Rainfall-Runoff-Inundation (RRI) Model

ver. 1.4.2.7

International Center for Water Hazard and Risk Management (ICHARM)
Public Works Research Institute (PWRI)

*)Disaster Prevention Research Institute (DPRI), Kyoto University

Takahiro SAYAMA*)

Rainfall-Runoff-Inundation Model User's Manual

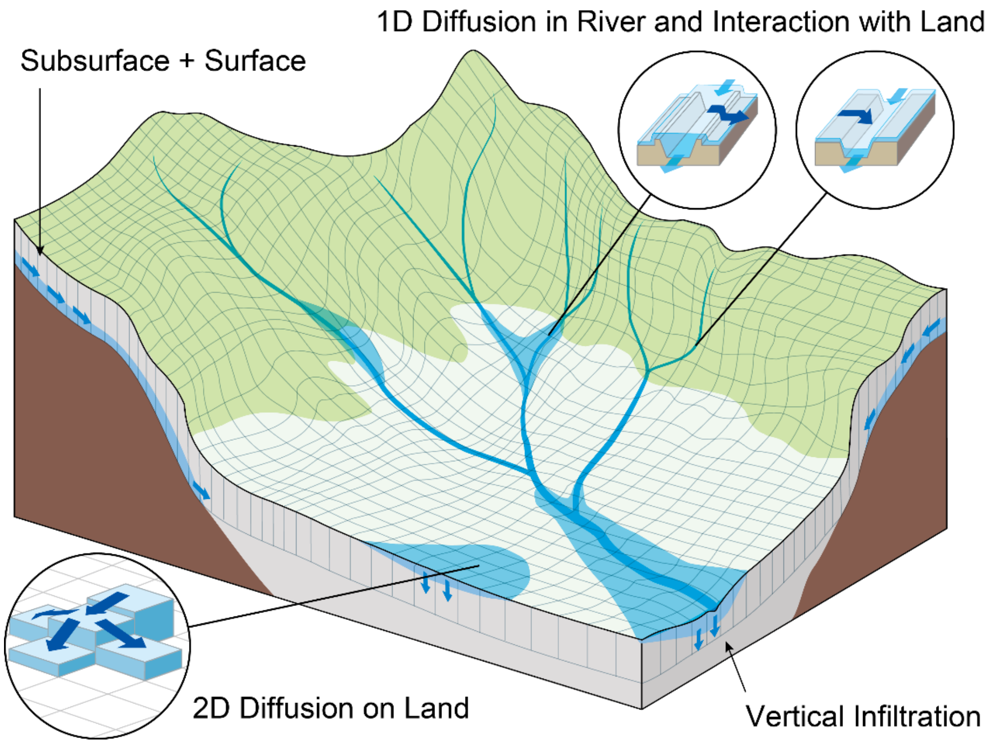
Table of Contents

Last updated on Dec. 7, 2022

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, Sayama et al., 2015a, Sayama et al., 2015b). 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) RRI is a 2D model simulating for rainfall-runoff and flood inundation simultaneously.
2) It simulates flows on land and in river and their interactions at a river basin scale.
3) It simulates lateral subsurface flow in mountainous areas and infiltration in flat areas.

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 specified as follows.

$$\frac{\tau_x}{\rho_w} = \frac{\tau_b}{\rho_w}\frac{u}{\sqrt{u^2+v^2}} \quad (4)$$

$$\frac{\tau_y}{\rho_w} = \frac{\tau_b}{\rho_w}\frac{v}{\sqrt{u^2+v^2}} \quad (5)$$

where τ_b is the bed shear stress defined as

$$\tau_b = \rho_w ghi \quad (6)$$

where i is the friction slope. Substitution of (6) and the Manning's formula into (4) and (5) yields

$$\frac{\tau_x}{\rho_w} = \frac{gn^2u\sqrt{u^2+v^2}}{h^{1/3}} \quad (8)$$

$$\frac{\tau_y}{\rho_w} = \frac{gn^2v\sqrt{u^2+v^2}}{h^{1/3}} \quad (9)$$

where n is the Manning's roughness coefficient. Using the relation of $\sqrt{u^2+v^2} = \frac{1}{n}i^{1/2}h^{2/3}$

and $q_x = uh$ and $q_y = vh$ for (8) and (9), we obtain

$$\frac{\tau_x}{\rho_w} = ngh^{-2/3} i^{1/2} q_x \quad (10)$$

$$\frac{\tau_y}{\rho_w} = ngh^{-2/3} i^{1/2} q_y \quad (11)$$

and i is defined as

$$i = \sqrt{\left(\frac{\partial H}{\partial x}\right)^2 + \left(\frac{\partial H}{\partial y}\right)^2} \quad (12)$$

The diffusion wave approximation, which omits the left side of (2) and (3), by substituting (10) and (11) into the shear stress terms yields

$$q_x = -\frac{1}{n} h^{5/3} \frac{\partial H}{\partial x} \frac{1}{i^{1/2}} \quad (13)$$

$$q_y = -\frac{1}{n} h^{5/3} \frac{\partial H}{\partial y} \frac{1}{i^{1/2}} \quad (14)$$

where q_x and q_y are the unit width discharges.

Note that the previous version of the RRI manual and the related papers described the following equations (15) and (16) to compute q_x and q_y. The equations (13) and (14) were derived based on the independent computations of q_x and q_y (e.g. Hunter et al. 2007) and used for the RRI model allowing flows in the diagonal directions with the octagonal grid-cells.

$$q_x = -\frac{1}{n} h^{5/3} \sqrt{\left|\frac{\partial H}{\partial x}\right|}\, sgn\left(\frac{\partial H}{\partial x}\right) \quad (15)$$

$$q_y = -\frac{1}{n} h^{5/3} \sqrt{\left|\frac{\partial H}{\partial y}\right|}\, sgn\left(\frac{\partial H}{\partial y}\right) \quad (16)$$

where *sgn* is the signum function. The RRI model spatially discretizes mass balance equation (1) as follows:

$$\frac{dh^{i,j}}{dt} + \frac{q_x^{i-1,j} - q_x^{i,j}}{\Delta x} + \frac{q_y^{i,j-1} - q_y^{i,j}}{\Delta y} = r^{i,j} - f^{i,j} \quad (17)$$

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 (1), (13) and (14), 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(13) and (14) with the following equations of

(18) and (19), 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 (18) and (19) ($h \leq d_a$) describe the saturated subsurface flow based on the Darcy law, while the second equations ($d_a \leq h$) 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.

$$q_x = \begin{cases} -k_a h \dfrac{\partial H}{\partial x} & (h \leq d_a) \\ -\dfrac{1}{n}(h-d_a)^{5/3} \dfrac{\partial H}{\partial x} \dfrac{1}{i^{1/2}} - k_a h \dfrac{\partial H}{\partial x} & (d_a < h) \end{cases} \quad (18)$$

$$q_y = \begin{cases} -k_a h \dfrac{\partial H}{\partial y} & (h \leq d_a) \\ -\dfrac{1}{n}(h-d_a)^{5/3} \dfrac{\partial H}{\partial y} \dfrac{1}{i^{1/2}} - k_a h \dfrac{\partial H}{\partial y} & (d_a < h) \end{cases} \quad (19)$$

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

Equations (20) and (21) 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).

$$q_x = \begin{cases} -k_m d_m \left(\dfrac{h}{d_m}\right)^{\beta} \dfrac{\partial H}{\partial x} & (h \leq d_m) \\ -k_a(h-d_m)\dfrac{\partial H}{\partial x} - k_m d_m \dfrac{\partial H}{\partial x} & (d_m < h \leq d_a) \\ -\dfrac{1}{n}(h-d_a)^{5/3} \dfrac{\partial H}{\partial x} \dfrac{1}{i^{1/2}} - k_a(h-d_m)\dfrac{\partial H}{\partial x} - k_m d_m \dfrac{\partial H}{\partial x} & (d_a < h) \end{cases} \quad (20)$$

$$q_y = \begin{cases} -k_m d_m \left(\dfrac{h}{d_m}\right)^{\beta} \dfrac{\partial H}{\partial y} & (h \leq d_m) \\ -k_a(h-d_m)\dfrac{\partial H}{\partial y} - k_m d_m \dfrac{\partial H}{\partial y} & (d_m < h \leq d_a) \\ -\dfrac{1}{n}(h-d_a)^{5/3} \dfrac{\partial H}{\partial y} \dfrac{1}{i^{1/2}} - k_a(h-d_m)\dfrac{\partial H}{\partial y} - k_m d_m \dfrac{\partial H}{\partial y} & (d_a < h) \end{cases} \quad (21)$$

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).

$$f = k_v \left[1 + \frac{(\varphi - \theta_i)S_f}{F}\right] \quad (22)$$

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 (18) and (19) (or (20) and (21)) can be used with setting f equals to be zero. (Note that the equations (18) and (19) (or (20) and (21)) 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 (13) and (14) can be used with the consideration of vertical infiltration by equation (22). 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. (13) and (14) are used when k_a and k_m are zero, (18) and (19) are used when k_m is zero, and (22) 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].

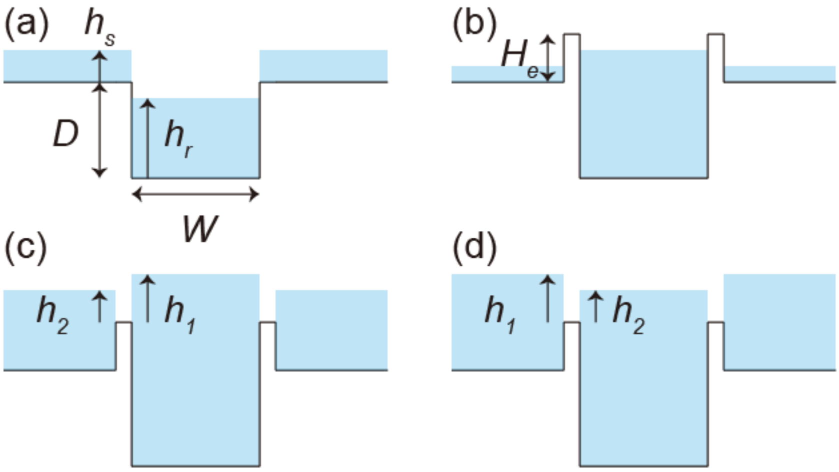
$$W = C_W A^{S_W} \quad (23)$$

$$D = C_D A^{S_D} \quad (24)$$

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.

$$q_{sr} = \mu_1 h_s \sqrt{g h_s} \quad (25)$$

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.

$$q_{rs} = \begin{cases} \mu_2 h_1 \sqrt{2 g h_1} & h_2/h_1 \leq 2/3 \\ \mu_3 h_2 \sqrt{2g(h_1 - h_2)} & h_2/h_1 > 2/3 \end{cases} \quad (26)$$

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 (26) 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 (13), (14) and (17), 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

$$k_1 = \Delta t f(t, h_t)$$

$$k_2 = \Delta t f(t + a_2 \Delta t, h_t + b_{21} k_1)$$

$$\cdots$$

$$k_6 = \Delta t f(t + a_6 \Delta t, h_t + b_{61} k_1 + \cdots b_{65} k_5)$$

$$h_{t+1} = h_t + c_1 k_1 + c_2 k_2 + c_3 k_3 + c_4 k_4 + c_5 k_5 + c_6 k_6 + O(\Delta t^6) \quad (27)$$

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

$$h_{t+1}^* = h_t + c_1^* k_1 + c_2^* k_2 + c_3^* k_3 + c_4^* k_4 + c_5^* k_5 + c_6^* k_6 + O\left({\Delta_t}^5\right) \quad (28)$$

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

$$\delta \equiv h_{t+1} - h_t^* = \sum_{i=1}^{6} (c_i - c_i^*) k_i \quad (29)$$

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}).

$$\Delta t_{post} = max\left(0.9 \Delta t \left|\frac{\delta_d}{\delta}\right|^{0.25}, 0.5 \Delta t\right) \quad (30)$$

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.

- Sayama, T., Tatebe, Y., Tanaka, S. (2015a) An emergency response-type rainfall-runoff-inundation simulation for 2011 Thailand floods, *Journal of Flood Risk Management*, doi:10.1111/jfr3.12147 (in print).

- Sayama, T., Tatebe, Y., Iwami, Y., Tanaka, S. (2015b) Hydrologic sensitivity of flood runoff and inundation: 2011 Thailand floods in the Chao Phraya River basin, *Nat. Hazards Earth Syst. Sci.*, 15, pp. 1617-1630, doi:10.5194/nhess-15-1617-2015.
- 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.

