discharge gv (m3/s)"
"g-ampt (m)"

ZONE T="Contour T ="
STRANDID=1, SOLUTIONTIME=
I=336, J=204, K=1, ZONETYPE=Ordered
DATAPACKING=POINT
DT=(SINGLE SINGLE SINGLE SINGLE SINGLE SINGLE SINGLE SINGLE SINGLE
SINGLE SINGLE SINGLE SINGLE)
```

: Year, Month, Day, Hour

: calculation time (l.10)

: output file number (l.13)

: Rainfall file (l.3)

: Dem file (l.4)

grid data of Rainfall (from l.14 to l.16)

Output file from RRI (from l.87 to l.96)

User needs to edit these lines in "calcTecplot.txt". All the lines except for the first line (start time) can be copied from "RRI_Input.txt" to be compatible with simulation setting.

Output file name for Tecplot (use "dat" for the extention)

All data outside the red border are header information, which is not necessary to be modified.

"VARIABLES =" …" shows the variables to be displayed on Tecplot. Edit this if necessary.

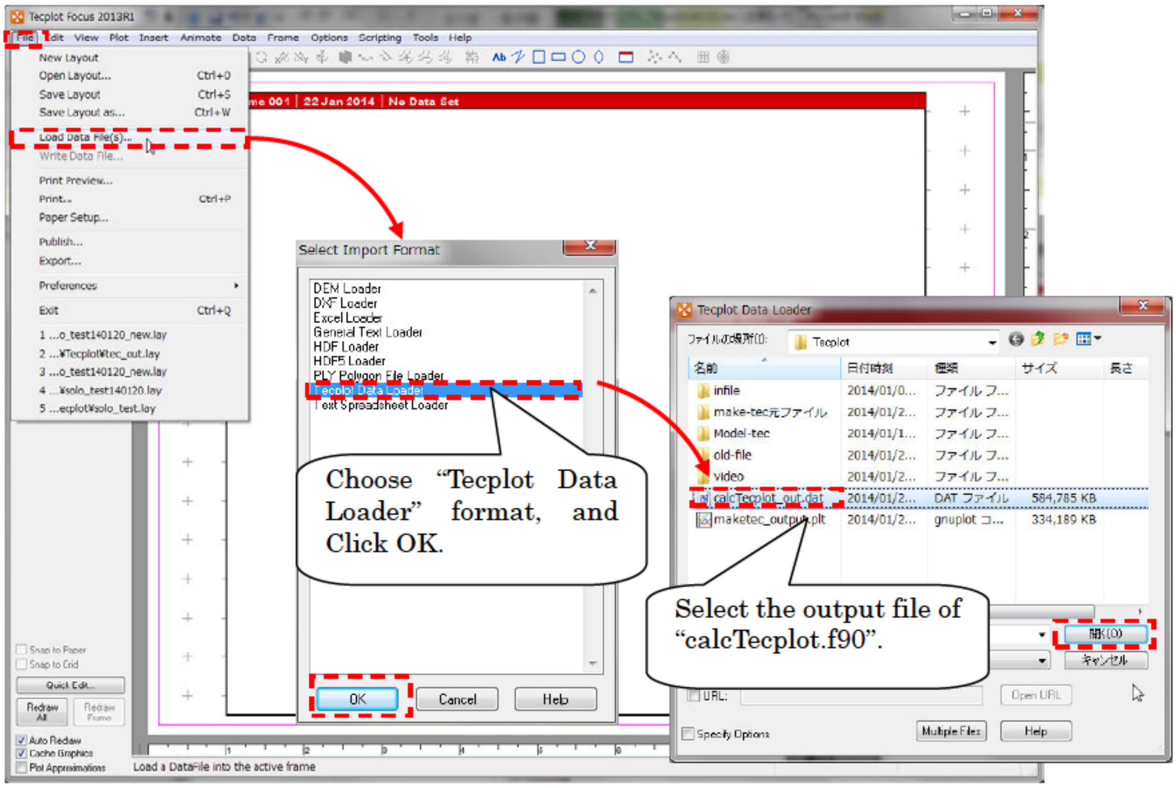
Use “RRI/etc/calcTecplot.f90” to prepare an input file for Tecplot (e.g. “calcTecplot_out.dat”). Prior to running calcTecplot.exe, edit “RRI/etc/calcTecplot.txt”, which sets the condition for generating the input file.

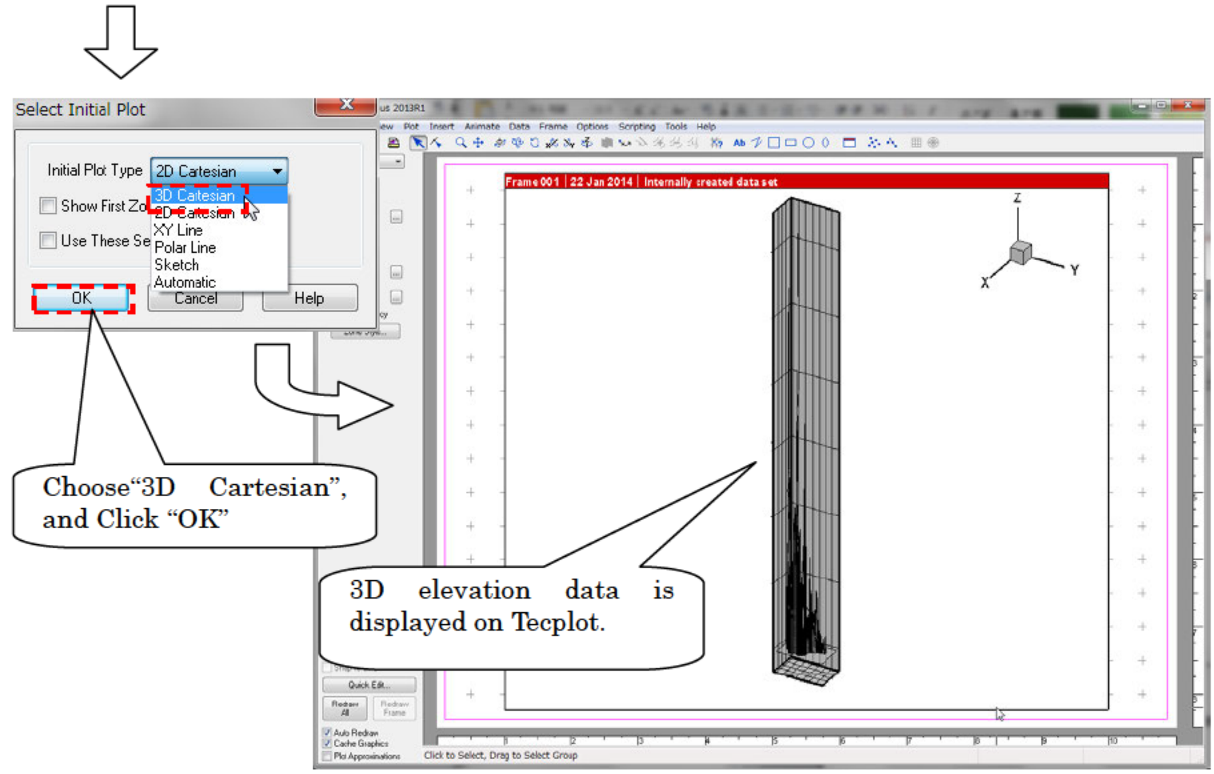
7.6.2 Displaying on Tecplot

① Start Tecplot, and load data file.

[File] > [Load Data file(s)] > [Tecplot Data Loader] > [calcTecplot_out.dat]

It takes several minutes to load the data file.



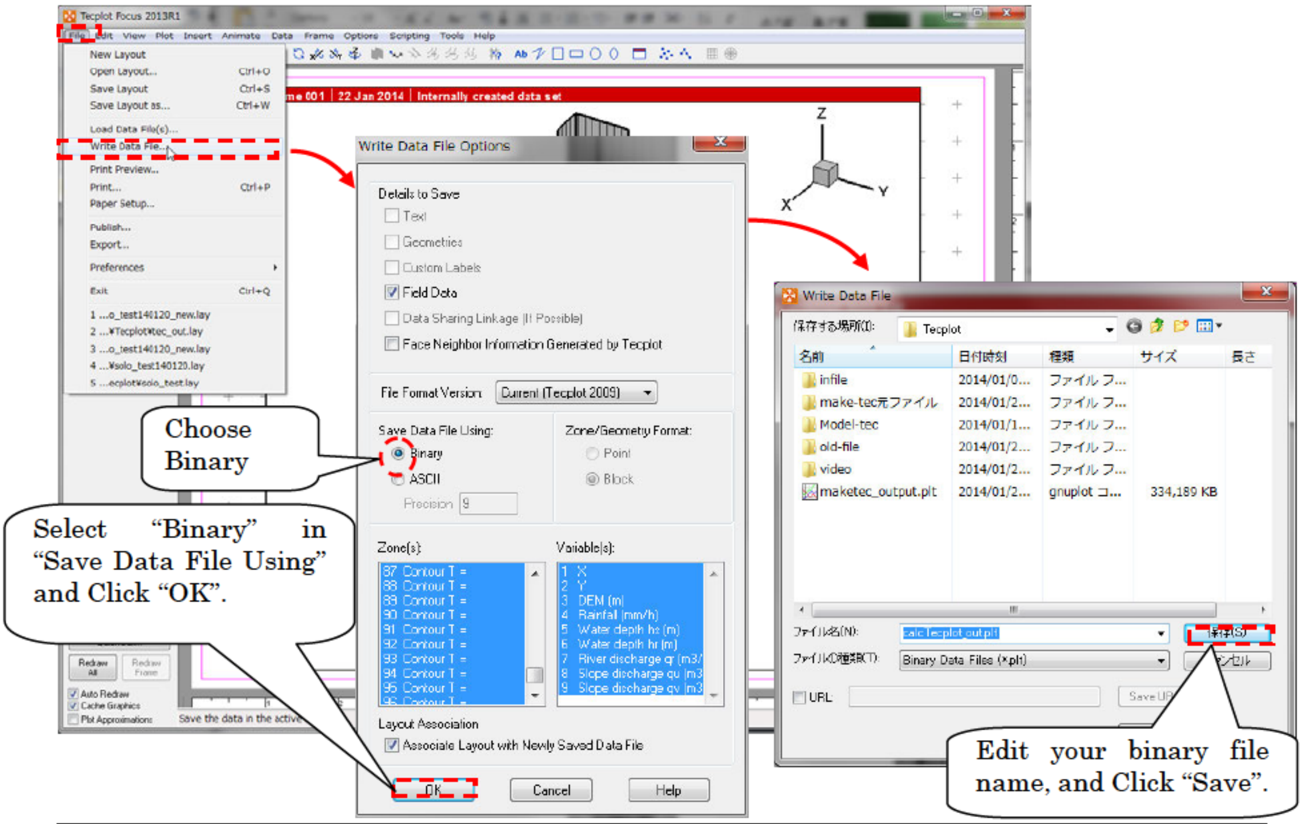


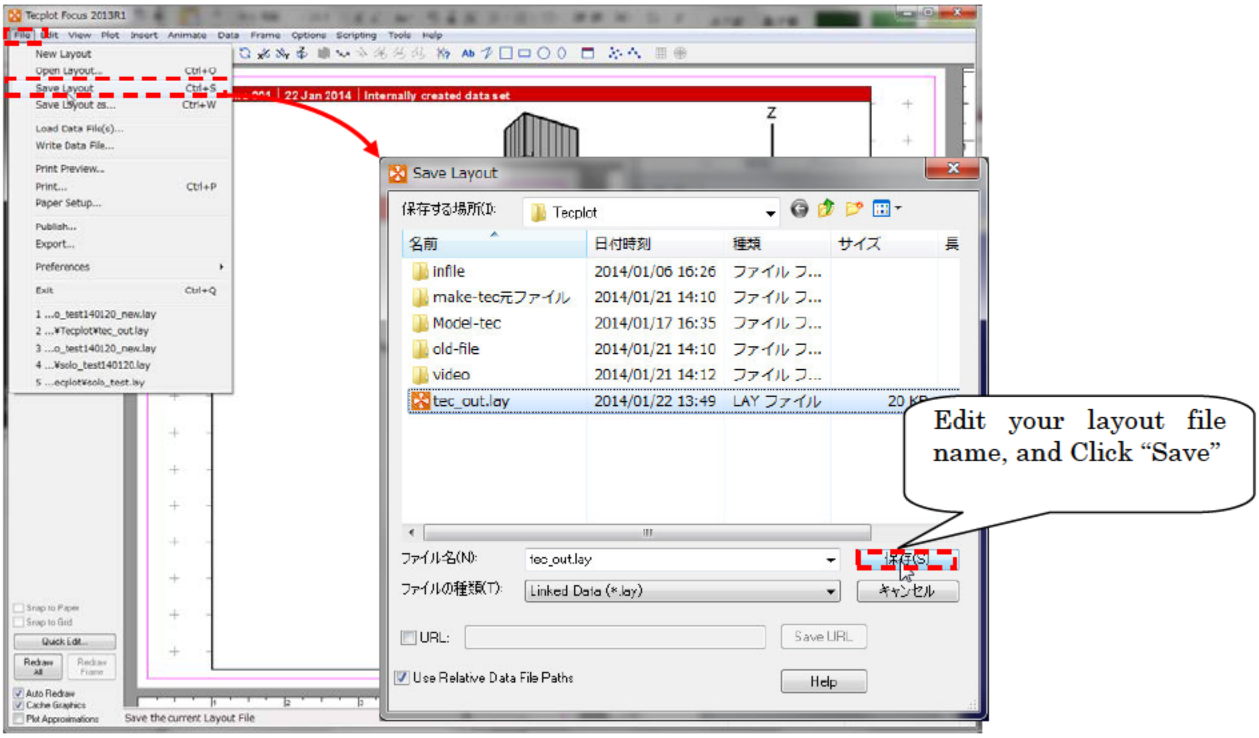
② Write data file (changing input data to binary data), and also save as layout file.