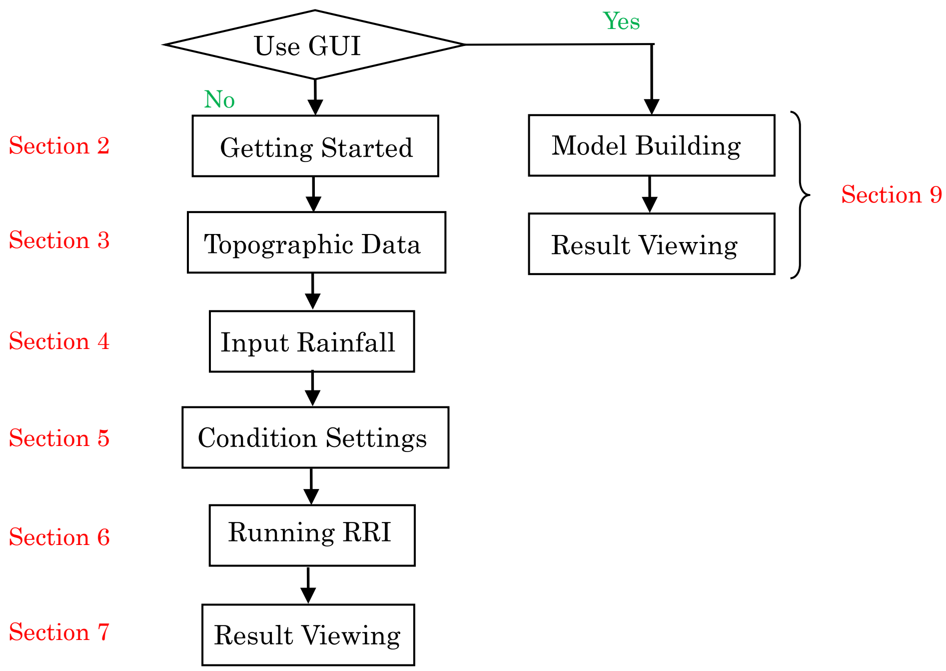
2. Getting Started

RRI model and related tools were originally developed with Fortran 90 computer language. The model has been operated on Command User Interface (CUI) such as Command Prompt on Windows. Since 2014, RRI-Graphical User Interface (GUI) has been also developed to support users for efficient model building and result visualization.

For non-experts in hydrologic modeling, it is recommended to use RRI-GUI to begin with by referring to Section 9 to learn the basic steps with RRI-GUI.

Refer to Section 2 (i.e. this following chapter) on the tutorial of RRI-CUI, followed by more detail descriptions in Sections 3 to 7. Section 8 shows an application example including some advanced model settings.

<u>If you use RRI-GUI not RRI-CUI, you can skip the following steps. Go directly to Section 9.</u>



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 Preparation for the use of RRI-CUI

1) Unzip "RRI_1_4_2_6.zip" and move it under a working directory (e.g. C:\).

2) Add a path to RRI-CUI folder with the following steps (e.g. for Windows)

1. Select Computer from the Start menu
2. Choose System Properties from the context menu
3. Click Advanced system settings → Advanced tab
4. Click on Environment Variables, under User's Variables, find **PATH**, and click to edit it. If you do not have the item PATH, you may select to add a new variable and add PATH as the name.
5. In the Edit windows, modify **PATH** by adding ";C:/RRI/RRI-CUI/bin/" (for 64 bit) or ";C:/RRI/RRI-CUI/bin32/" (for 32 bit) at the end of line.

   Note: do not delete existing PATH settings. Only add the above item to the existing line. Also do not forget to add ";" to separate it from the existing path folders.
6. Click OK and close Command Prompt windows if opened.

3) If your computer has no Intel Fortran installed, run
RRI/RRI-CUI/etc/w-fcompxe/w_ifort_runtime_p_2022.2.0.9553.exe (for 64 bit) or
RRI/RRI-CUI/etc/w-fcompxe/old/w_fcompxe_redist_ia32_2013.5.198.msi (for 32 bit),
which installs necessary library files to execute RRI programs compiled by Intel Fortran.

4) Open Command Prompt by Start → All Programs → Accessories → Command Prompt
(If your computer has Intel Fortran installed, you may also operate it from Start → All Programs → Intel(R) Software Development Tools → Intel(R) Fortran Compiler ** → Fortran Build Environment for Applications running on …)

2.2 Run RRI Model

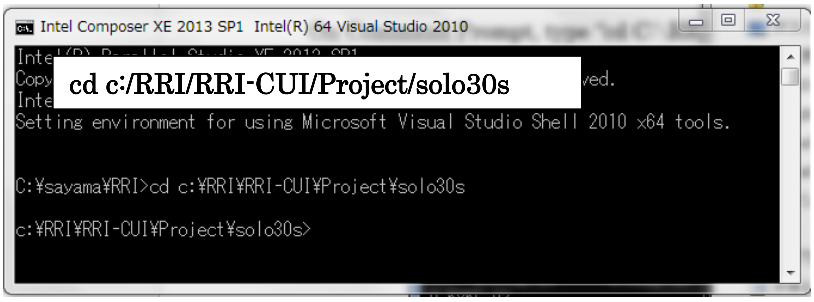
Open "RRI_Input.txt" under "RRI-CUI/Project/solo30s" with a text editor and look inside the file. This is a control file used by RRI Model. By editing the RRI_Input.txt file, you change the simulation settings.

```
RRI_Input_Format_Ver1_4_2

./rain/rain.dat
./topo/adem.txt
./topo/acc.txt
./topo/adir.txt

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

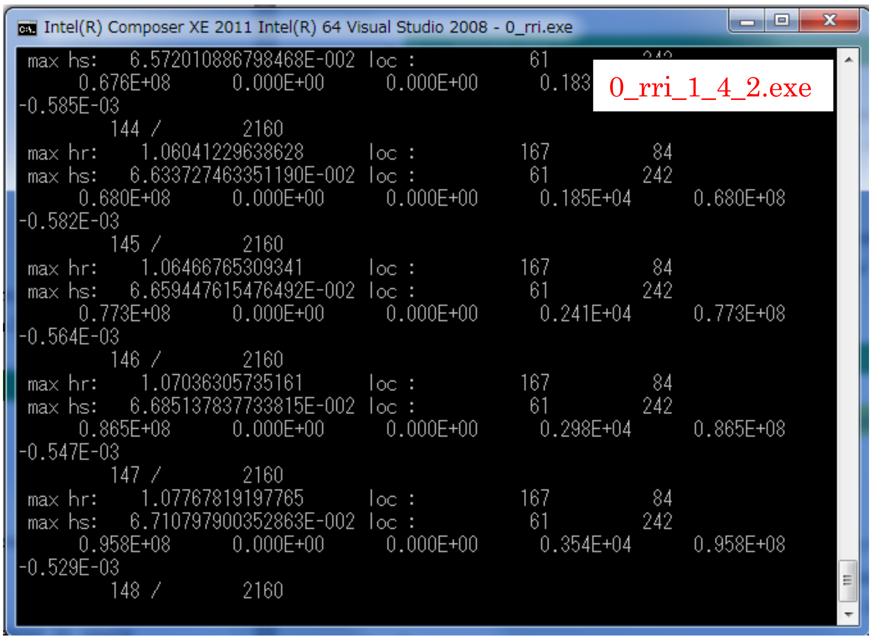
RRI_Input.txt

For example, L3 specifies the path to an input rainfall file and L4 – L6 specify the paths to input topography files (adem, acc, and adir).

Open Command Prompt and type in "cd C:/RRI/RRI-GUI/Project/solo30s/" to change the current directly.



Type in "0_rri_1_4_2.exe" and enter to execute RRI Model with RRI_Input.txt.



Confirm the output files are successfully created inside the directory of "RRI/RRI-CUI/Project/solo30s/out/". Note that "**hr_000001.out**" represents the spatial distribution of **river water depths [m]** at the output time step 1. "**hs_000001.out**" and "**qr_000001.out**" represent those of **slope water depths [m]** and **river discharge [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-CUI/etc/gp466-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-CUI/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.

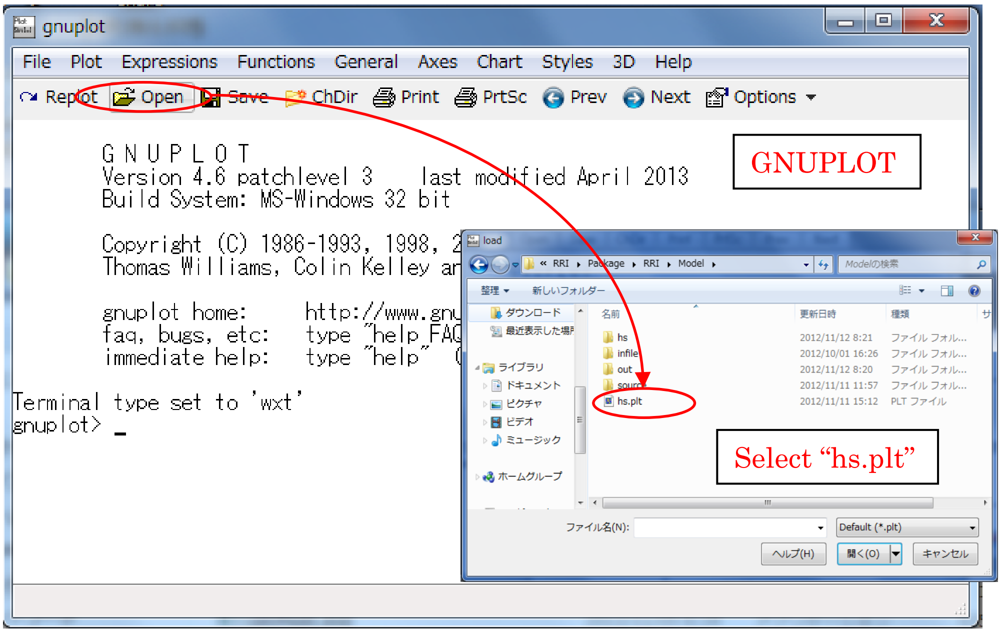
hs_plt.txt

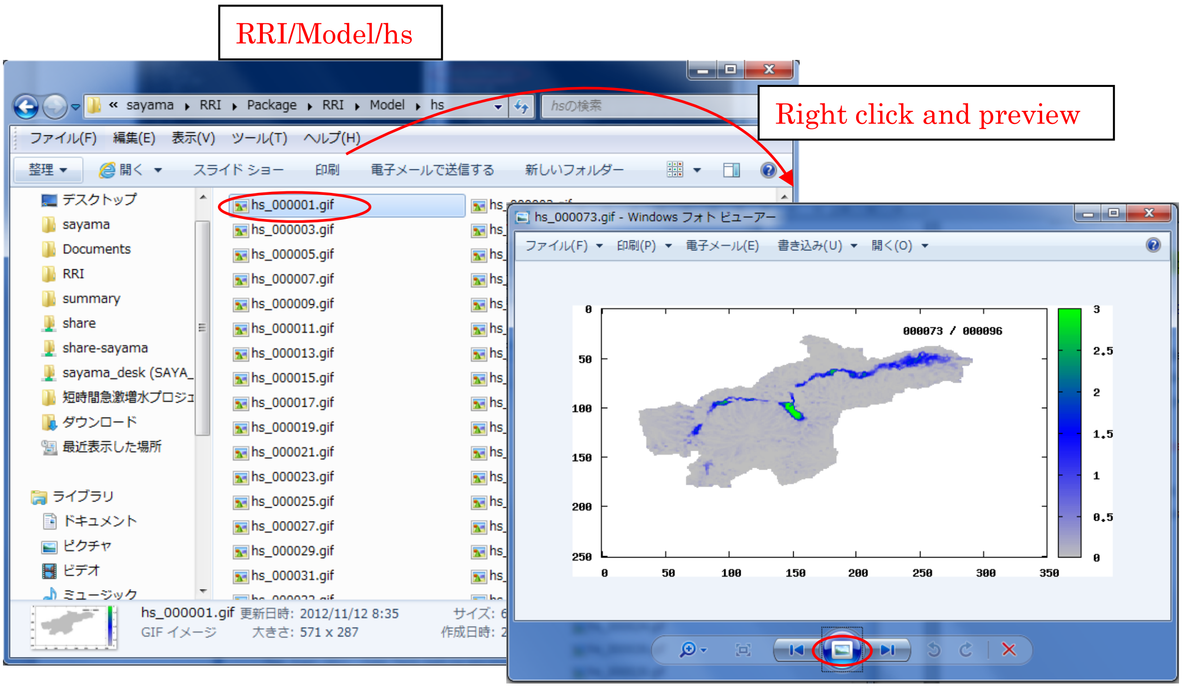
```
reset

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-CUI/Project/solo30s/hs.plt”, which is a script file to create gif files from the RRI output (see above figure).

Look at “RRI-CUI/Project/solo30s/hs” directory, where gif files are newly created. Check the created gif files by preview.





2.3.2 Compute hydrograph

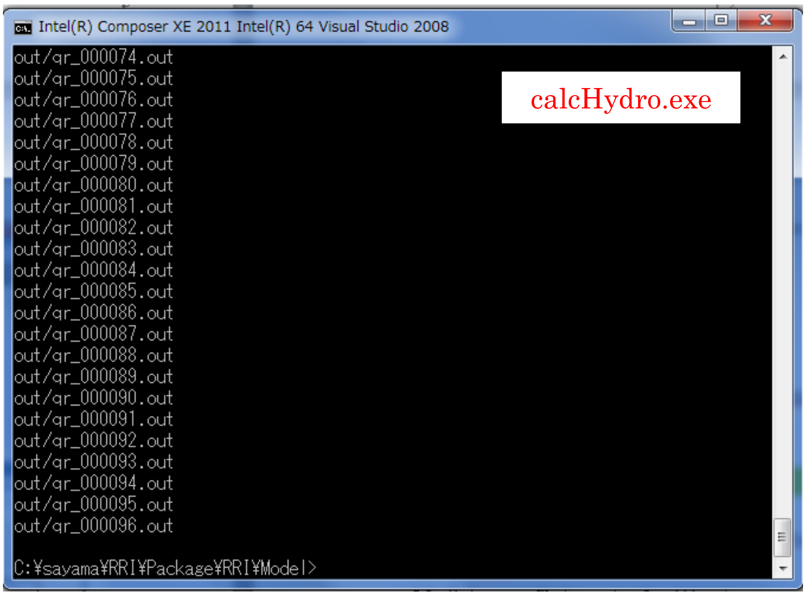
Look at "RRI/RRI-CUI/Project/solo30s/calcHydro.txt" (see "./etc/calcHydro/00_readme.txt" for more details)

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

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

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

On Command Prompt with current folder at "./Project/solo30s/", type in "calcHydro.exe" to compute time series data from RRI output files.



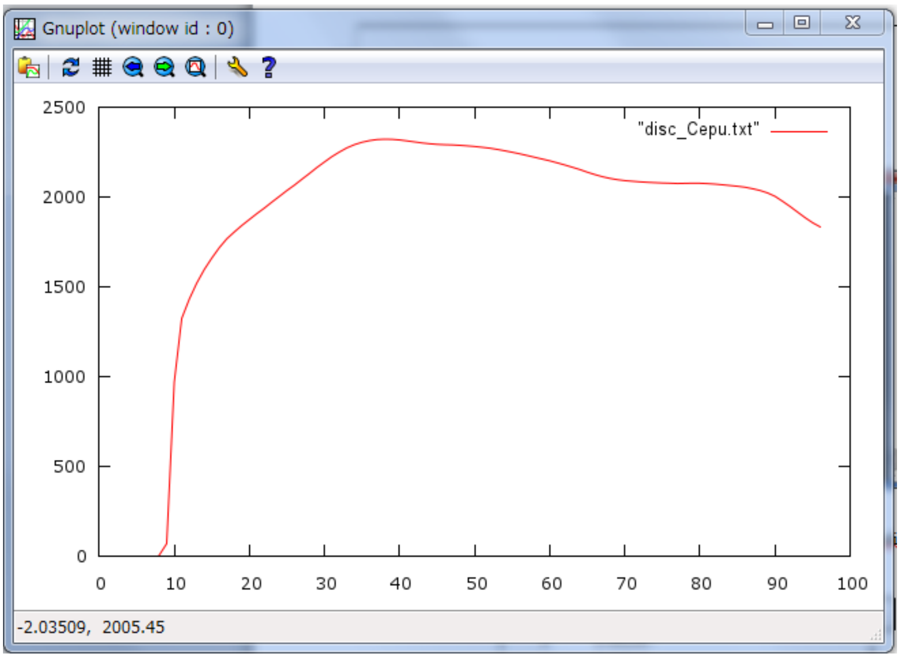
Confirm that a hydrograph file named "disc_Cepu.txt" is created.

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 "hydro.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 "./Project/solo30s/calcPeak.txt" (see "./etc/calcPeak/00_readme.txt" for more details) and edit the file if necessary.

L1 : [in] dem file (e.g. ./topo/adem.txt)

L2 : [in] output file without numbers or extension (e.g. ./out/hs_)

L3 : [in] the number of output files (e.g. 96)

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

On Command Prompt with current folder at "RRI/Project/solo30s/", type in "calcPeak.exe" to compute peak inundation depth.