[File] > [Write Data file…]

[File] > [Save Layout] ..

By Making the binary data (*.plt), user can reduce the amount of time to reload layout file. User needs longer time to reload without the binary file.

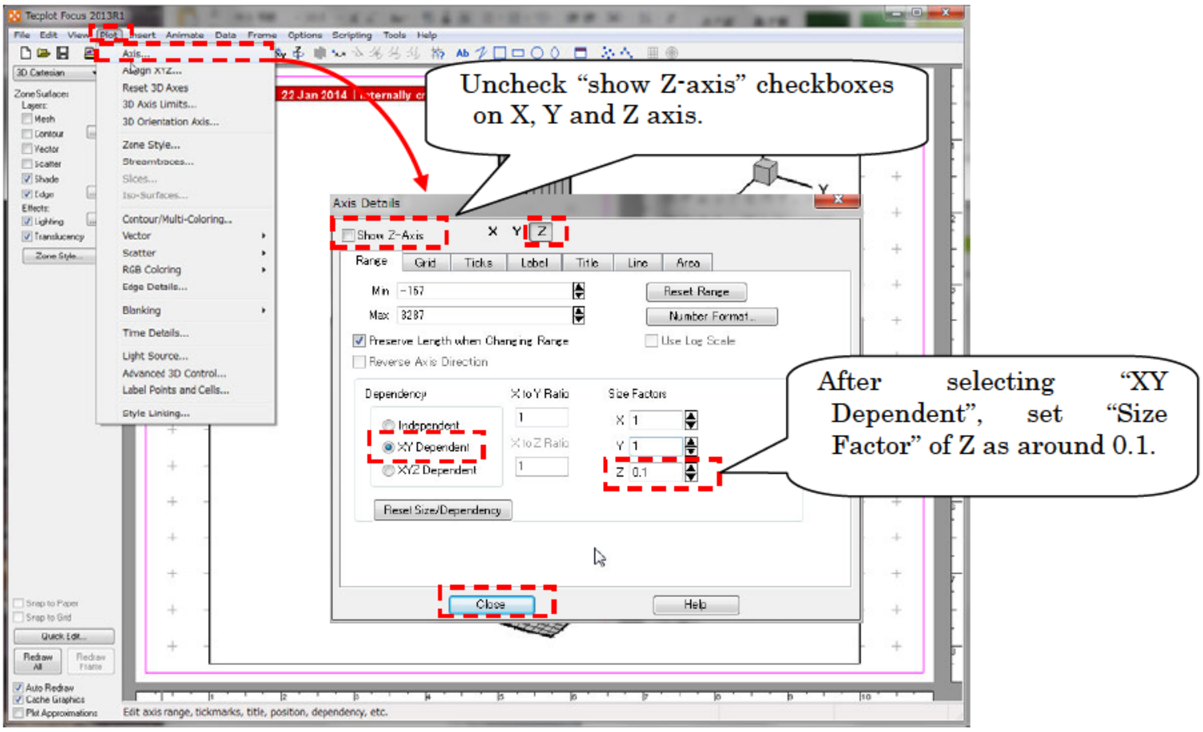




7.6.3 Edit display options on Tecplot

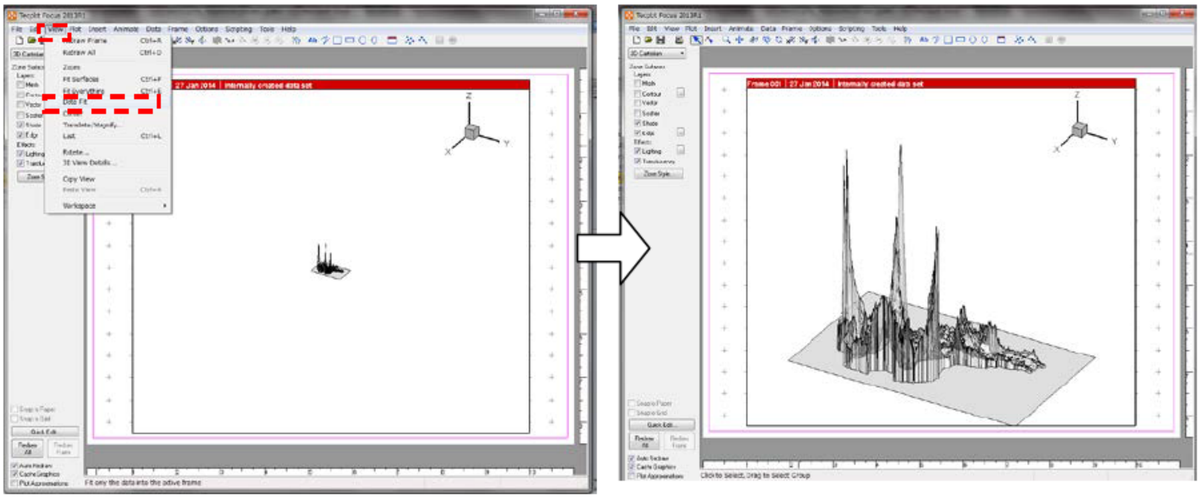
① Edit the ratio of XYZ and hide the axes.

[Plot] > [Axis ...] Select "XY Dependent" in Dependency on "Z" tab and input Size Factors in Z (following example shows the Size Factors Z is set to 0.1). Uncheck "Show X(Y,orZ)-axis" on X, Y and Z tab to hide the axis.



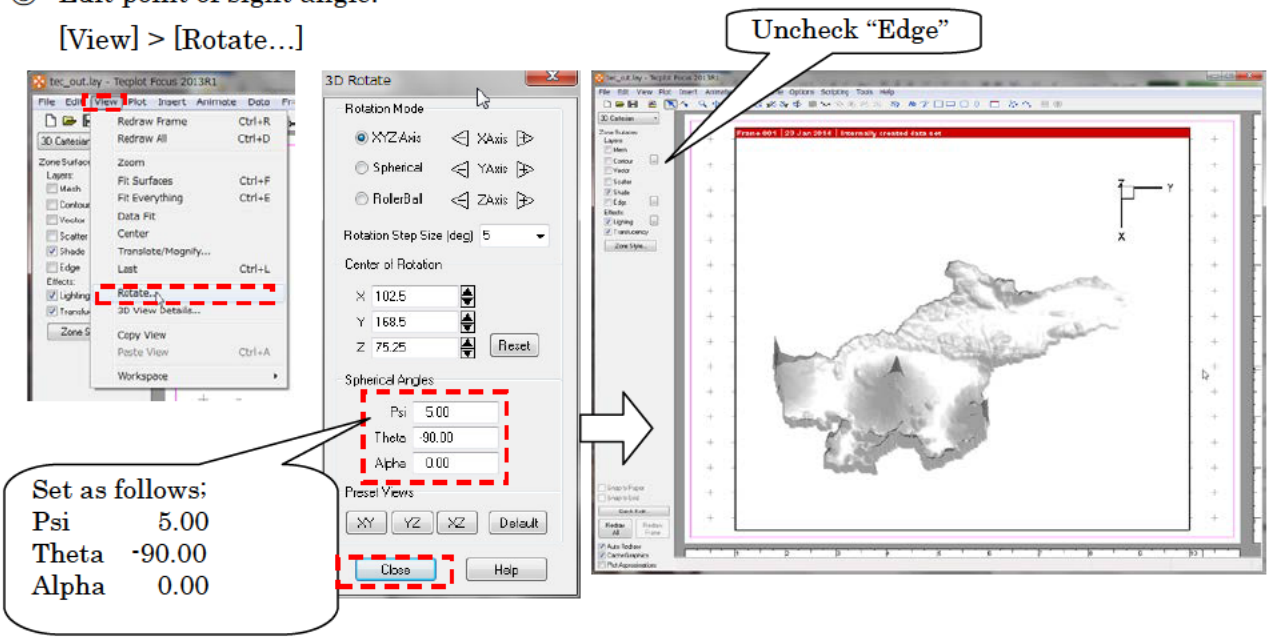
② Fit the data display range to the target range.

[View] > [Data fit]



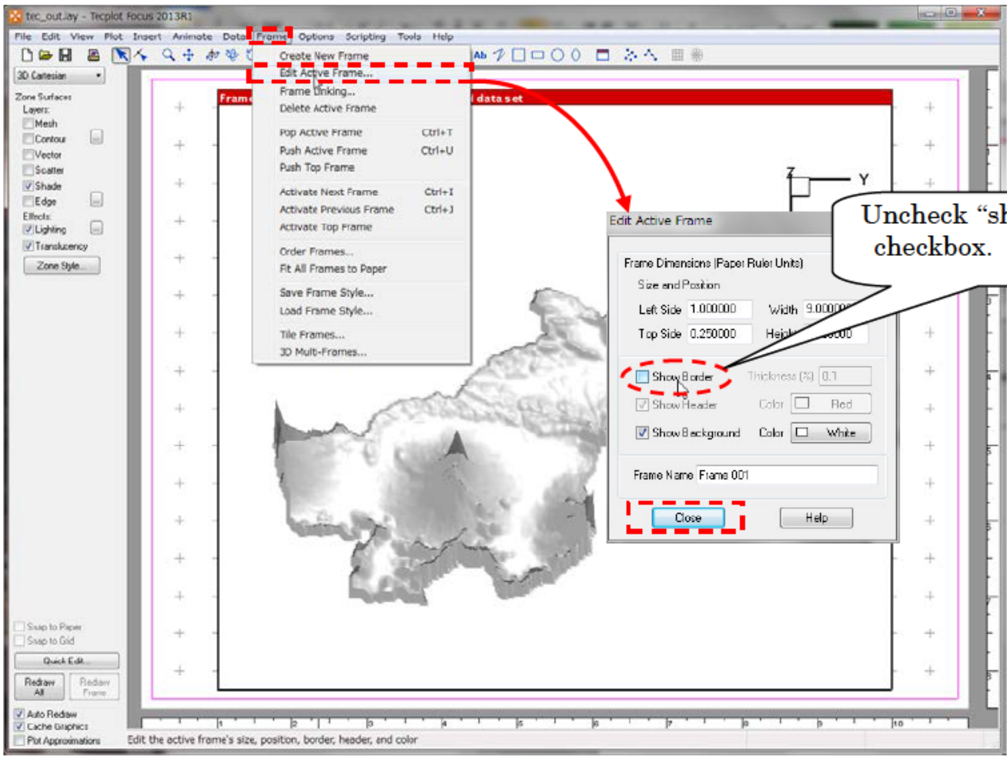
③ Edit point of sight angle.

[View] > [Rotate…]



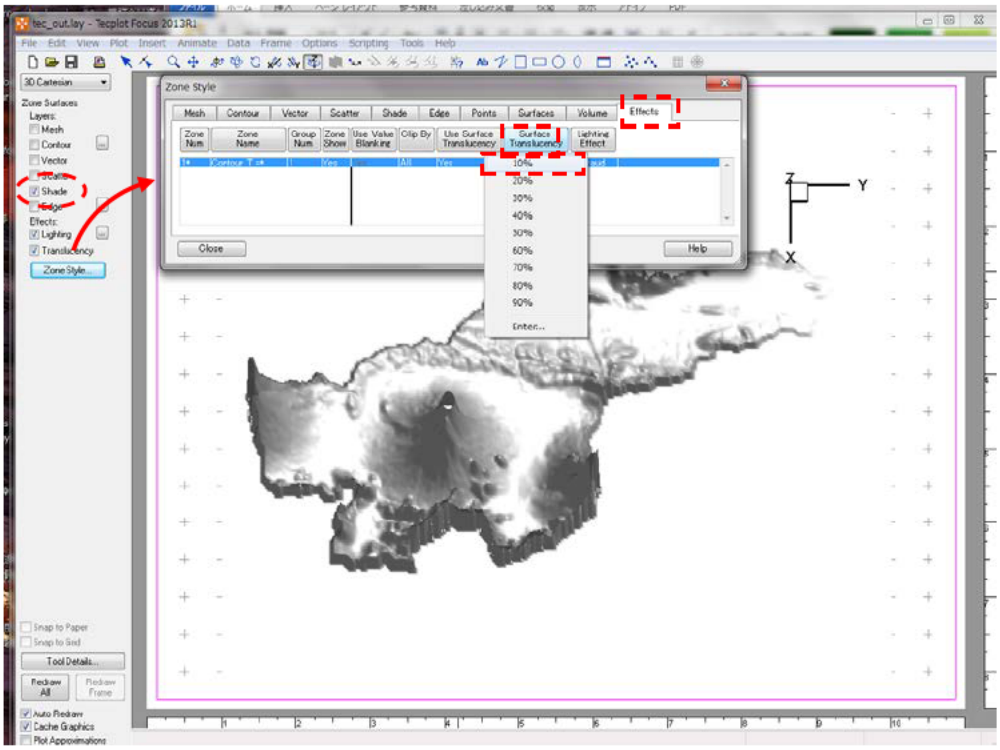
④ Delete unnecessary frame

[Frame] > [Edit Active Frame…]



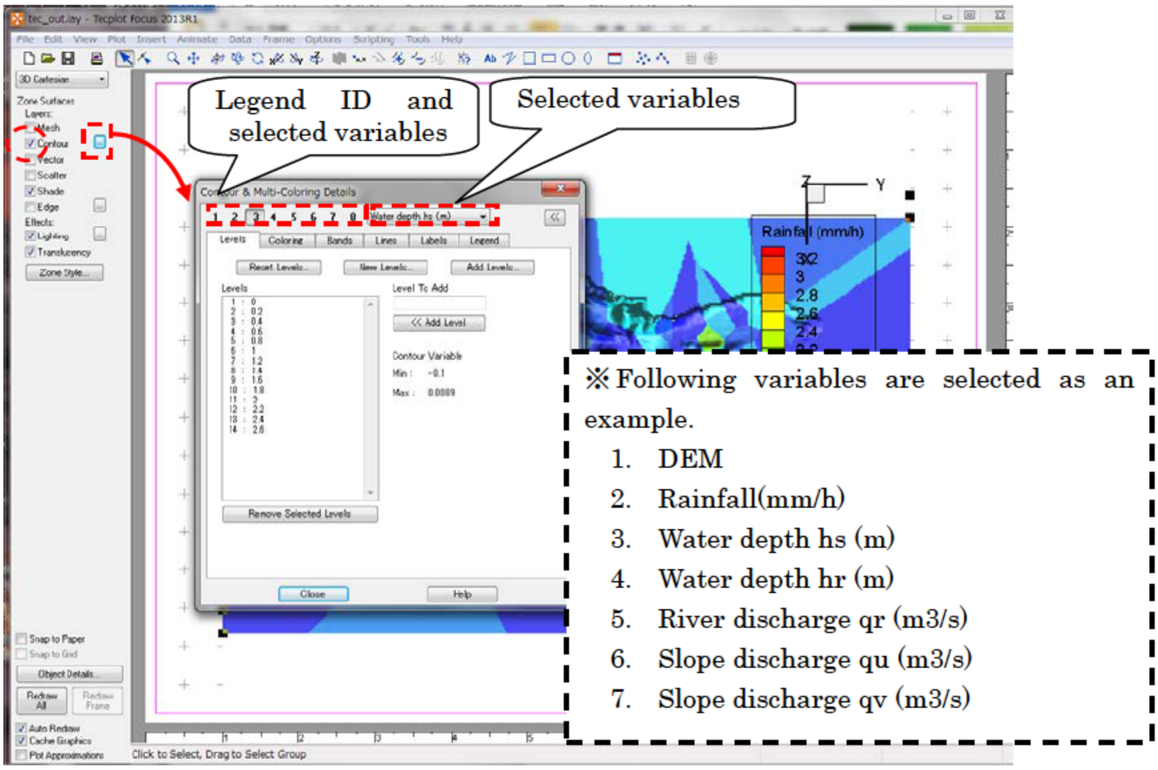
⑤ Edit translucency of shade

Click "Zone Style" and edit the value of "Surface Translucency" on "Effects Tab" to change the translucency of shade (e.g. 10%).