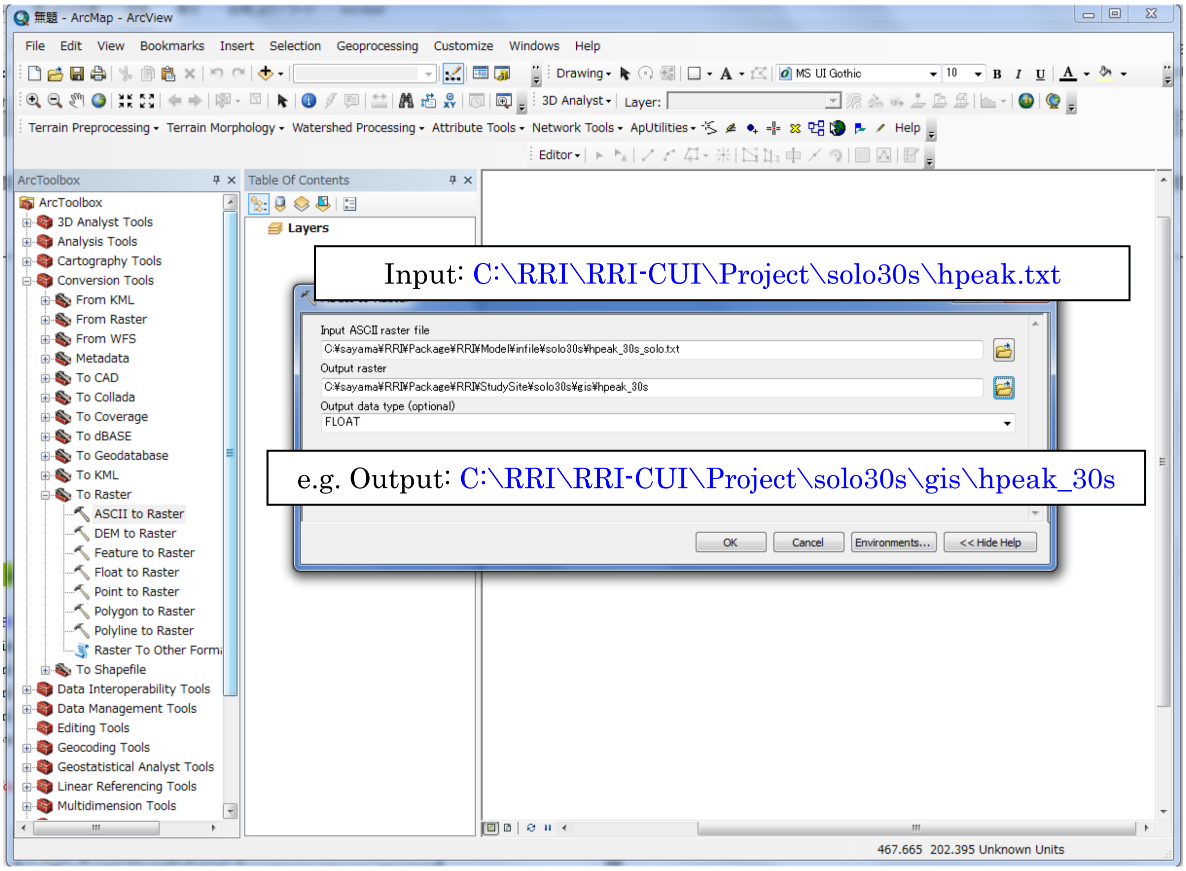
```
ncols           336
nrows            204
xllcorner       110.2
yllcorner       -8.3
cellsize        0.00833333333333
NODATA_value    -9999
 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999 -9999
-9999 -9999 -9999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

hpeak.txt

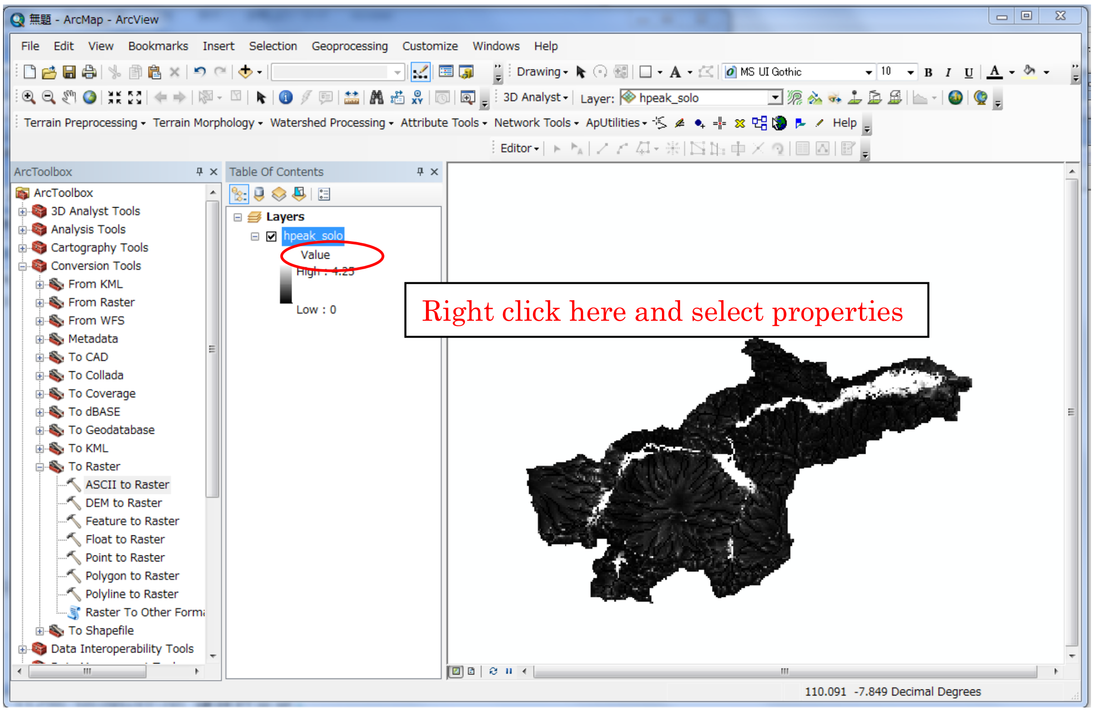
Visualize created “hpeak.txt” on ArcGIS by converting it from ASCII to Raster.

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)

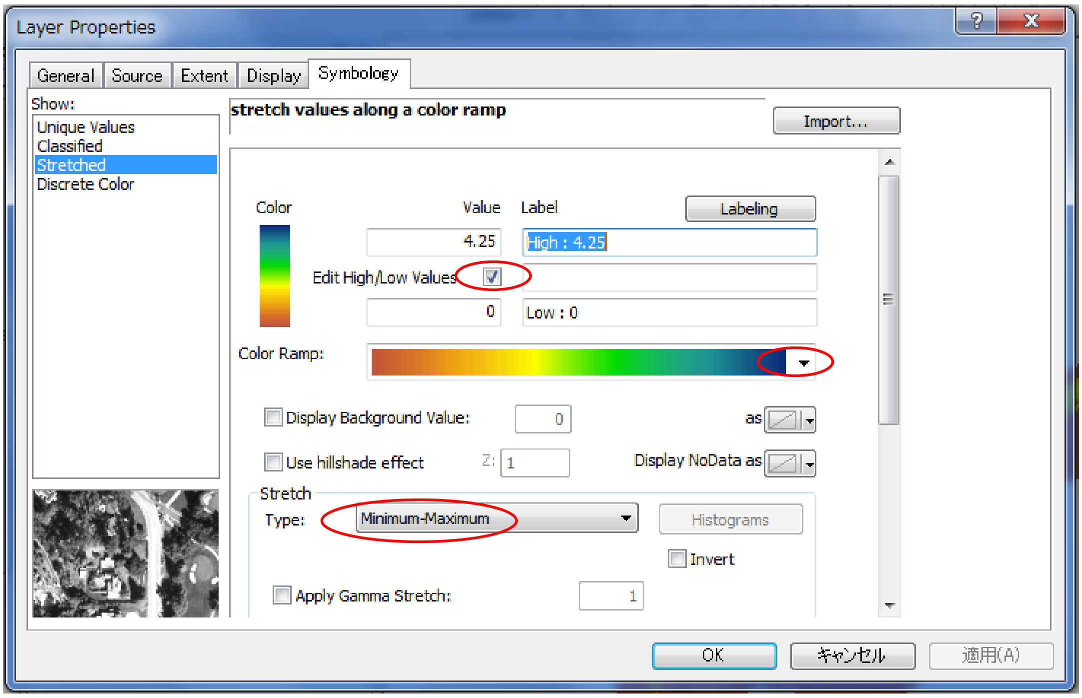
2) From ArcToolbox → [Conversion Tools] → [To Raster] → [ASCII to Raster]



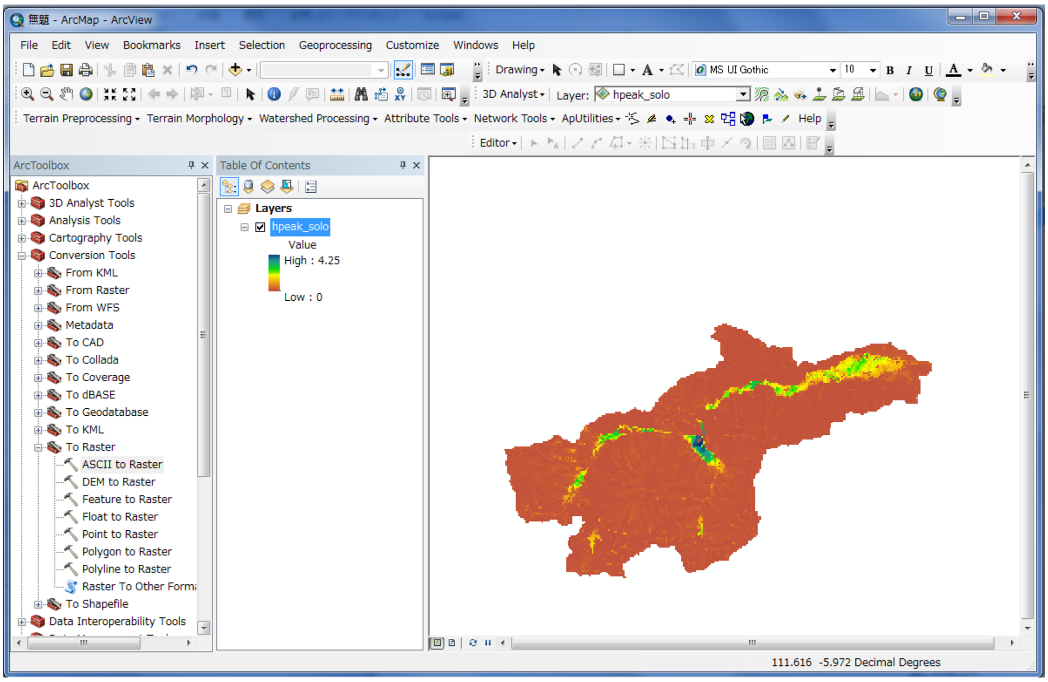
3) For the input data, select “hpeak.txt”. For the output raster, a user may use “RRI/StudySite/solo30s/gis/hpeak_30s”.



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.

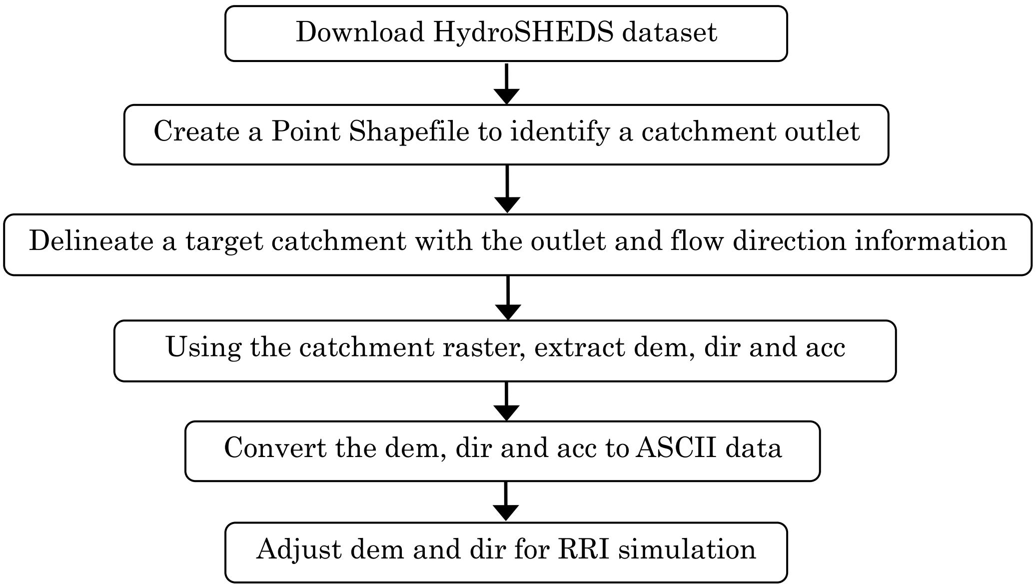
Create a New Project Folder

When you prepare a new input topographic data, create a new project folder:

Copy "newProject" folder including all the files and folders inside and save it with a new project name under "RRI-CUI/Project/".

Note: In the package, "RRI-CUI /Project/solo15s" is prepared in advance for the tutorial.

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

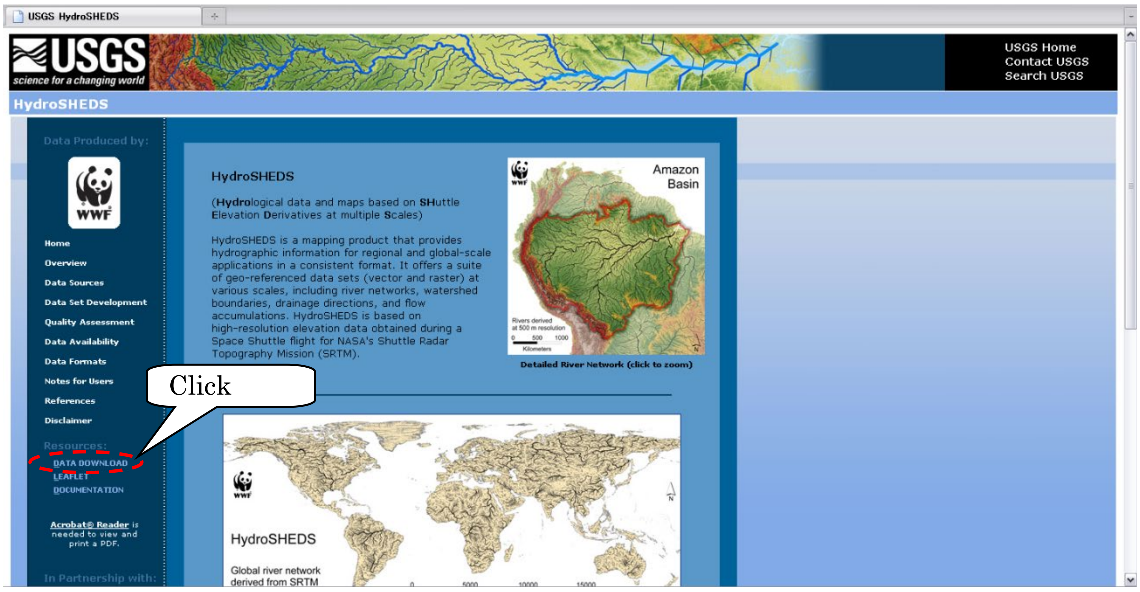
3 arc-second, 15 arc-second, and 30 arc-second are available.

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, choose “Void-filled DEM”. For 15 sec and 30 sec, only “Conditioned DEM” is available, but in fact they are the same as previously named as “Void-filled DEM” (i.e. DEM along rivers are **not** deepened).

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

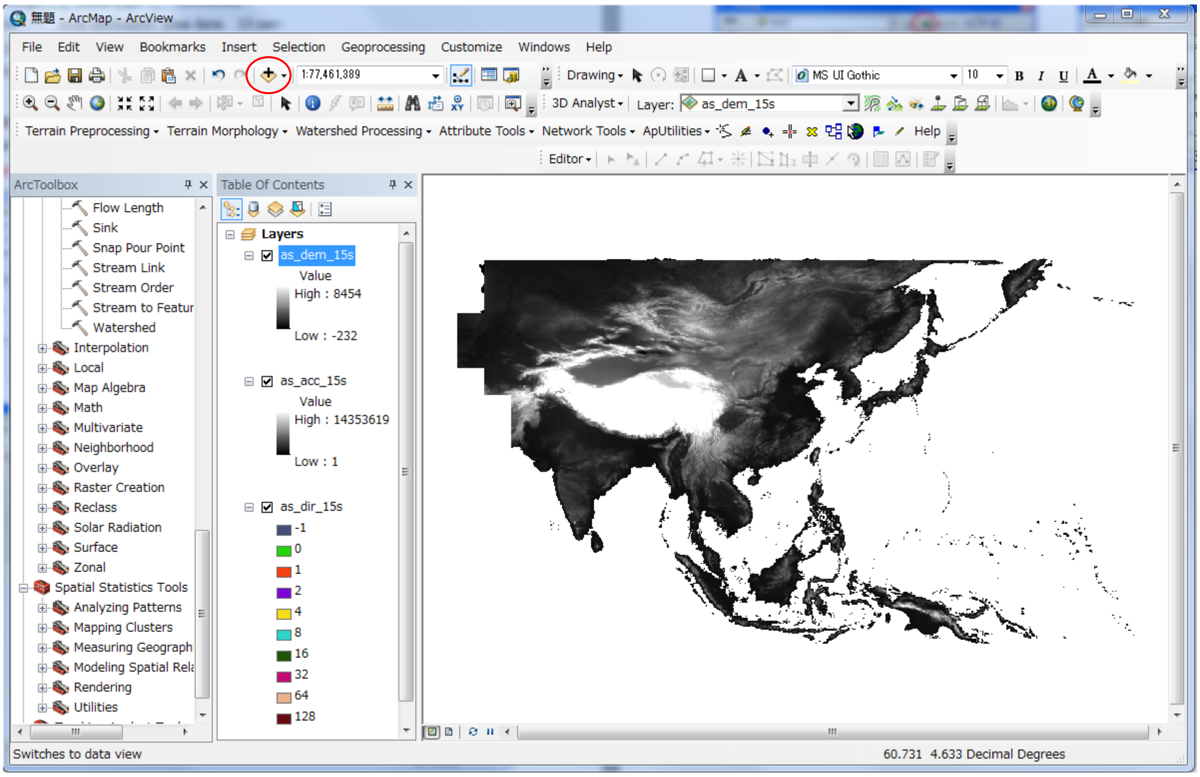
as_dem_15s → Asia digital elevation data 15 sec

as_dir_15s → Asia flow direction data 15 sec

3.2 Delineating HydroSHEDS Data using ArcGIS

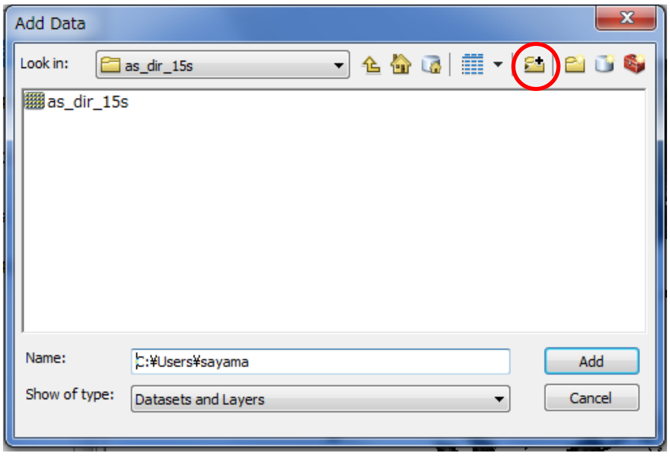
(If ArcGIS is inaccessible, skip this section and go to 3.3 to use free GLASS GIS)

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.

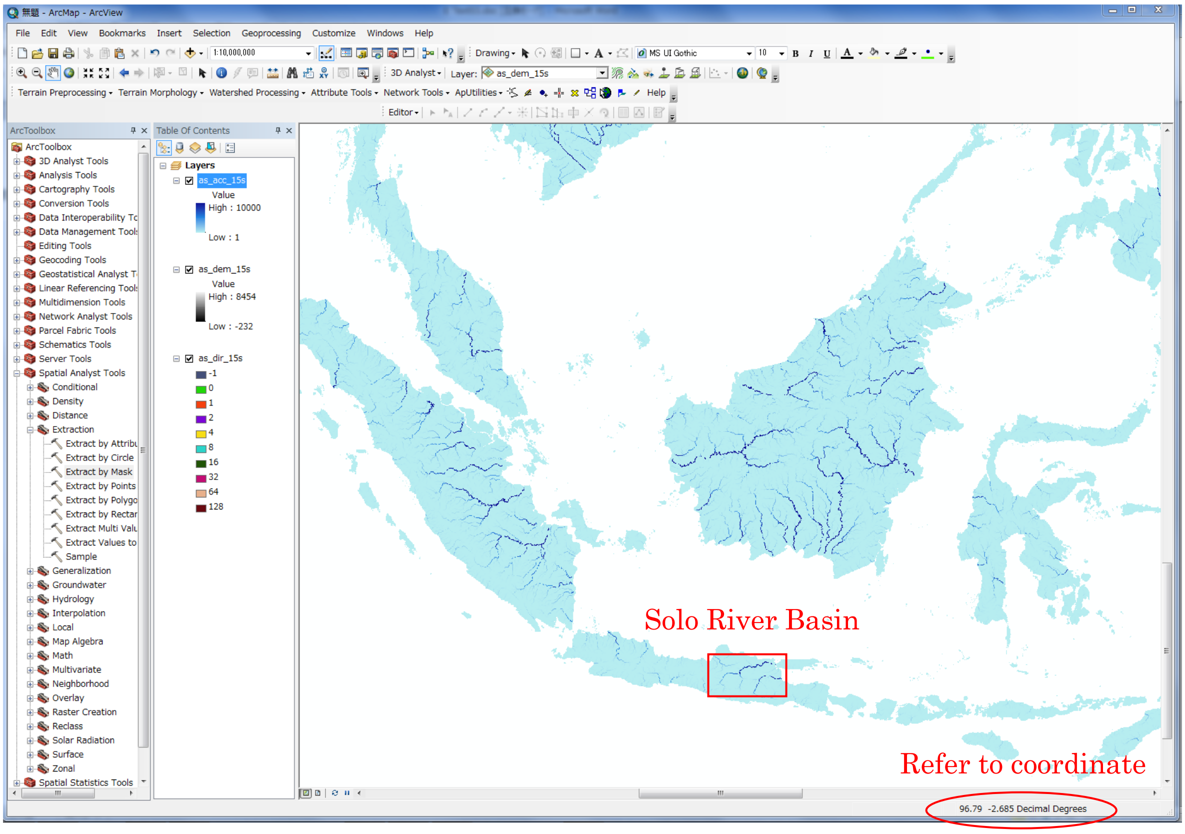


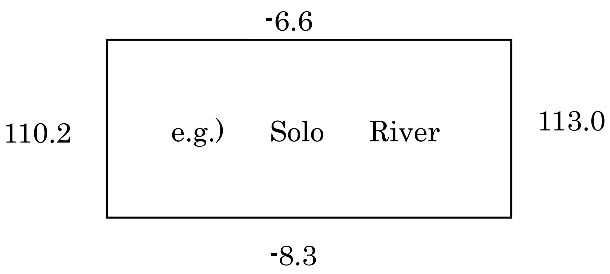
※Selecting the folder to connect

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



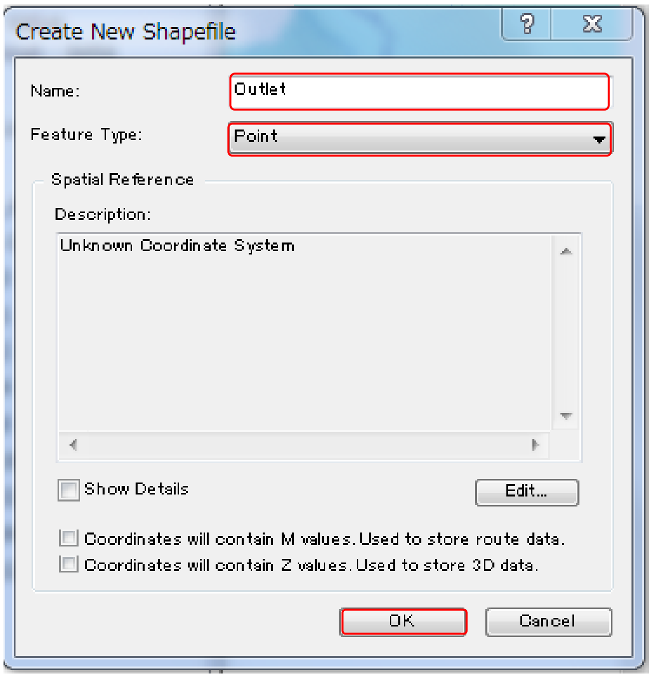
③ Display the flow accumulation data (i.e. as_acc_15s) on top screen (change the color range to show river network clearly). Then find your target river and decide the rectangular range, which covers all upstream contributing area. (At this stage, the following rectangle range should be just written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