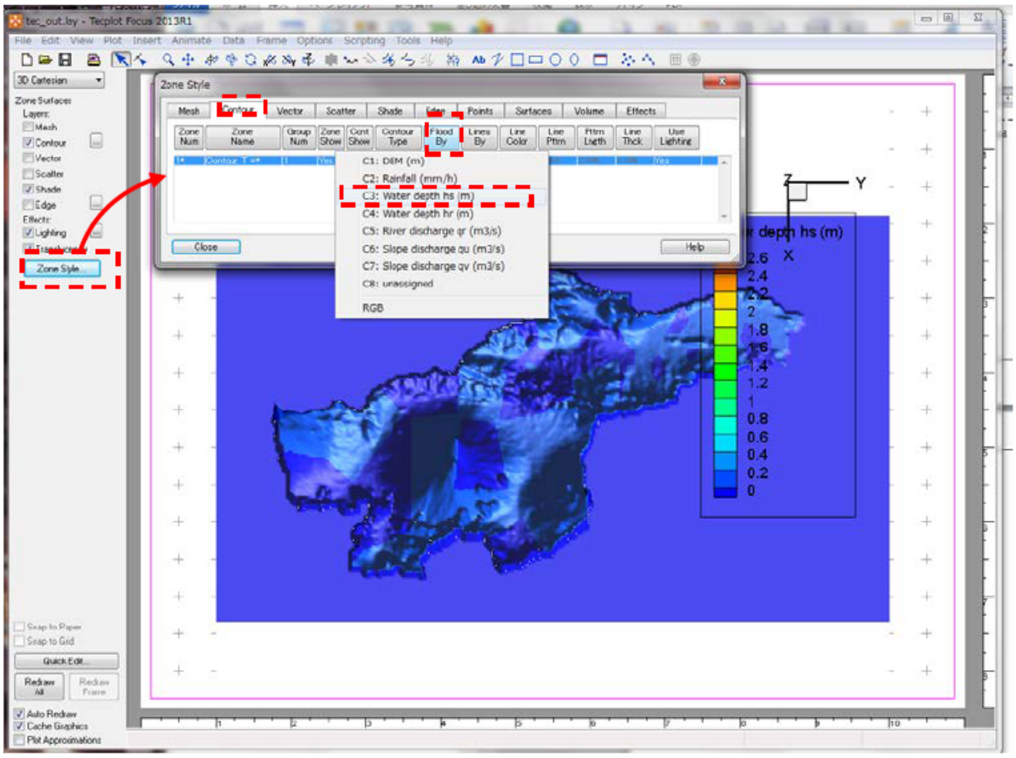


7.6.4 Draw contour figure on Tecplot

① Select variables to draw contour. User can select variables up to eight variables. The legends of variables are automatically set. The method to edit them is described in ③.

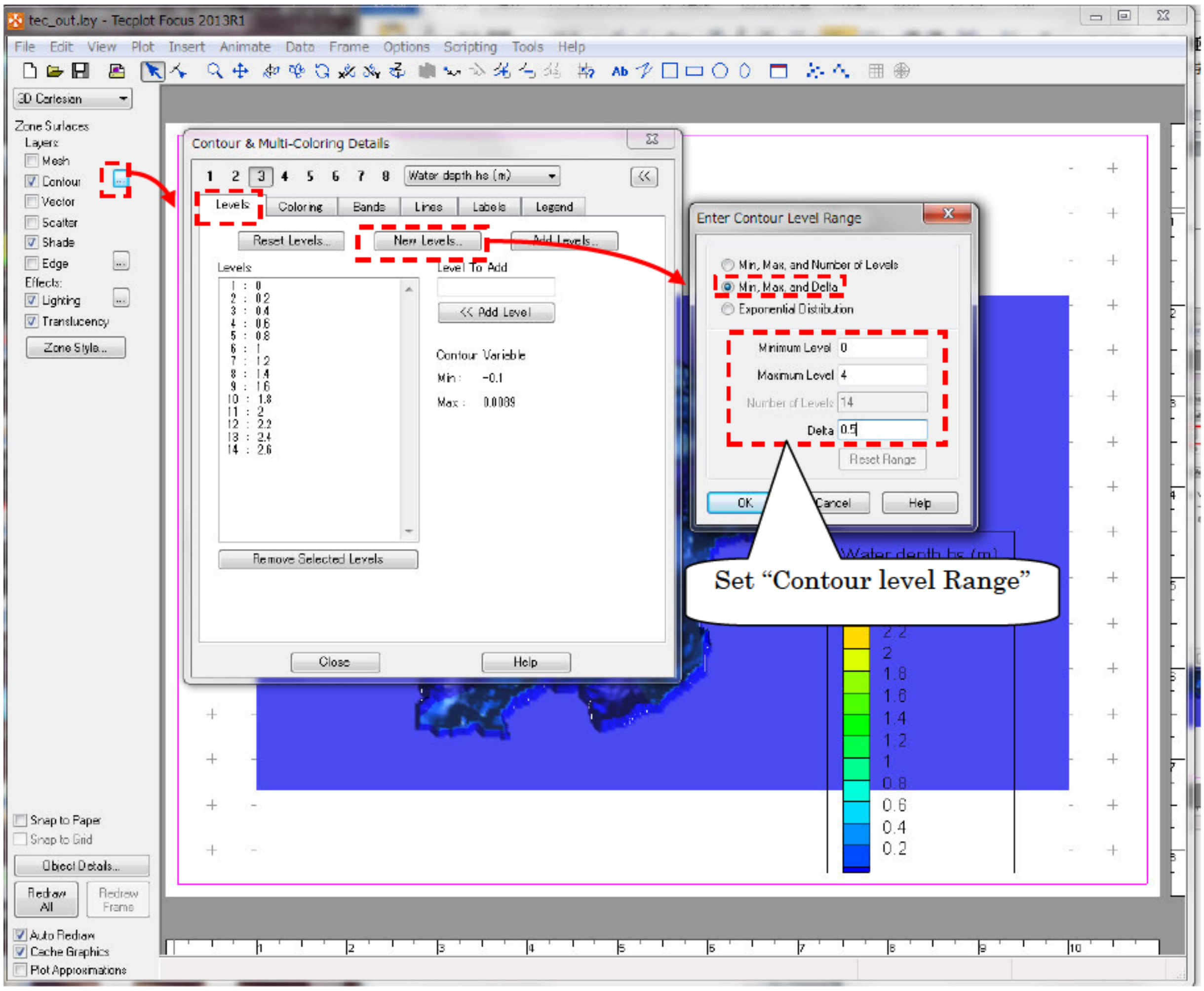


② Select variable to display. User can select variable from variables identified in ①. Click "Zone Style" and edit "Flood By" on "Contour" tab to edit target variable and its legend. "Water depth hs (m)" is selected in the following figure as an example.

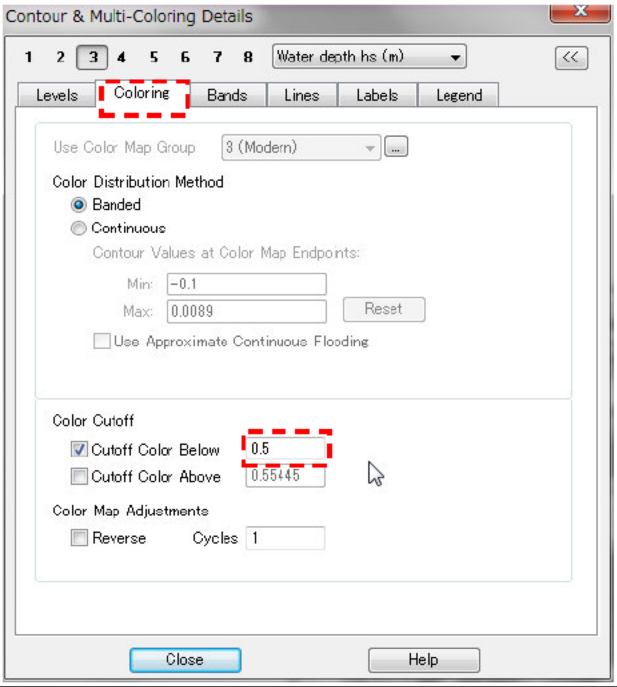


③ Edit legend.