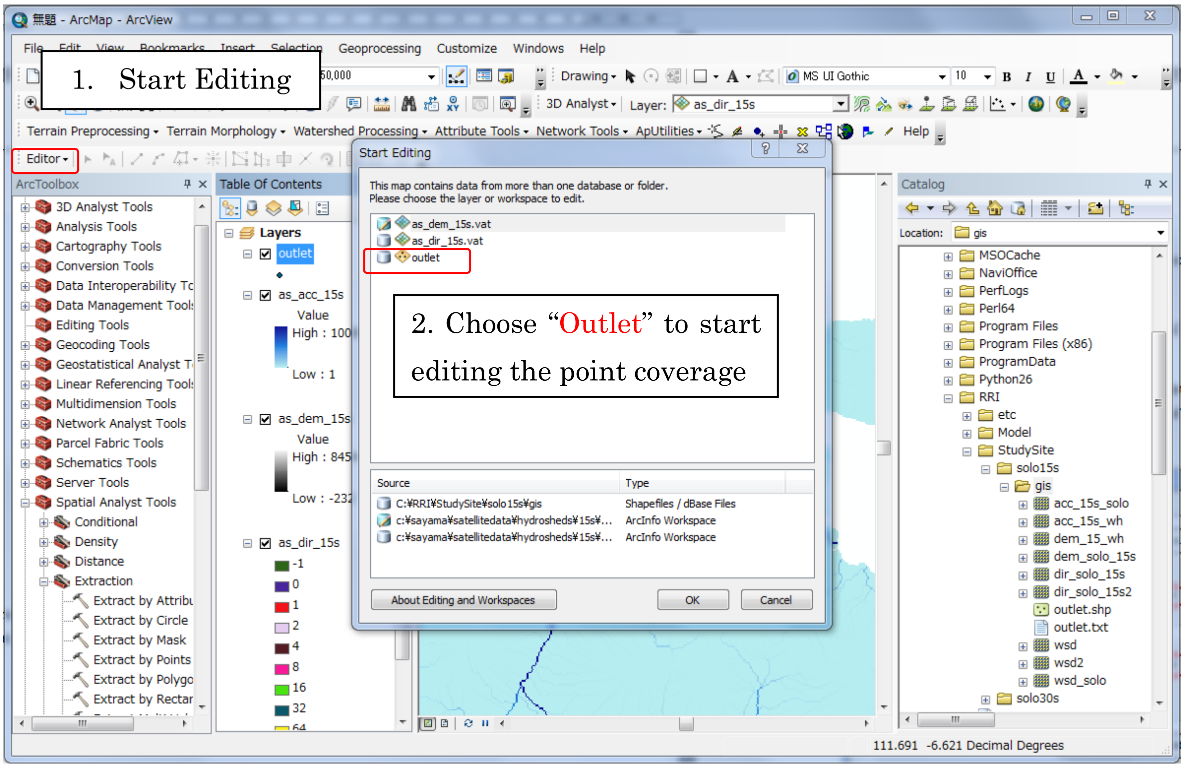


(The range should be written down on your notebook.)

④ Show arc catalog (from the main menu, [Windows] → [Catalog]).
On the arc catalog, “Folder Connections” to a working folder (e.g. RRI/Project/solo15s/gis/) and right click to choose New → Shapefile to create a point Shapefile (e.g. Outlet).

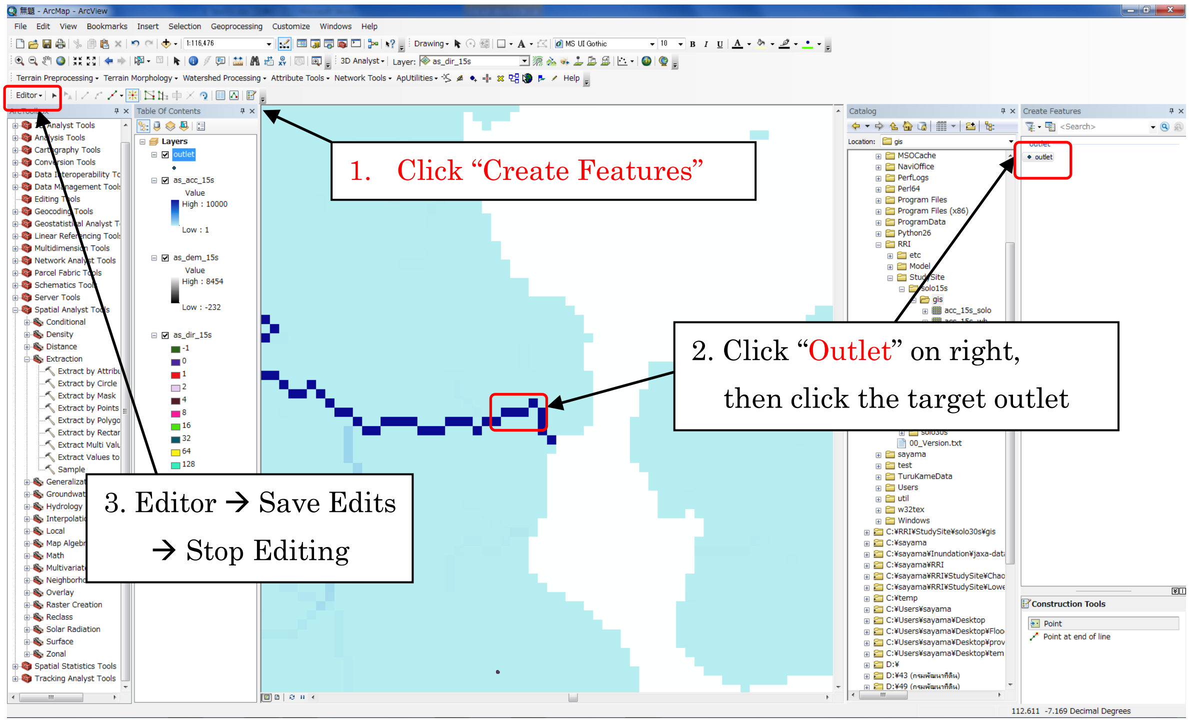


⑤ From the main menu [Customize] → [Toolbars] → [Editor]
On the Editor, choose [Start Editing], then Choose “Outlet” (the new Shapefile) to start editting.



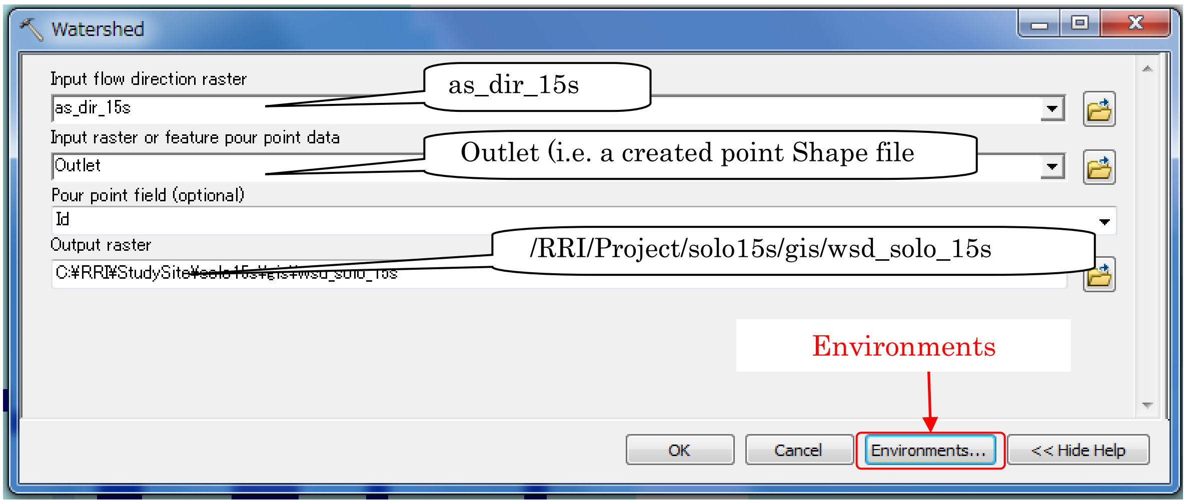
Clicking "Outlet", so that you can bring a point to indicate the target outlet.

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

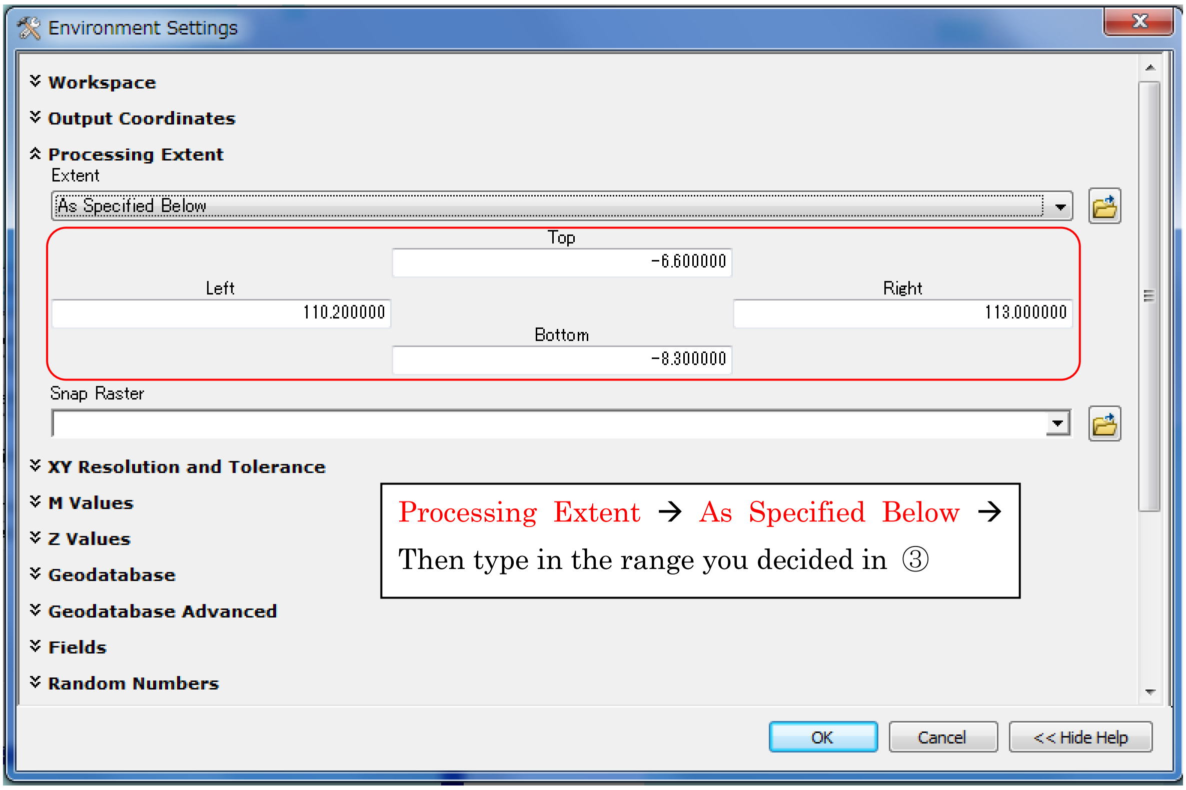


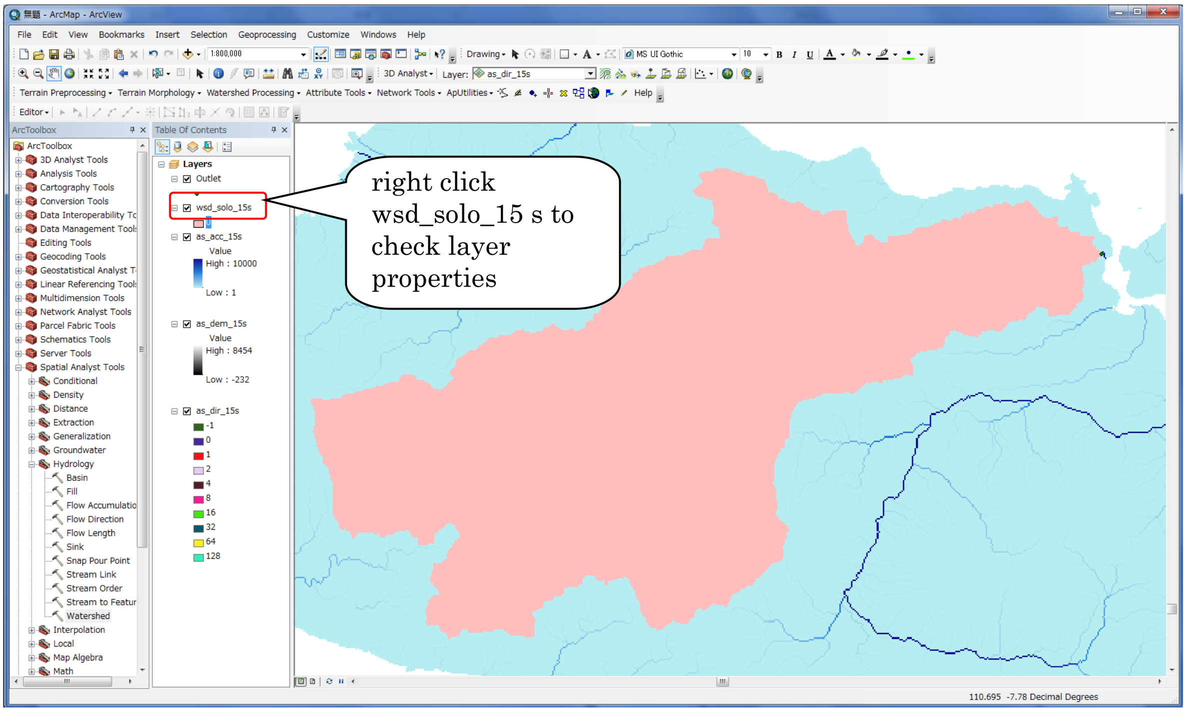
⑥ Using [ArcToolbox] → [Spatial Analyst Tools] → [Hydrology] → [Watershed], delineate a watershed with the defined outlet.

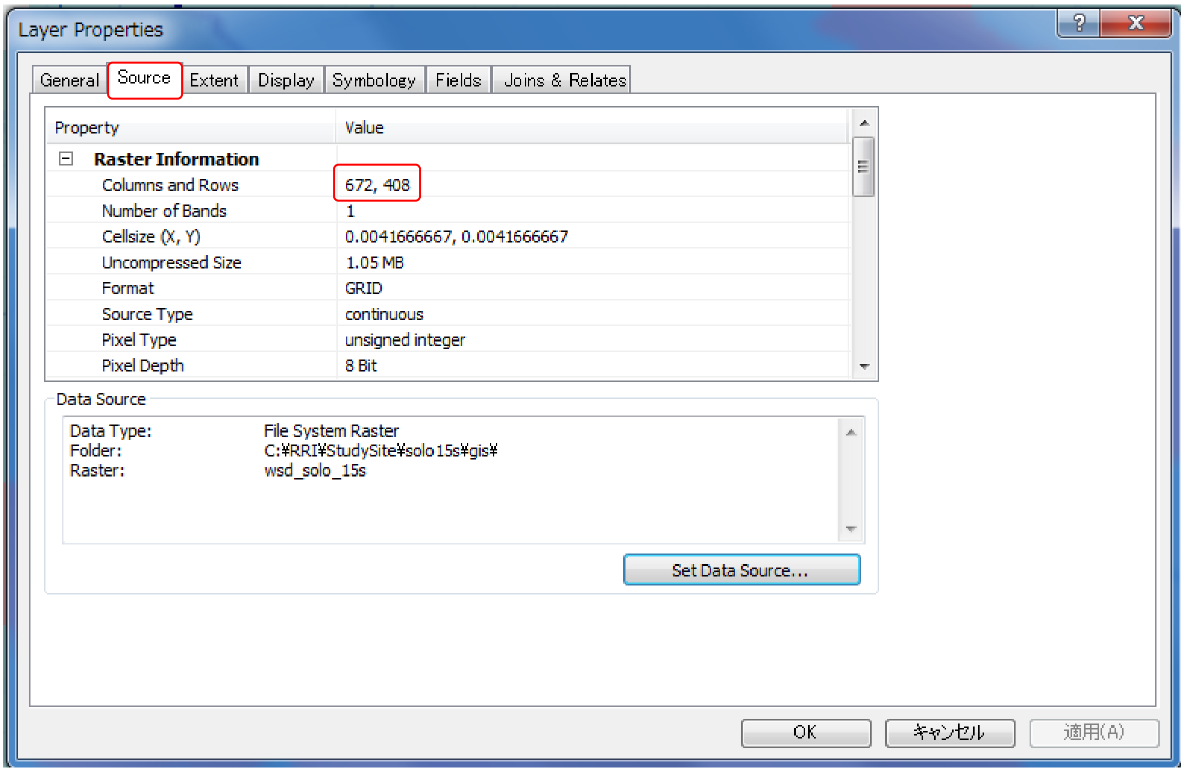
(IMPORTANT) To use [Spatial Analyst Tools] on ArcGIS, you must have the extension and activate it by choosing [Cusomize] → [Extentions] → add a check for [Spatial Analyst].



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

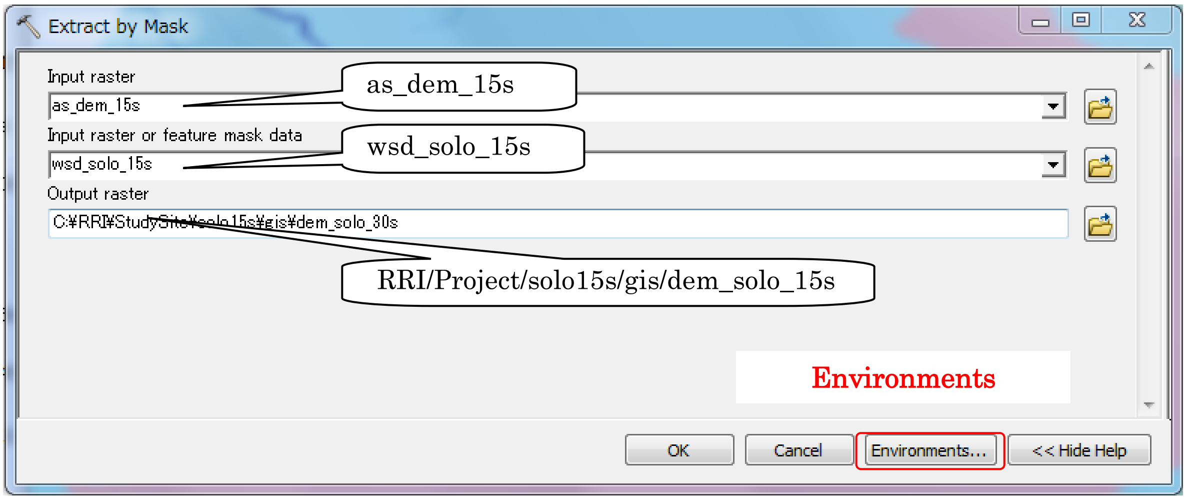


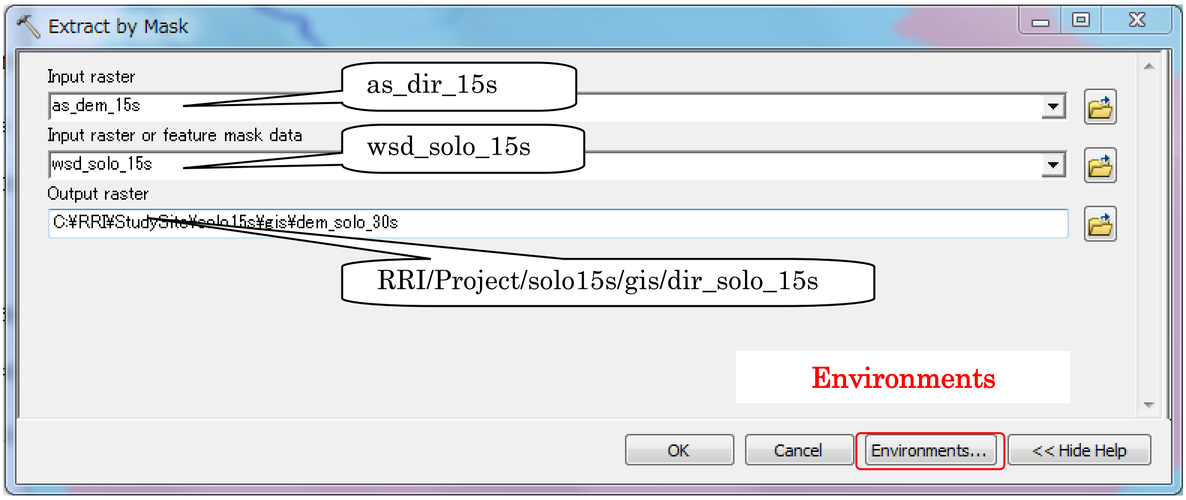


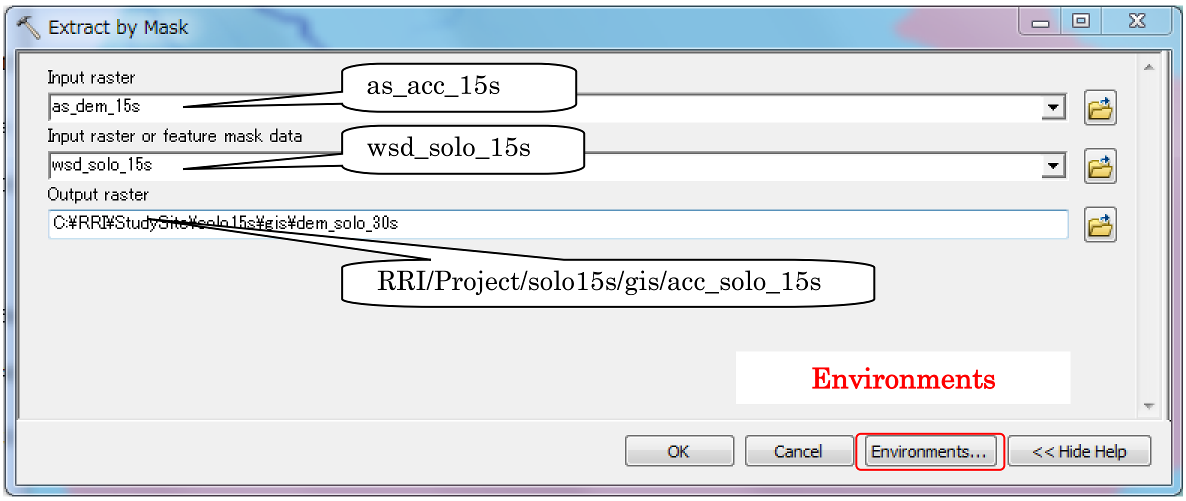


⑦ Right click the created watershed raster (e.g. wsd_solo_15s) and check layer properties. Under the "Source" tab, you can check "Columns and Rows". This will be the number of columns and rows for the topographic data used by RRI Model. If it exceeds more than 1000, using coarser resolution data is recommended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

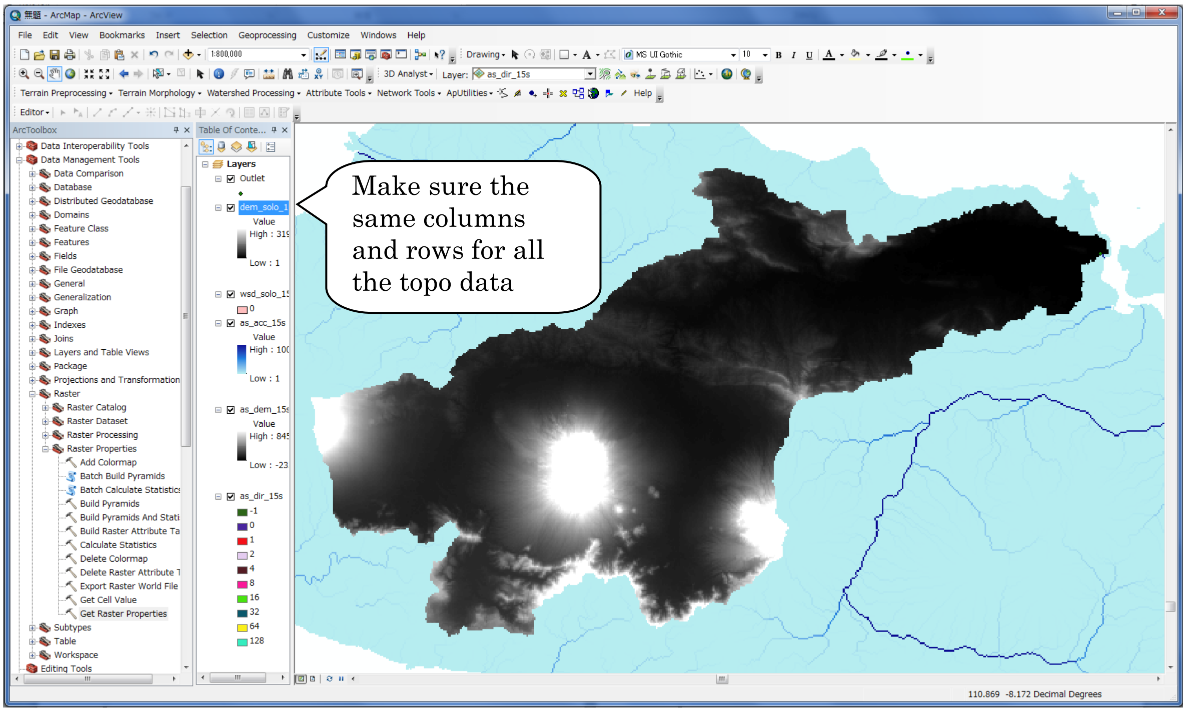
⑧ [Spatial Analyst Tools] → [Extraction] → [Extract by Mask], prepare dem (elevation), acc (flow accumulation) and dir (flow direction) masked by the delineated watershed.







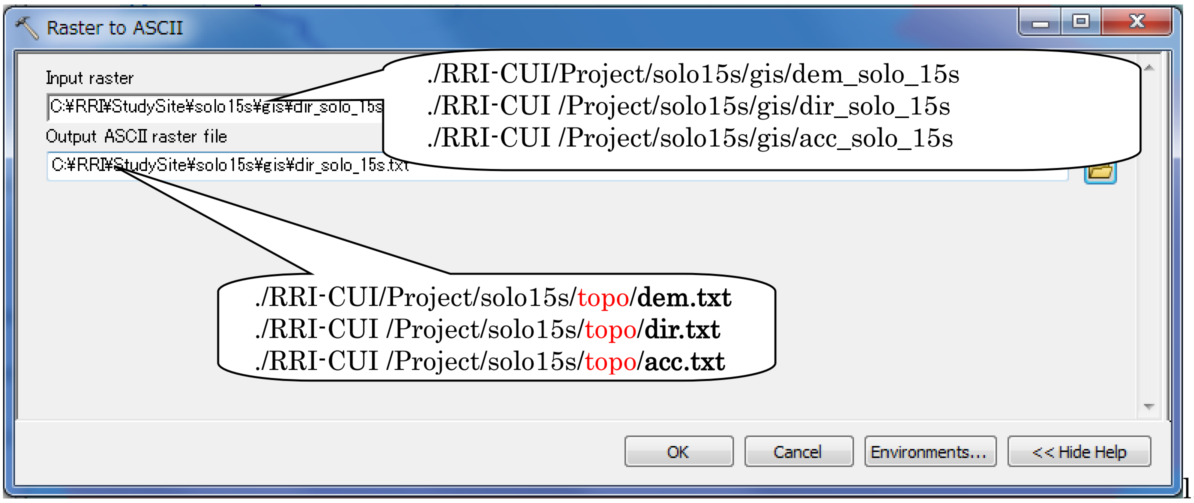
(IMPORTANT) Analysis range must be specified from the “environment” the same as above.



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.

The output files should be named as "**dem.txt**", "**dir.txt**" and "**acc.txt**" to be saved under "topo" folder in your project folder (e.g. "./RRI-CUI/Project/solo15s/topo/").



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                                          dir.txt
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
```