User can edit color legend of contour as follows;

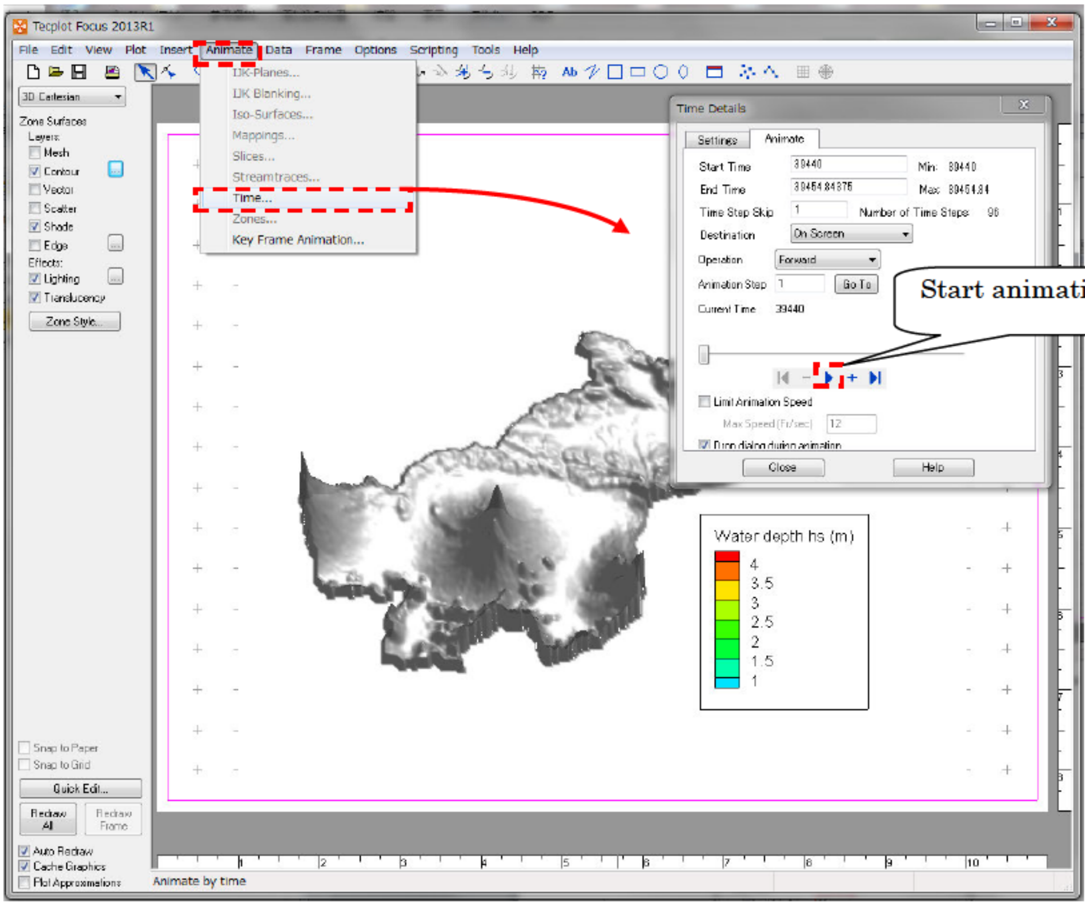


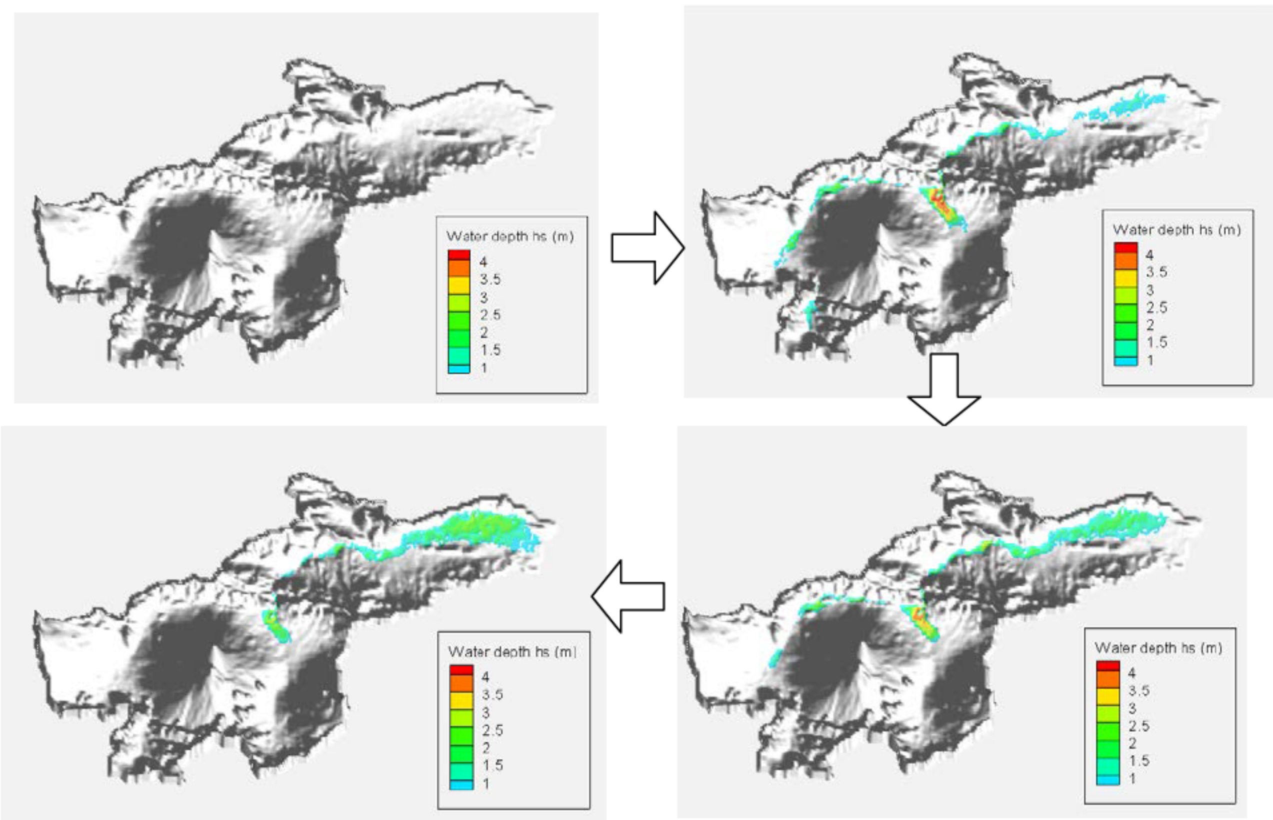
④ User also can edit “cut off” to display upper and lower color limits. Color up to 0.5m is cut in the following figure as an example.



⑤ User can check the time series of the contour figure.

[Animate] > [Time…]



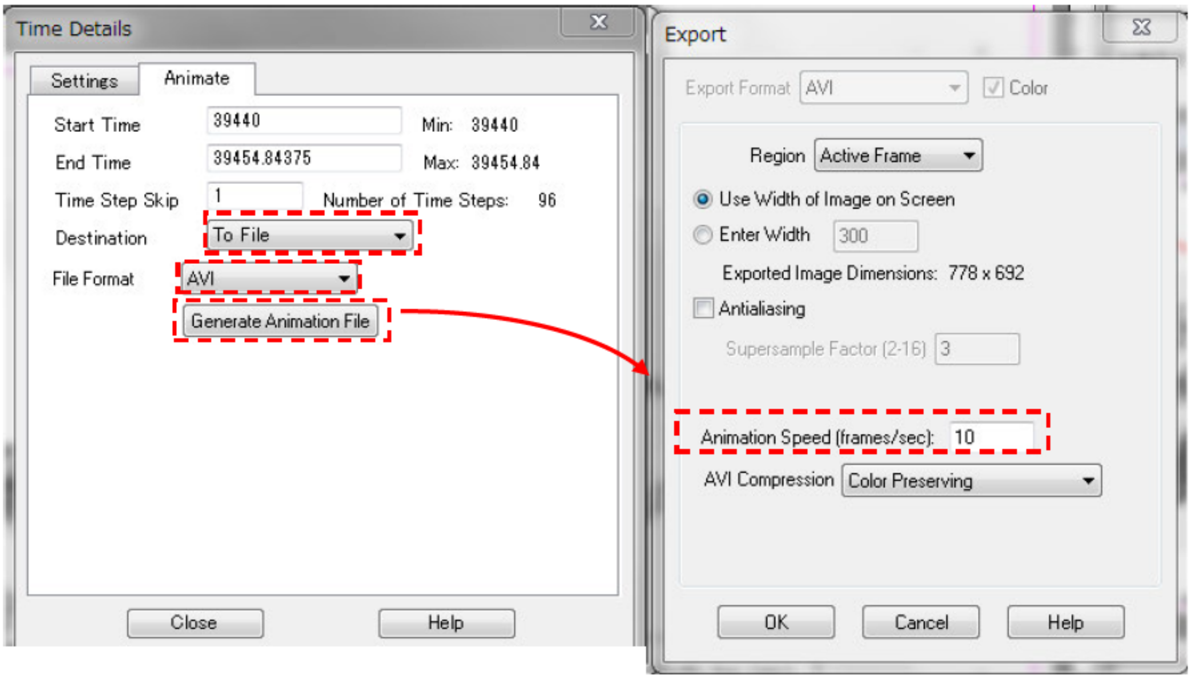


⑥ Export animation file

User can export animation file.

[Animate] > [Time…]

Select “To File” in “Destination” and “AVI” in “File Format” on “Animates” tab in “Time Details”. If user needs to edit animation speed, click “Generate Animation File” and edit “Animation Speed” if necessary.



7.6.5 Supplement of display

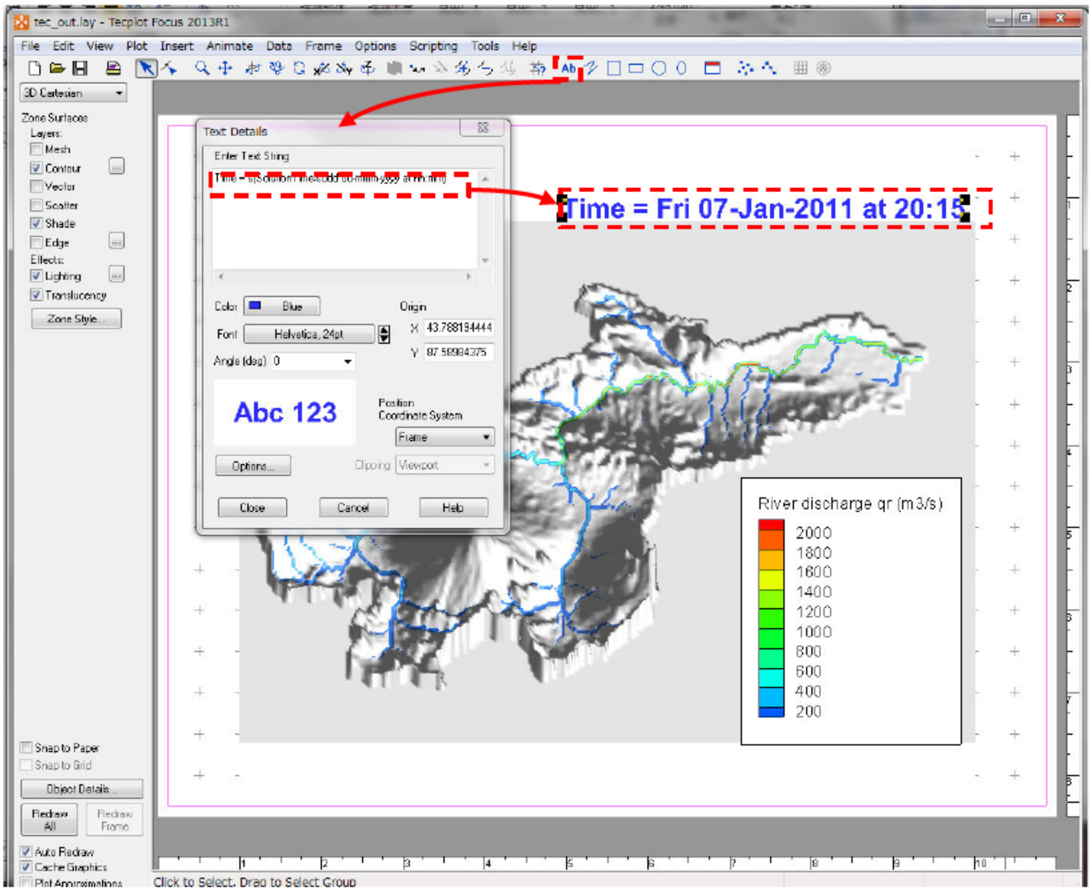
① Display timestamp information on animation

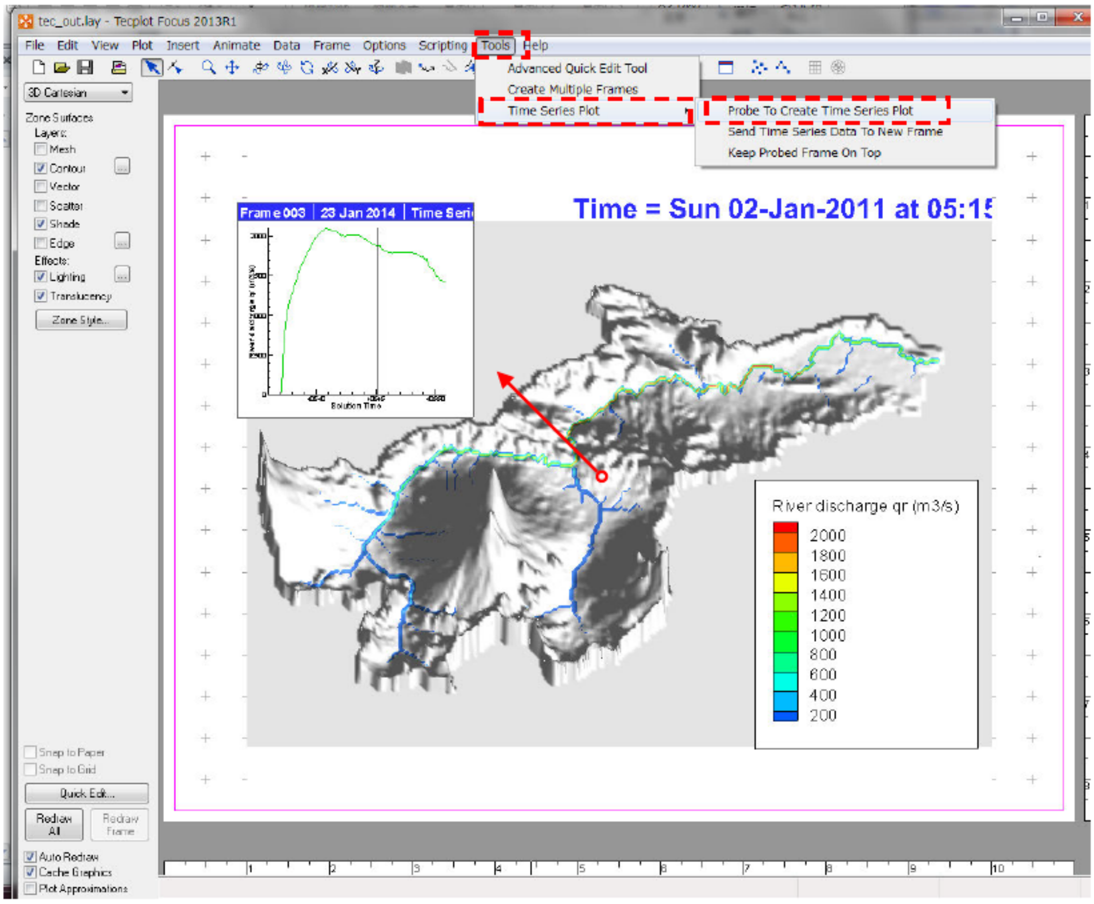
If user needs to display timestamp information on the contour figure, add textbox and input as follows;

Time = &(SolutionTime%ddd dd-mmm-yyyy at hh:mm)

② Display time series graph on plane view.

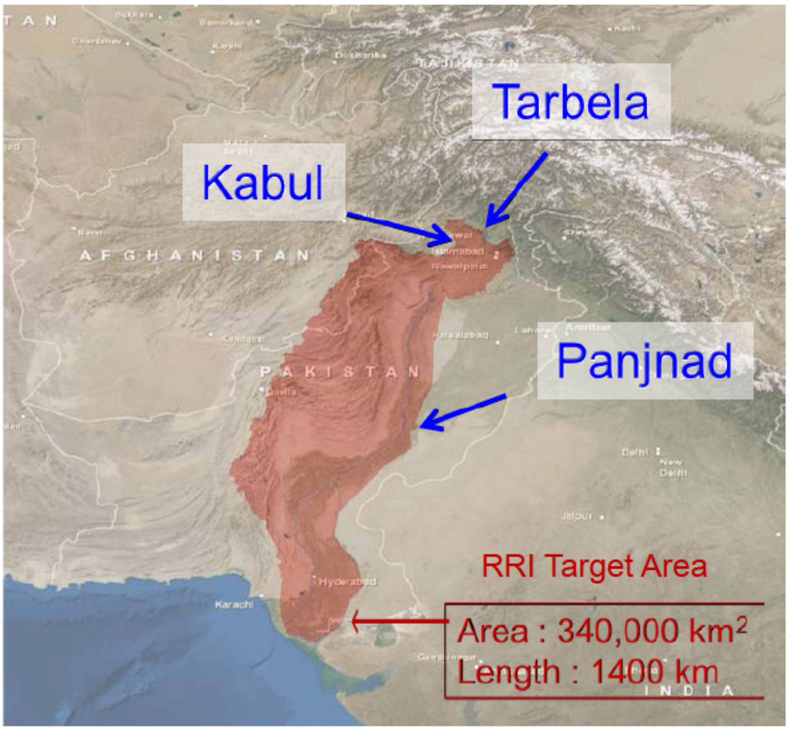
Select [Tools] > [Time Series Plot] > [Probe To Create Time Series Plot] and identify the position by left click with the pointer “+” to display the time-series. Note that the variable selected as “Flood By” will be shown on the time series graph. Hence user needs to change the setting of Zone Style and “Flood By” to display different time-series (e.g. qr).


tec_out.lay - Tecplot Focus 2013R1
File Edit View Plot Insert Animate Data Frame Options Scripting Tools Help
Text Details
Enter Text String
Time = Fri 07-Jan-2011 at 20:15
Abc 123
Close
Cancel
Help
River discharge qr (m3/s)
2000
1800
1600
1400
1200
1000
800
600
400
200
Click to Select. Drag to Select Group


tec_out.lay - Tecplot Focus 2013R1
File Edit View Plot Insert Animate Data Frame Options Scripting Tools Help
Advanced Quick Edit Tool
Create Multiple Frames
Time Series Plot
Probe To Create Time Series Plot
Send Time Series Data To New Frame
Keep Probed Frame On Top
Time = Sun 02-Jan-2011 at 05:15
River discharge qr (m3/s)
2000
1800
1600
1400
1200
1000
800
600
400
200

8. Application Example

This section presents the application of RRI Model to the lower Indus River basin. The target area is below Tarbela dam, Kabul and Panjdnad points as indicated below. The simulation domain is about 340,000 km^2 and the river length is about 1,400 km. In this example, the river discharge boundary conditions are prepared based on observed discharge records during 2010 floods to force the model with rainfall records.

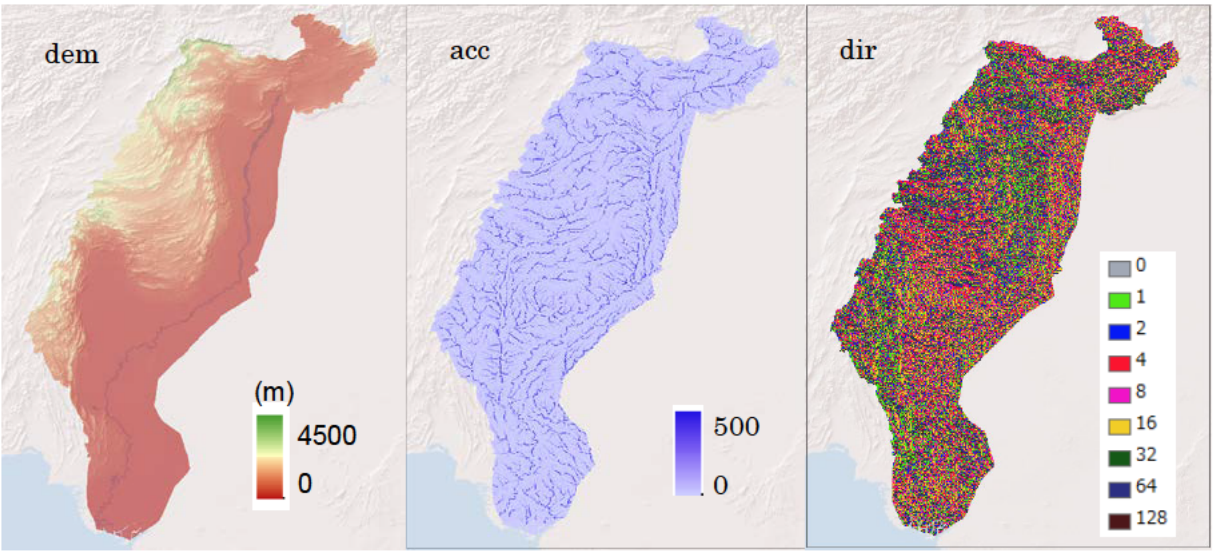


A polygon covering the simulation target (the red mask in the above figure) was prepared first. The flow direction data in HydroSHEDS (30sec) was used to identify the entire Indus River basin. Then the upstream areas above Tarbela, Kabul and Panjnad were removed from the entire Indus River basin.

The background image of the above figure can be obtained from the following site (http://goto.arcgisonline.com/maps/World_Imagery) and used in ArcGIS.

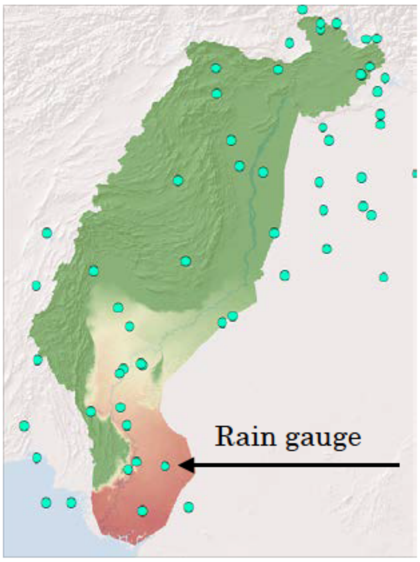
8.1 On Input Topography

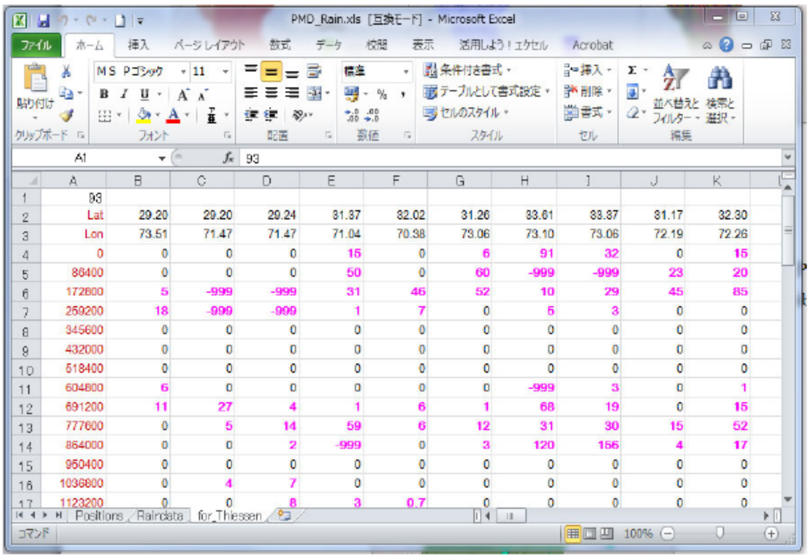
By using the catchment polygon, dem, acc and dir datasets were clipped for the catchment area. The function embedded in ArcGIS ([Spatial Analyst Tools] → [Conditional] → [Con]) was used to mask the target area out of the regional datasets of HydroSHEDS (30 second resolution). Then "demAdjust2" program was used to adjust dem and dir to create adem and adir.



8.2 On Input Rainfall

Ground gauged rainfall records provided by Pakistan Meteorological Department (PMD) were used for the simulation. The green dots in the left figure below show their spatial distribution. The below right figure is the formatted ground gauged rainfall data with the latitude and longitude information. Total 93 data was used to create spatially distributed rainfall data.



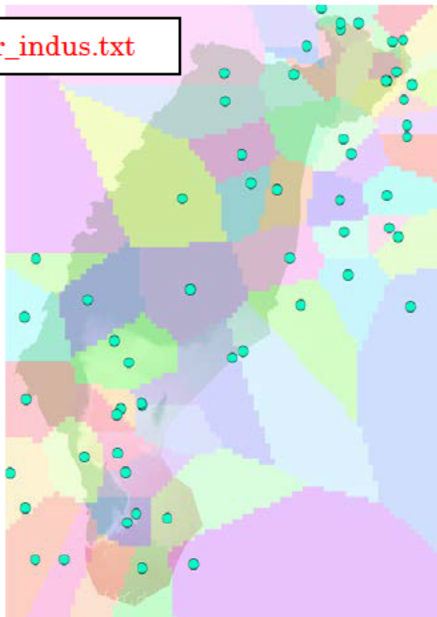


	A	B	C	D	E	F	G	H	I	J	K
1	93										
2	Lat	29.20	29.20	29.24	31.37	32.02	31.26	33.61	33.87	31.17	32.30
3	Lon	73.51	71.47	71.47	71.04	70.38	73.06	73.10	73.06	72.19	72.26
4	0	0	0	0	15	0	6	91	32	0	15
5	86400	0	0	0	50	0	60	-999	-999	23	20
6	172800	5	-999	-999	31	46	52	10	29	45	85
7	259200	18	-999	-999	1	7	0	5	3	0	0
8	345600	0	0	0	0	0	0	0	0	0	0
9	432000	0	0	0	0	0	0	0	0	0	0
10	518400	0	0	0	0	0	0	0	0	0	0
11	604800	6	0	0	0	0	0	-999	3	0	1
12	691200	11	27	4	1	6	1	68	19	0	15
13	777600	0	5	14	59	6	12	31	30	15	52
14	864000	0	0	2	-999	0	3	120	156	4	17
15	950400	0	0	0	0	0	0	0	0	0	0
16	1036800	0	4	7	0	0	0	0	0	0	0
17	1123200	0	0	8	3	0.7	0	0	0	0	0

Note that the first column of the excel sheet represents the time stamp of the rainfall data in second. For example, at the row of 172800 sec, the daily rainfall [mm/d] between time 86400 and 172800 sec was stored. Then all the data was copied to a text editor and save it as ASCII.

The ASCII file is the input data of /etc/rainThiessen program that generates the spatially distributed rainfall data. Note that the "gauge_map_lower_indus.txt" is also created after running /etc/rainThiessen program, so that one can check the spatial representation of each rain gauge (see the figure below after converting from the ASCII to Raster with ArcGIS).

gauge_map_lower_indus.txt



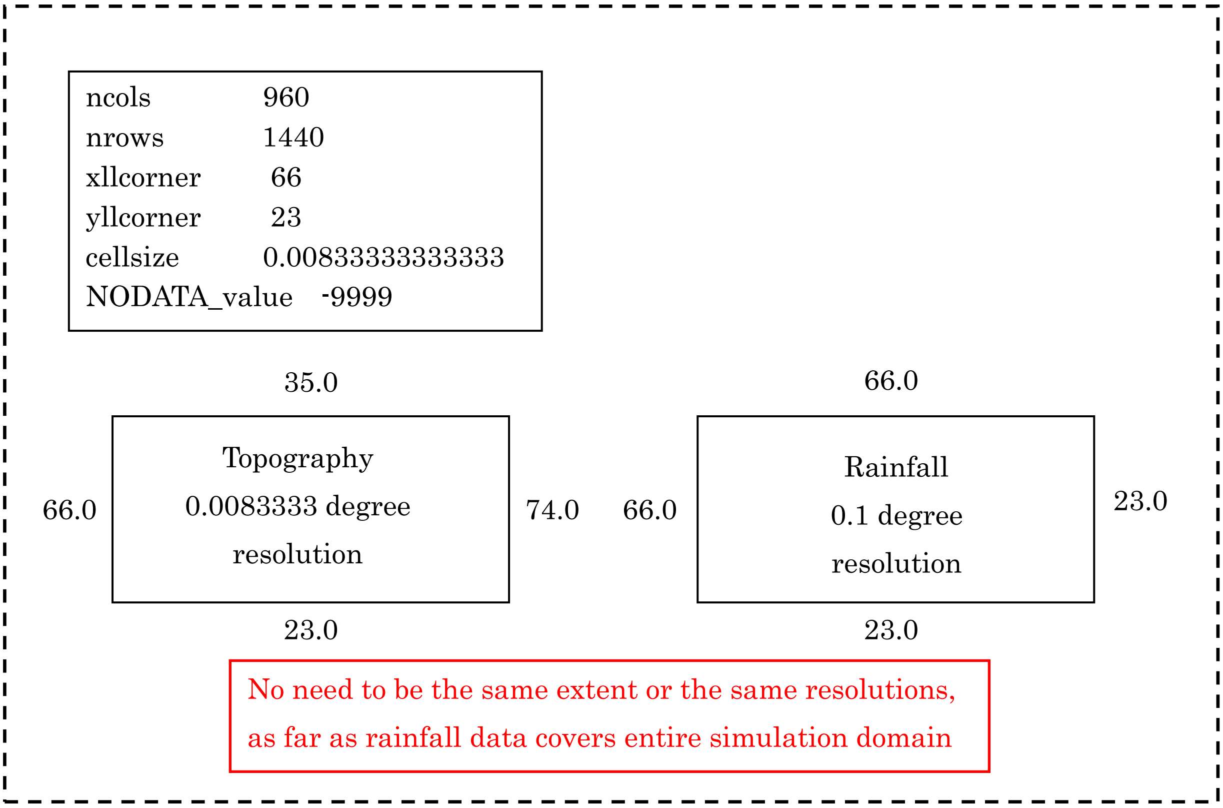
Here is the sample of the rainThiessen program input file (rainThiessen.txt).