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)

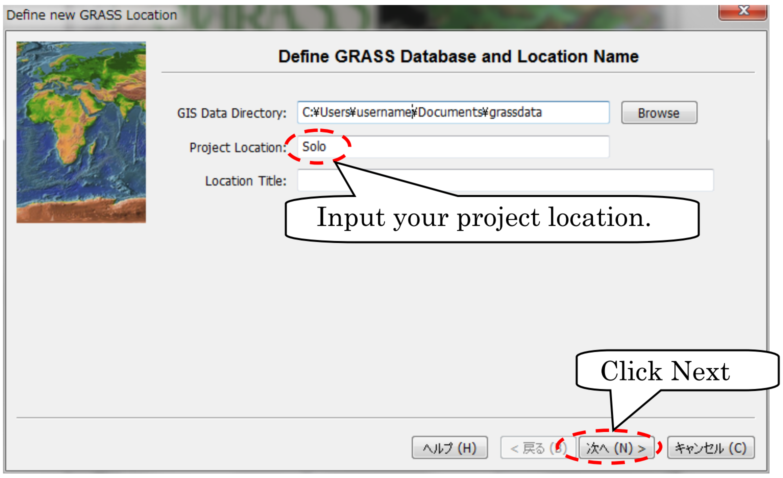
3.3 Delineating HydroSHEDS Data using GLASS GIS (optional)