rainThiessen.txt

```
./indus/gauge_1d_2010.txt
24
./indus/rain_lower_indus_gauge_2010.dat
./indus/gauge_map_lower_indus.txt
ncols    80
nrows 120
xll 66.0
yll 23.0
cellsize 0.1
```

in degree

The rainfall data must cover all the simulation domain. However, it is not necesssary to have the same resolution or the same coverage area. For exmaple, 0.1 degree (approx. 10 km) may be fine enough to distribute the ground gauged rainfall for this case. Thus above rainThiessen.txt read by the rainThiessen program specifies the output resolution of 0.1 degree.



RRI_Input.txt

```
RRI_Input_Format_Ver1_4_1

./infile/lowerindus/rain/rain_lower_indus_gauge_2010.dat
./infile/lowerindus/adem2_lid1k.txt
./infile/lowerindus/acc_lid1k.txt
./infile/lowerindus/adir_lid1k.txt

0               # utm(1) or latlon(0)
1               # 4-direction (0), 8-direction(1)
2568            # lasth
600             # dt
60              # dt_riv
104             # outnum
66.0d0          # xllcorner_rain
23.0d0          # yllcorner_rain
0.1 0.1         # cellsize_rain
```

xllcorner_rain, yllcorner_rain
cellsize_rain (x, y) are specified
in RRI_Input.txt

8.3 On Input Evapotranspiration

Current version of RRI Model does not have a function to estimate evapotranspiration from climate variables. However, by giving evapotranspiration rate as one of the input files, the model takes the equivalent amount of water from surface and subsurface storages.

The format of the evapotranspiration input is the same as rainfall. Hense the grid cell size and time step of evapotranspiration file can be arbitrary set. For example, to set the constant rate of evapotranspiration, one can prepare the following input file (e.g. evp_4mm.txt), in which the value of 0.166667 mm/h corresponds to 4 mm/d of evapotranspiration.

```
0 1 1
0.166667
10000000 1 1
0.166667
```

evp_4mm.txt

To read the evapotranspiration input file, set flag 1 on the L71 and specify the input file name. The coordinate of south west corner (xllcorner and yllcorner) as well as the cellsize (x and y direction) must be also set in L73-L75.

L71	1	
L72	./infile/lowerindus/evp_4mm.txt	
L73	66.0d0	# xllcorner_evp
L74	23.0d0	# yllcorner_evp
L75	1000.d0 1000.d0	# cellsize_rain

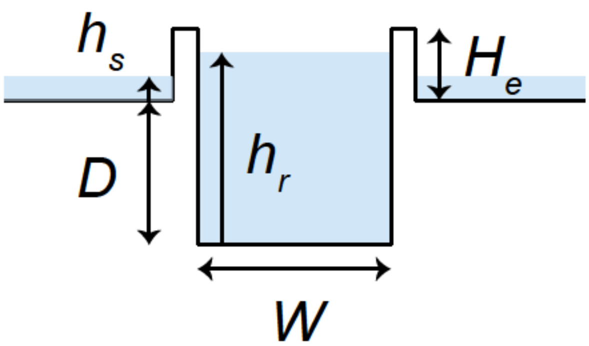
RRI_Input.txt

xllcorner_evp, yllcorner_evp cellsize_rain (x, y) are specified in RRI_Input.txt

Note that if sufficient water exists on a slope grid cell, and if the grid cell store water in the Green Ampt-Model, the model takes water from the cumulative water in GA model. If a user wants to avoid the evapotranspiration from the GA model, use flag "2" instead of "1" on L71.

8.4 On River Channel Geometry Setting

RRI Model assumes the rectangle shape for all river cross sections. To determine river cross sections (incl. width W, depth D and levee height H_e), the following two options are available.



A) Use empirical equations with parameters defined in RRI_Input.txt

B) Read the values from files and specify the files in RRI_Input.txt

RRI_Input.txt

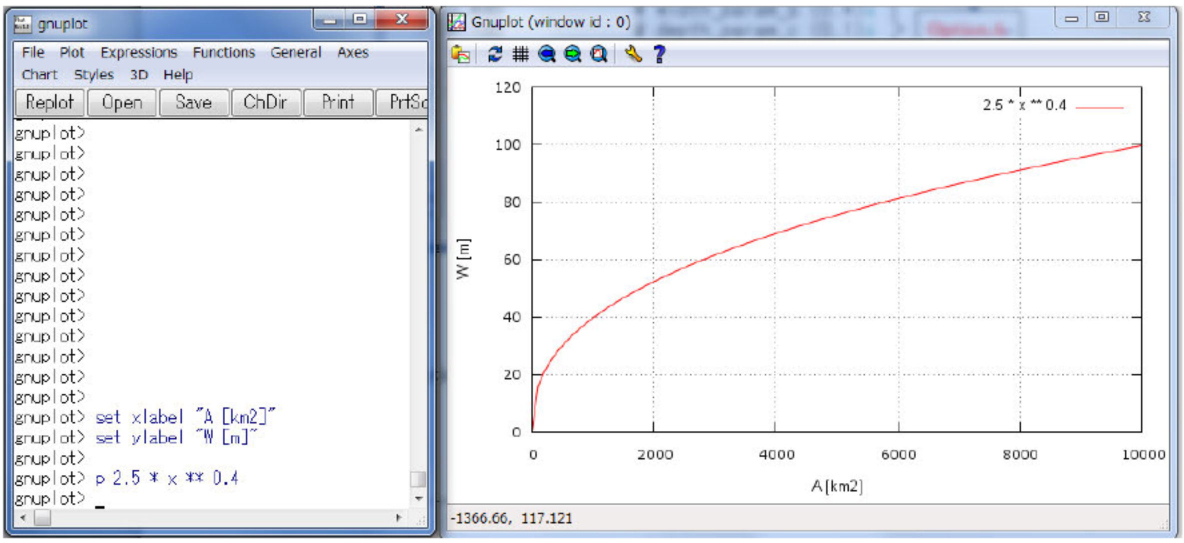
L38	100	# riv_thresh	Option A
L39	2.5d0	# width_param_c	Option A
L40	0.4d0	# width_param_s	Option A
L41	0.1d0	# depth_param_c	Option A
L42	0.4d0	# depth_param_s	Option A
L43	0.d0	# height_param	Option A
L44	20	# height_limit_param	Option A
L45			
L46	1	← 0 : Option A / 1 : Option B (Read from files)	
L47	./infile/lowerindus/width_lid1k.txt		Option B
L48	./infile/ lowerindus /depth_lid1k.txt		Option B
L49	./infile/ lowerindus /height_lid1k.txt		Option B
L50			

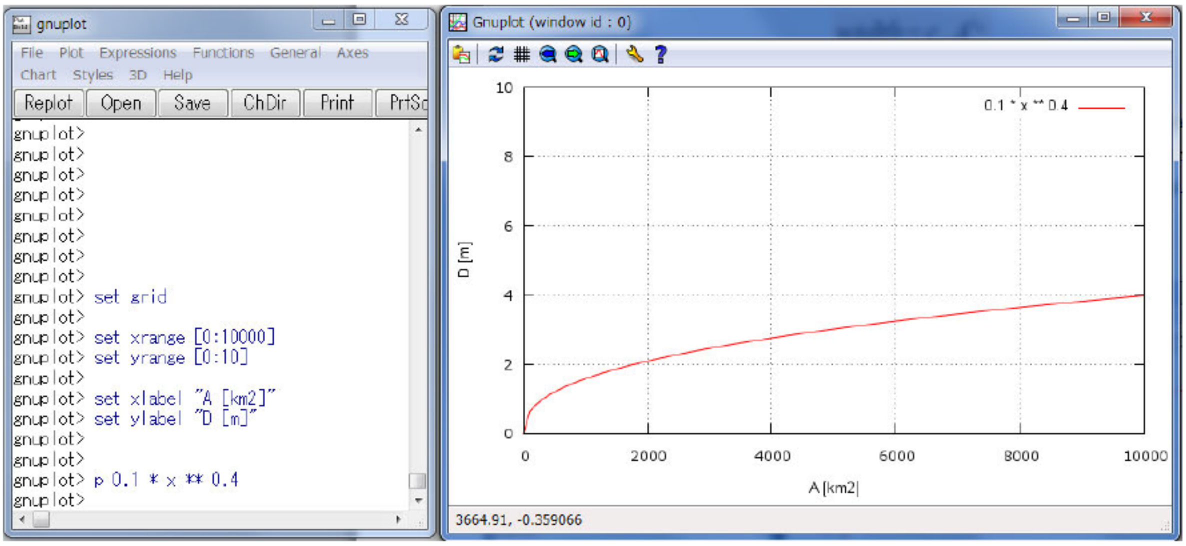
A) For the first option, the parameters of the following empirical equations must be appropriately set to represent target catchment condition (L38 – L44 of RRI_Input.txt).

$$width = c_w A^{s_w}$$

$$depth = c_d A^{s_d}$$

where *A* in the equations is the upstream catchment area [km²] for each river grid-cell. The unit of width and depth are [m]. The parameter "riv_thresh" defines the threshold of flow accumulation (i.e. number of upstream cells) to distinguish river grid cells or slope grid cells. Recall that for RRI model, slope exists even on a river grid cell.





B) For the second option, a user can prepare three files separately to represent width, depth, and height distributions. All those files must have the same number of row, column and resolution as the topography data (i.e. adem, acc and adir). The format of these data is ArcGIS ASCII format (i.e. the same as the topography data).

Note that the width file (e.g. ./infile/lowerindus/width_lid1k.txt) is used to decide whether each grid-cell has river or not (*width* > 0 is treated as a river grid cell). The values of depths and heights must be appropriately defined on a cell where the *width* > 0.

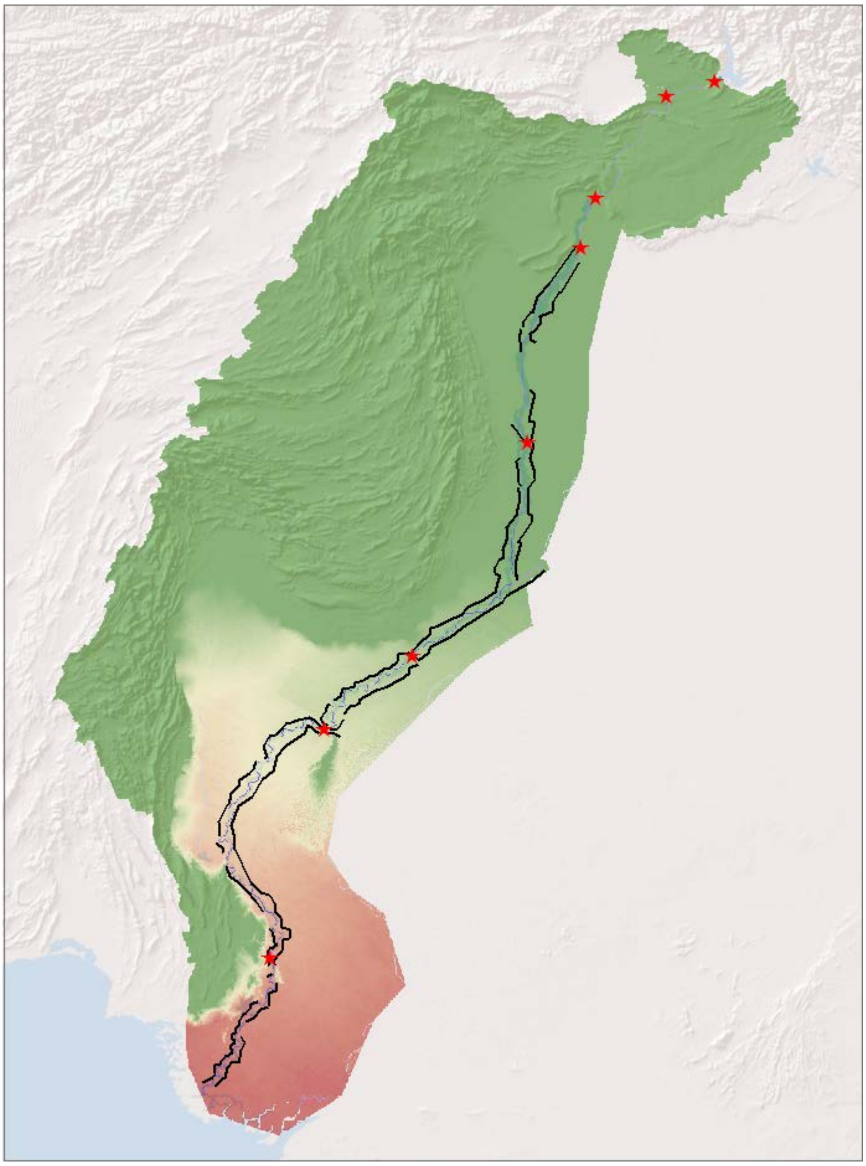
To support for creating the width, depth and height files, a Fortran program called /etc/makeRiver2/ can be used. The program reads "acc" file to calculate the upstream catchment area A [km^2] for each grid cell and a user can define different equations or fixed values within the program to create the three river cross section files.

8.5 On Embankment Setting

There are two kinds of embankment settings in RRI simulation.

- A) Embankment along rivers
- B) Embankment on slope grid cells

A) The first type of embankment is illustrated in the figure of a river cross section. The effect of embankment is considered during the interaction of water between river and slope. To include the first type of embankment, the height value (*height* > 0) must be set on river grid cells (*width* > 0). Because of the RRI Model basic structure, a river is set as a centerline of a slope grid, it is not possible to apply different embankment height for different side of the river for this option.



B) The second type of embankment represents roads, railways or other structures that prevent water to across. Since the embankment along the main Indus River is located a few kilometers apart from the main channel (see above figure), this second type suits better. The location information of the embankment was converted to raster data having the same resolution with topographic data on ArcGIS. The above mentioned "*height*" file specified in RRI_Input.txt can contain the height information (and therefore the embankment location information) on slope grid cells.

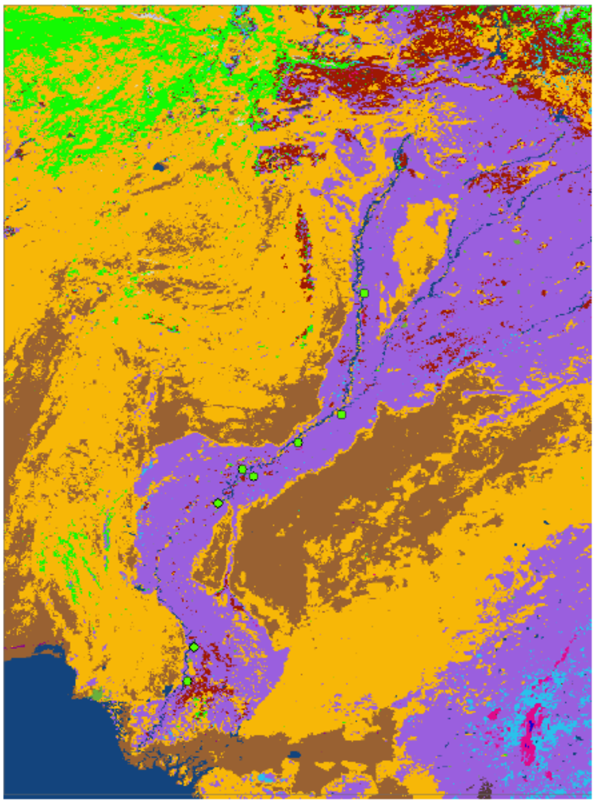
Note that even if a user intends to set a continuous embankment apart from a main river, if a tributary joins into the river and if the "*height*" value is set on a river grid cell where *width* >

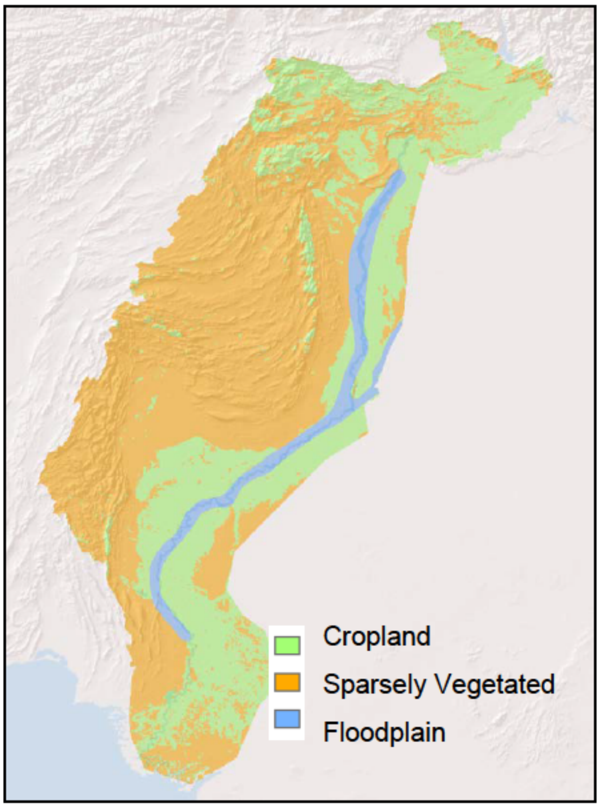
0, the embankment would be regarded as the embankment of Type A. As a result, the set embankment will be discontinuous at the location.

To avoid the situation and elevate DEM even on the tributary (or river grid cells), one can use the flag of “2” on L46.

8.6 On Land Class Setting

The effects of land cover (or soil type) can be reflected by assigning different model parameters. In this example, GLCC-V2 (Global Land Cover Characterization) provided by USGS was used. The original land cover data (left) is too detail to assign all different parameters; therefore, similar land cover types were merged into two categories: Cropland and Sparsely Vegetated, and also overlaid additional Floodplain polygon.





For re-classing the original land cover data, ArcGIS function [Spatial Analyst Tools] → [Reclass] → [Reclass by ASCII File] was used. The following lookup table was prepared by a text editor to define the re-class. Different lookup tables may be defined for different projects. Note that the number of the raster data (in this case 1, 2 and 3) corresponds to the column of parameter sets in RRI_Input.txt. Thus provide sequential numbers starting from 1 for representing different land covers.

Finally the re-classed land cover was converted to the ArcGIS/ASCII format and saved it as "lu_lid1k.txt". Note that the file can be read by RRI Model by indicating the file link in "RRI_Input.txt".

Lookup Table (Indus Example)

1 : 1
2 : 1
3 : 1
4 : 1
5 : 1
6 : 1
7 : 1
8 : 2
9 : 1
10 : 1
11 : 1
12 : 1
13 : 1
14 : 1
15 : 1
16 : 1
17 : 1
18 : 1
19 : 2
20 : 1
21 : 1
22 : 1
23 : 1
24 : 1
99 : 1
100 : 1

Legend of GLCC-V2

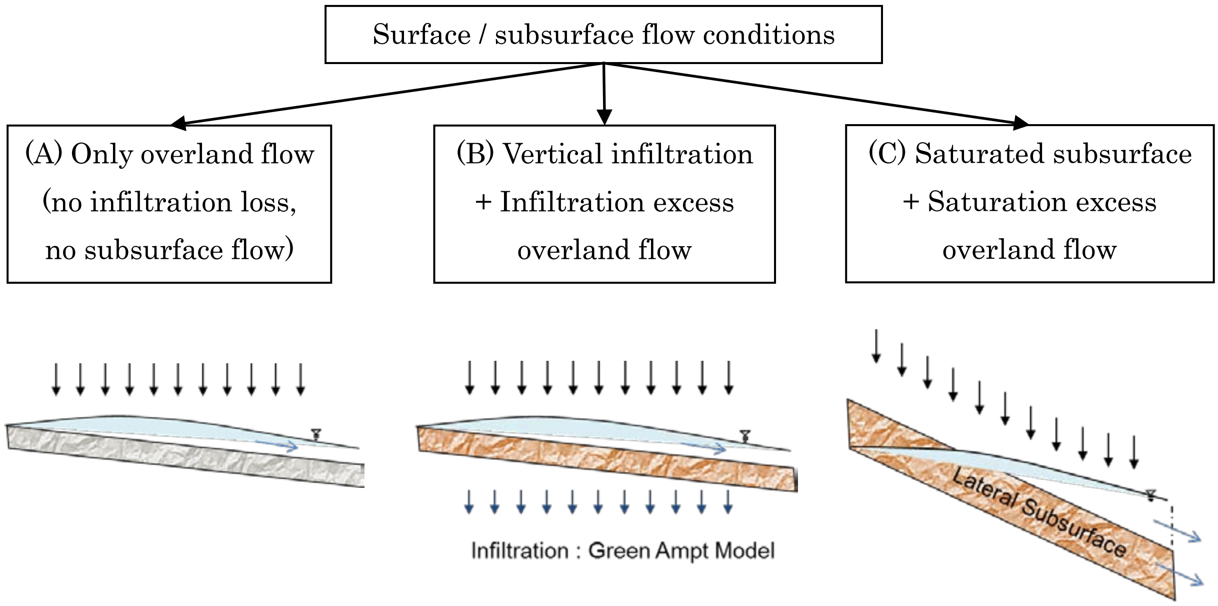
USGS Land Use/Land Cover System Legend (Modified Level 2)

Value	Description
1	Urban and Built-Up Land
2	Dryland Cropland and Pasture
3	Irrigated Cropland and Pasture
4	Mixed Dryland/Irrigated Cropland
5	Cropland/Grassland Mosaic
6	Cropland/Woodland Mosaic
7	Grassland
8	Shrubland
9	Mixed Shrubland/Grassland
10	Savanna
11	Deciduous Broadleaf Forest
12	Deciduous Needleleaf Forest
13	Evergreen Broadleaf Forest
14	Evergreen Needleleaf Forest
15	Mixed Forest
16	Water Bodies
17	Herbaceous Wetland
18	Wooded Wetland
19	Barren or Sparsely Vegetated
20	Herbaceous Tundra
21	Wooded Tundra
22	Mixed Tundra
23	Bare Ground Tundra
24	Snow or Ice
99	Interrupted Areas
100	Missing Data

8.7 On Parameter Setting

Model parameter values are defined in RRI_Input.txt. In this section, the general idea to decide model parameters are described first, then a calibrated model parameter set for the Indus River basin will be shown as an example.

For each land cover class, decide (A), (B) or (C) in the following figure depending on infiltration and subsurface processes, so that the number of calibration parameters will be limited.



Example of parameter values (their recommended ranges)

Parameters	Notation	(A)	(B)	(C)
n (River) ($m^{-1/3}s$)	ns_river		0.03d0 (0.015 ~ 0.04)	
n (Land) ($m^{-1/3}s$)	ns_slope		0.3 d0 (0.15 ~ 1.0)	
Soil depth (m)	soildepth		1.0 d0 (0.5 ~ 2.0)	
Porosity (ϕ) (-)	gammaa		0.471d0 (0.3 ~ 0.5)	
k_{sv} (m/s)	ksv	0.d0	5.56d-7	0.d0
S_f	faif	inactive	0.273d0	inactive
k_a (m/s)	ka	0.d0	0.d0	0.1d0 (0.01-0.3)
Unsat. porosity (-)	gammam		Inactive	0.d0
β	beta	inactive	Inactive	inactive

Note: 0.d0 is used in RRI_Input.txt to represent double precision of 0.0.

For case (A), where only overland flow without infiltration or subsurface flow process are considered, set both ksv and ka equal to 0.

For case (B), where vertical infiltration + infiltration excess overland flow are considered, set ka = 0, and the parameter "da" is equal to "soil depth" times "porosity".

For case (c), where saturated subsurface + saturation excess overland flow are considered, set ksv = 0, and the infiltration limit (defined as a parameter in the previous versions of the RRI model) equals to "soil depth" times "porosity".

Note that the parameter values in the above table are just one example values (approximate ranges).

Note that even though the values in inactive part do not affect the simulation result, a double precision value like 0.0d0 must be filled in RRI_Input.txt (see the sample below).

* Set "kgv = 0.d0" to avoid groundwater computation, whose algorithm is under development and not completed at RRI ver1.4.1β.

** If both ka and ksv are set to be non-zero, RRI will stop with an error message.

The following figure shows an example of parameter settings

RRI_Input.txt

```
L18 | 0.03d0                      # ns_river
L19 | 3                           # num_of_landuse
L20 | 1 1 1                       # diffusion(1) or kinematic(0)
L21 | 0.15d0 0.15d0 0.15d0        # ns_slope
L22 | 1.0d0 1.0d0 1.0d0           # soildepth
L23 | 0.4d0 0.4d0 0.4d0           # gammaa
L24 |
L25 | 5.556d-7 6.056d-7 0.d0      # ksv
L26 | 0.273d0 0.1101d0 0.d0       # faif
L27 |
L28 | 0.1d0 0.1d0 0.1d0           # ka
L29 | 0.0d0 0.0d0 0.0d0           # gammam
L29 | 8.0d0 8.0d0 8.0d0           # beta
L31 |
L32 | 0.d0 0.d0 0.d0              # kgv
L33 - L36 are inactive under kgv = 0.d0
```

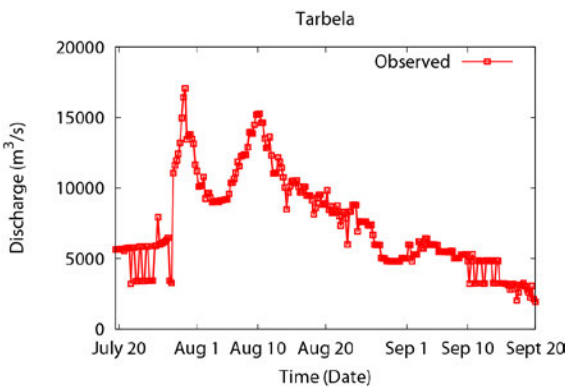
Reference Table : Green-Ampt Infiltration Parameters for different soil texture

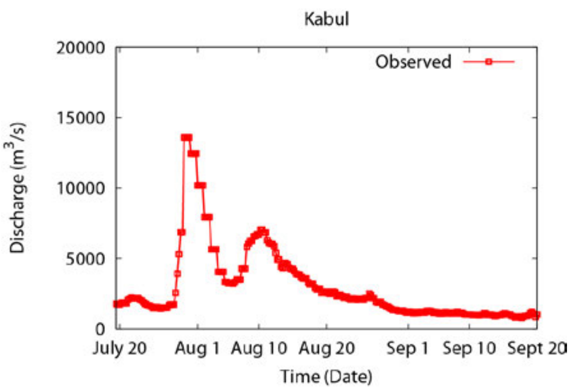
Soil texture class	k_{sv} (m/s)	ϕ [gammaa]	S_f (m) [faif]
Sand	6.54E-05	0.437	0.0495
Loamy sand	1.66E-05	0.437	0.0613
Sandy loam	6.06E-06	0.453	0.1101
Loam	3.67E-06	0.463	0.0889
Silt loam	1.89E-06	0.501	0.1668
Sandy clay loam	8.33E-07	0.398	0.2185
Clay loam	5.56E-07	0.464	0.2088
Silty clay loam	5.56E-07	0.471	0.273
Sandy clay	3.33E-07	0.43	0.239
Silty clay	2.78E-07	0.479	0.2922
Clay	1.67E-07	0.475	0.3163

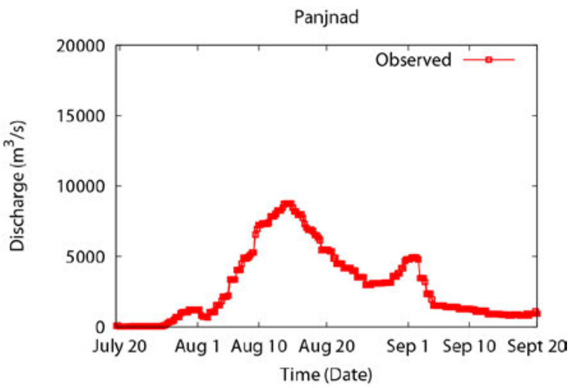
From Rawls, W.J. et al.., 1992. Infiltration and soil water movement. In: Handbook of hydrology. New York: McGrow-Hill Inc., 5.1–5.51. (Units are conveted for RRI Model)

8.8 On Boundary Condition

The following river boundary conditions were set based on the observed discharges at the three locations during the 2010 flood.