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

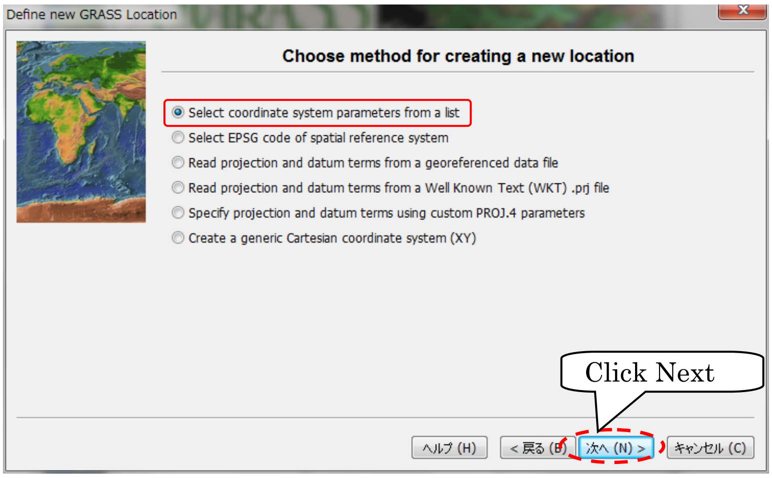
① Install the latest GRASS GIS (Latest GRASS in December 2013 is ver 6.4.3.) (GRASS website: http://grass.osgeo.org/).

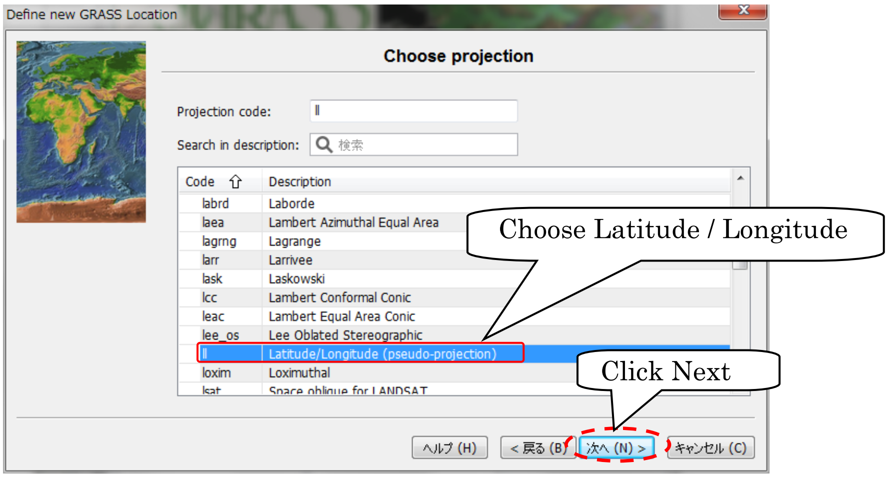
② Start GRASS GIS GUI, and click "Location wizard".

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

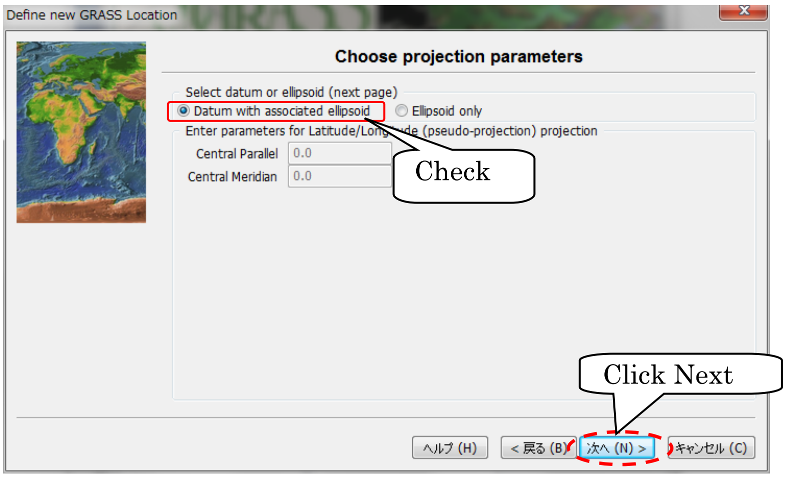


④ Select "Select coordinate system parameters from a list" and "Latitude/longitude (Pseudo-projection)" as a projection.