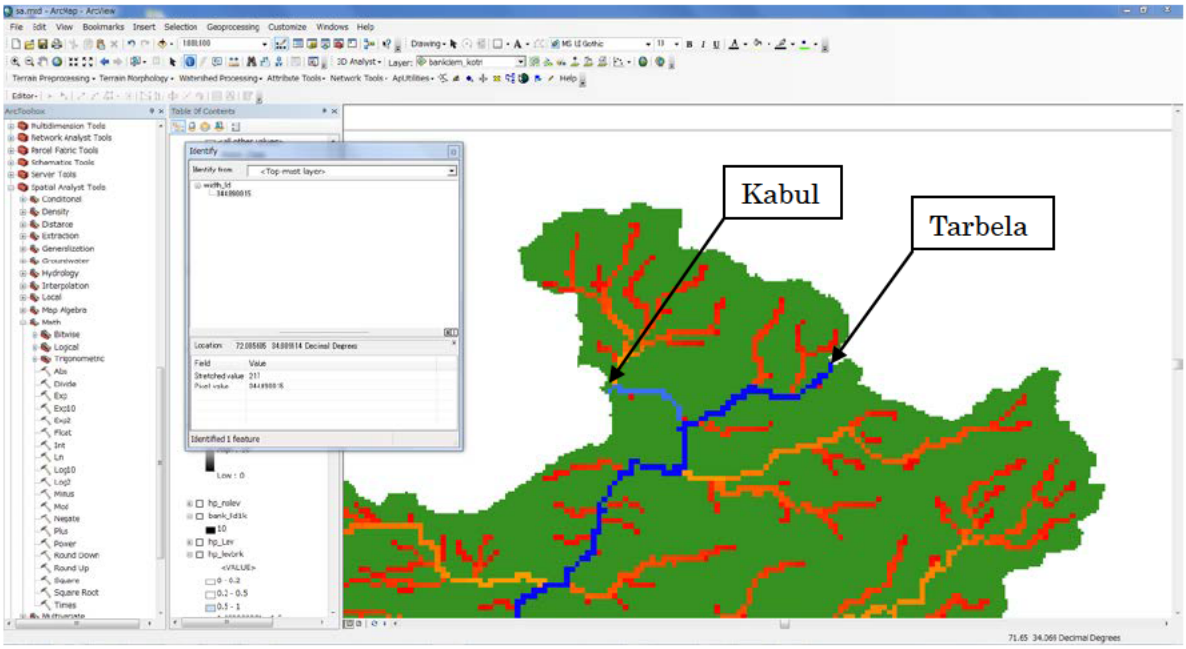


The steps to set river discharge boundary conditions are described below.

① Find locations to provide the boundary conditions.

Viewing acc values on ArcGIS can help you to identify appropriate position with lat lon information along a river channel. Use i (identify) icon to find out the coordinate.

Then use the "/etc/coordinate.xls" to convert from the lat lon coordinate to loc_i and loc_j. See Section 7.3 on the conversion in detail.



② Prepare a 1D boundary condition file with the following format.

The number of boundary condition setting points

The loc_i and loc_j of all points to give boundary conditions

3				
loc_i	110	119	680	
loc_j	803	719	602	
0	3936.041733	3007.249151	917.4658428	
21600	3936.041733	3007.249151	917.4658428	
43200	3936.041733	3007.249151	917.4658428	
64800	3879.408039	3044.061053	917.4658428	
86400	4813.86399	3015.744206	917.4658428	
108000	4700.596602	2944.952088	917.4658428	
129600	4842.180837	2899.645133	917.4658428	
151200	4672.279755	2922.29861	1093.030294	
…..	…..	….	….	

Time series of the boundary condition data (units: [m³/s] for river discharge, and [m] for water depth)

③ Settings in RRI_Input.txt

After preparing the boundary conditon file (e.g. disc_lid1k_2010.txt) and move the file in the appropriate folder (e.g ./infile/lowerindus/), edit the RRI_Input.txt file as follows.

RRI_Input.txt

```
L51  0 0 0 0
L52  ./infile/lowerindus/hs_init_dummy.out
L53  ./infile/lowerindus/hr_init_lid1k.txt
L54  ./infile/lowerindus/hg_init_sample.out
L55  ./infile/lowerindus/gampt_ff_init_dummy.out
L56
L57  0 0
L58  ./infile/wlev_bound_dummy.txt
L59  ./infile/hr_bound_dummy.txt
L60
L61  0 1
L62  ./infile/qs_bound_dummy.txt
L63  ./infile/lowerindus/bounds/bound_lid_2010.txt
L64
```

Write the file name of river discharge boundary

Another option is to use two-dimension format for setting boundary conditons. In that case, prepare the following "setBound.txt" first as the input file to "/RRI/etc/setBound" program, which creates the input boundary conditon file (e.g. ./disc_lid1k_2010.txt) on two dimensional basis. The two-dimensional boundary condition files can be read with flag 2 on L61.

setBound.txt

```
../../Model/infile/lowerindus/adem_lid1k.txt
../../Model/infile/lowerindus/acc_lid1k.txt
../../Model/infile/lowerindus/adir_lid1k.txt
./infile/lowerindus/disc_Constant.txt
../../Model/infile/lowerindus/disc_lid1k_Constant.txt
3
119 719
110 803
680 602
```

In the above example of "setBound.txt", L1 to L3 are the paths to the topography files (dem, acc and dir). L4 is the 1D discharge file (input) prepared above and the L5 is the output of the setBound program. L6 indicates the number of points to give the boundary conditions, followed by the positions in loc_i and loc_j.

The created boundary condition files have the same format as the rainfall file. However, unlike rainfall files, the number of columns and rows must be exactly the same as the topography data, so that RRI Model knows where to give the boundary.

Note that discharge boundary conditions including along river and on slope must have the information of the directions. In other words, they should be vector values rather than the scalar values. To decide the direction of the discharge boundary conditions, RRI Model refers to the flow direction in "dir" file.

Water level boundary conditions on slope and/or river can be also set by changing the value on L57 to 1 and specitying the boundary condition file name. The file format is the same as the river discharge boundary condion.

8.9 On Initial Condition

RRI Model can take initial conditions for water depths on slope and river as well as the cumulative water depth in the Green-Ampt model. The format of the files is the same as the output of those variables, so that one can use the output of the RRI as the input for the next simulation.

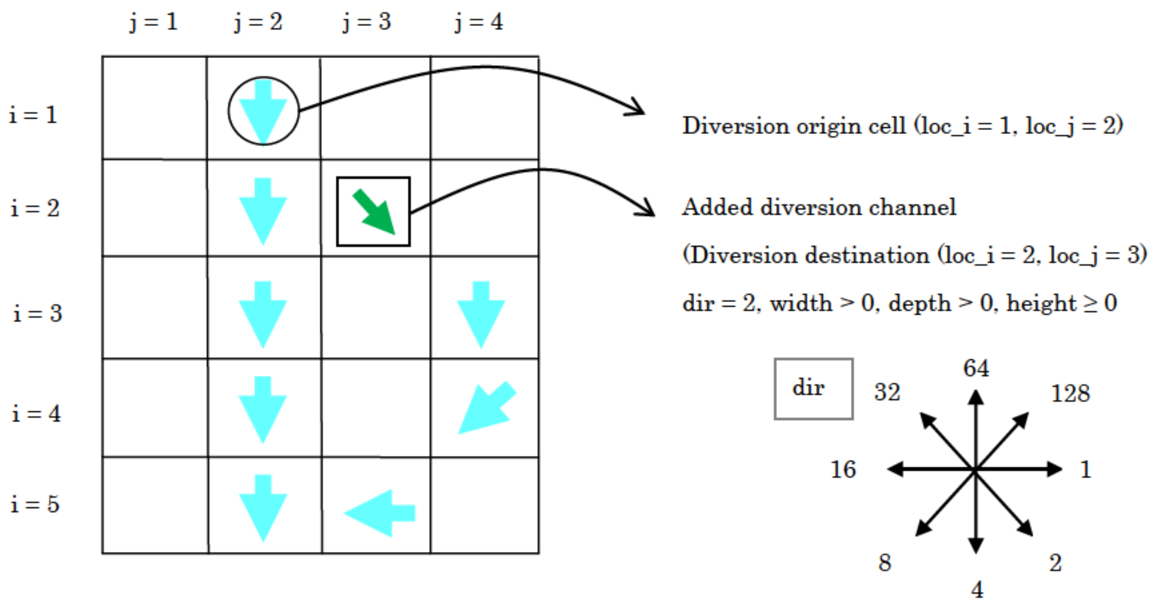
This feature enables the continuous long-term simulation. In order to read the initial conditions, L49 to L52 in the RRI_Input.txt must be edited in a same manner as the example of the boundary condition setting.

8.10 Diversion option (for advanced users)

RRI model can simulate the effect of diversion in a simple way. The portion of the diversion from a main channel to a diversion channel must be pre-defined by a model user and described in RRI_Div.f90 program. The followings are the basic steps to activate the option.

① Edit input river cross section files (i.e. width, depth, height) and flow direction files to add necessary diversion channels (e.g green arrow for the below figure).

② Check a origin cell (loc_i_org, loc_j_org) and a destination cell (loc_i_dest, loc_j_dest). Both the origin and destination cells must be specified on river grid-cells. Typically these two are adjacent, but not necessary (i.e. diverted water can jump into an apart cell).



③ Prepare a file to specify the origin and destination cells based on the following format. One can list up multiple lines if more than one diversion should be considered.

div_sample.txt

```
1    2    2    3
```

loc_i_org	loc_j_org	loc_i_dest	loc_j_dest

④ Edit "RRI_Div.f90" source code. The simplest way is to decide a ratio to divert from a main channel to a diversion channel. Be sure to recompile the program once the souce code is modified. The part of "RRI_Div.f90" program that users may edit is as follows.

RRI_Div.f90

```
do l = 1, div_id_max
 k = div_org_idx(l)
 kk = down_riv_idx(k)
 kk_div = div_dest_idx(l)
 if( l.eq.1 ) then
  if( qr_idx(k) .gt. 0.d0 ) then
   qr_div_idx(k) = qr_idx(k) * 0.1
   qr_idx(k) = qr_idx(k) - qr_div_idx(k)
  endif
 endif
enddo
```

In this program, the variable "div_id_max" is defined automatically as the number of the diversions (i.e. the number of lines in "div_sample.txt"). Users need to edit the above blue lines to define the portion (or other rule) for each diversion. Add another set of lines for another diversion started with if (l.eq.2) etc. For the above example, 10% of flow from the main stream at (loc_i = 1, loc_j = 2) is diverted to the stream at (loc_i = 2, loc_j = 3).

⑤ Activate this option by setting flag 1 on L70 and specify the diversion file name on L71 in RRI_Input.txt.

8.11 Dam option (for advanced users)

RRI model can simulate the effect of dam reservoirs in a simple way. The dam model has two parameters: outflow discharge and maximum storage volume. The model takes storage volume as a state variable, which continues being updated based on simulated inflow and outflow. The outflow is maintained at a certain discharge rate that is lower than the inflow rate until the storage volume reaches the dam's maximum storage level. After the storage volume exceeds the maximum level, the model is designed to release the water at the same rate as the inflow rate. The parameters must be determined based on dam operation records. The followings are the basic steps to activate the dam model.

① Prepare a dam parmmeter file by the following format.

dam_sample.txt

```
2
Bhumibol    166    71     5800000000    150
Sirikit     135    166    3510000000    500
```

dam names, loc_i_dam, loc_j_dam, storage volume [m^3], constant discharge [m^3/s]

② Activate the dam model by setting flag 1 on L65 and specify the dam file name on L66 in RRI_Input.txt.

土木研究所資料
TECHNICAL NOTE of PWRI
No.4277 February 2014

編集・発行 ©独立行政法人土木研究所

本資料の転載・複写の問い合わせは

独立行政法人土木研究所 企画部 業務課
〒305－8516 茨城県つくば市南原１－６ 電話 029－879－6754