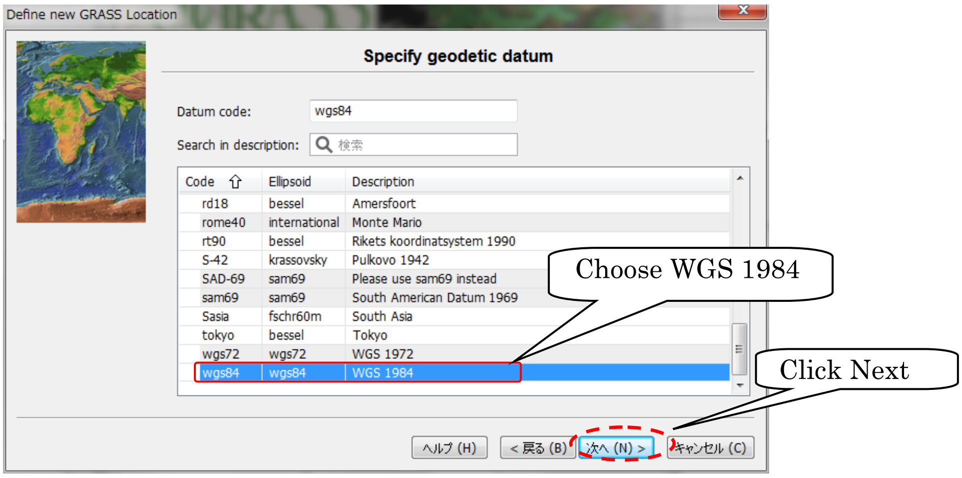




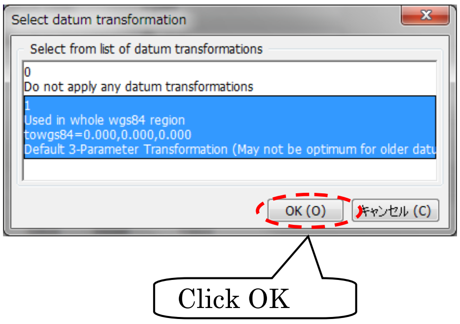
⑤ Check “Datum with associated ellipsoid” and click “NEXT“.



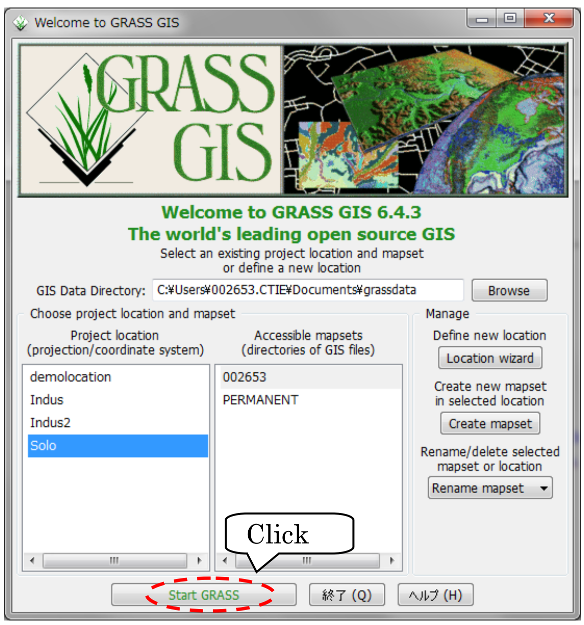
⑥ Select “WGS 1984” and as a geodetic datum and click “NEXT”.



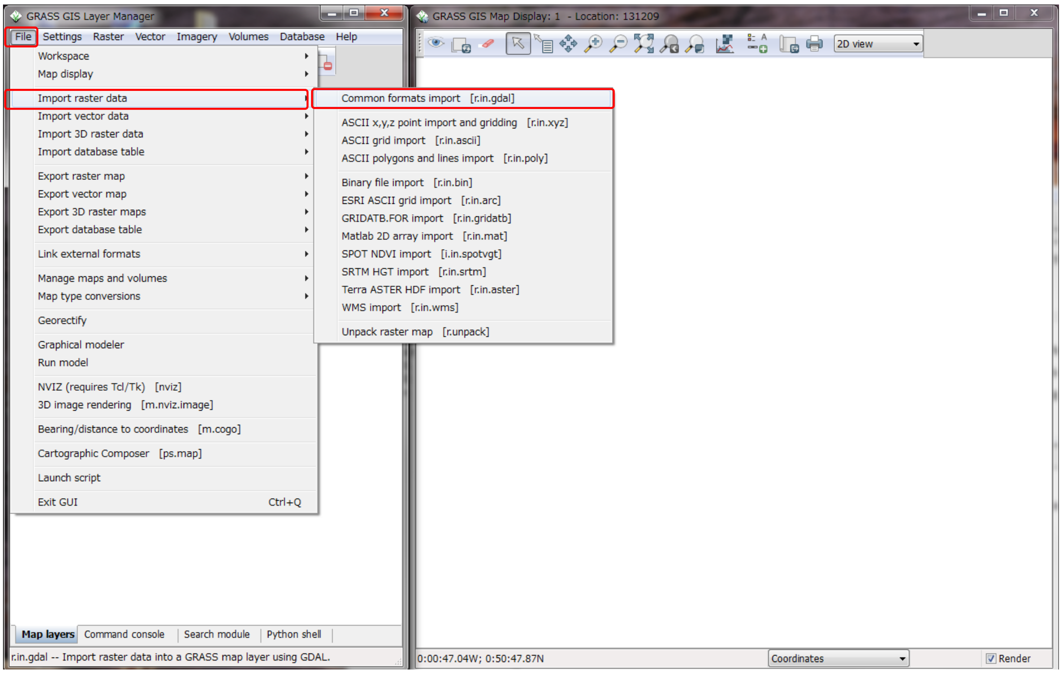
⑦ Click “OK” on “Select datum transformation” window and click “FINISH” on Summary window. (Select “Cancel” for default resolution setting).



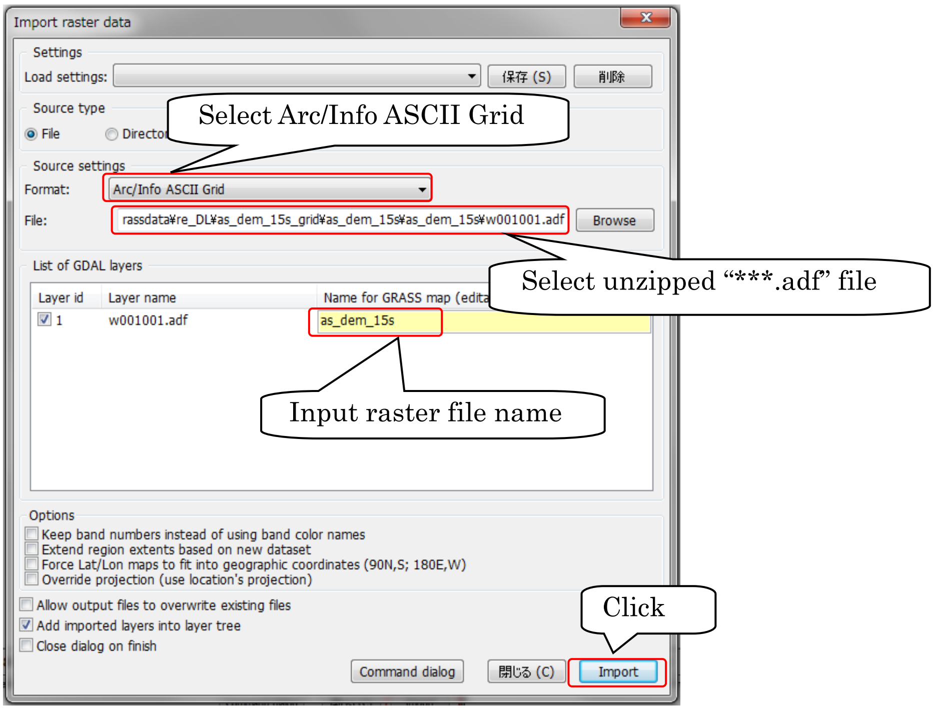
⑧ Click “Start GRASS” to start GRASS GIS.



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

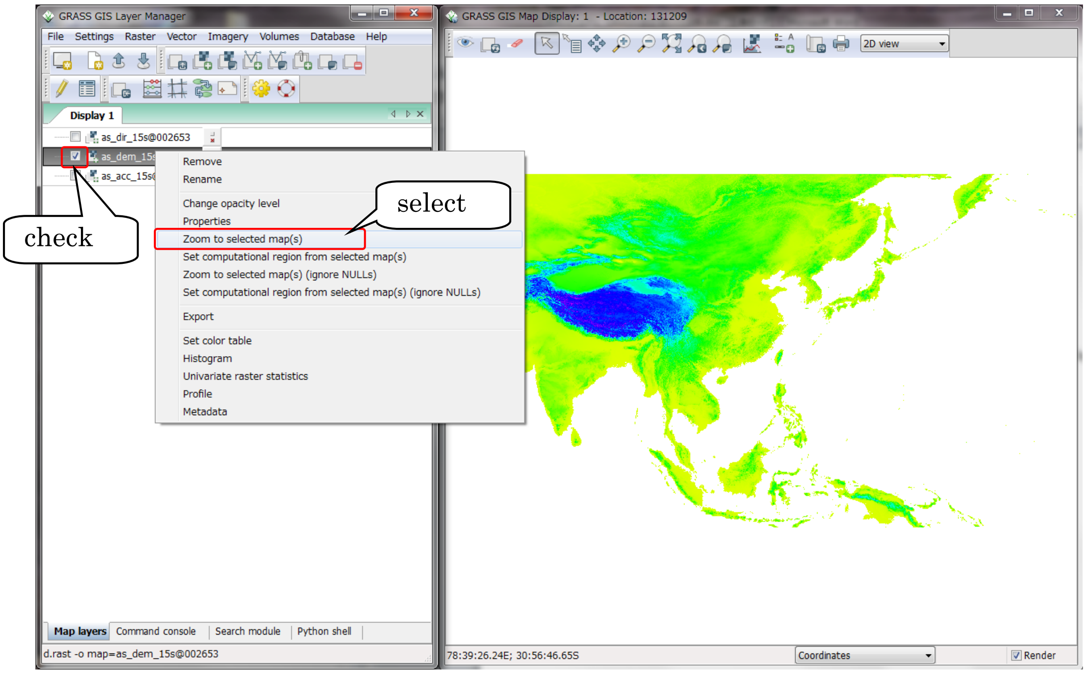


⑩ Select "Arc/Info ASCII Grid" from the "Format" list and select unzipped HydroSHEDS raster file name (e.g. w001001.adf for dem). Input "Name for GRASS map (editable)" as "as_dem_15s" for example and click Import.

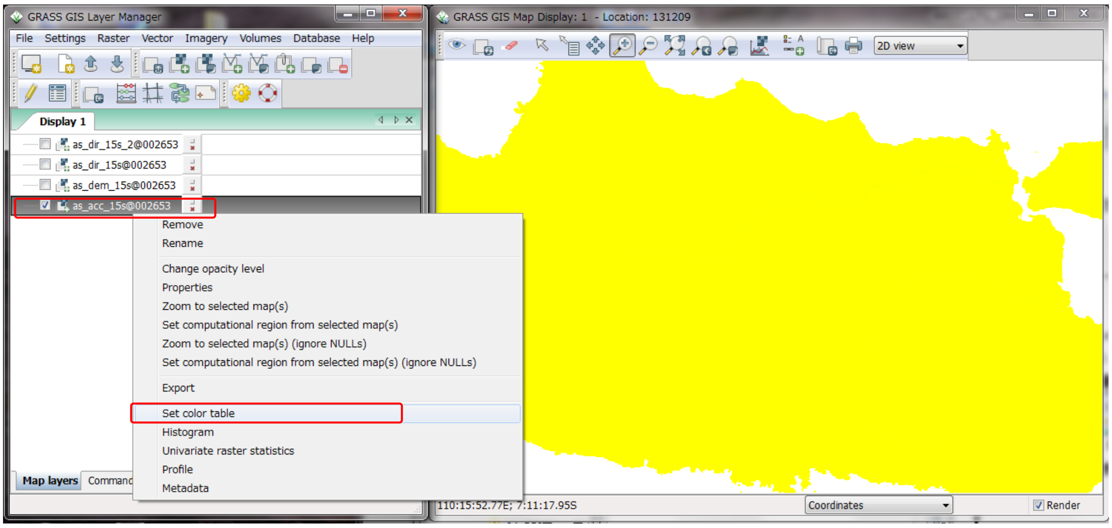


⑪ Perform the same operation for all the three types (dem, dir, acc) of topography data.

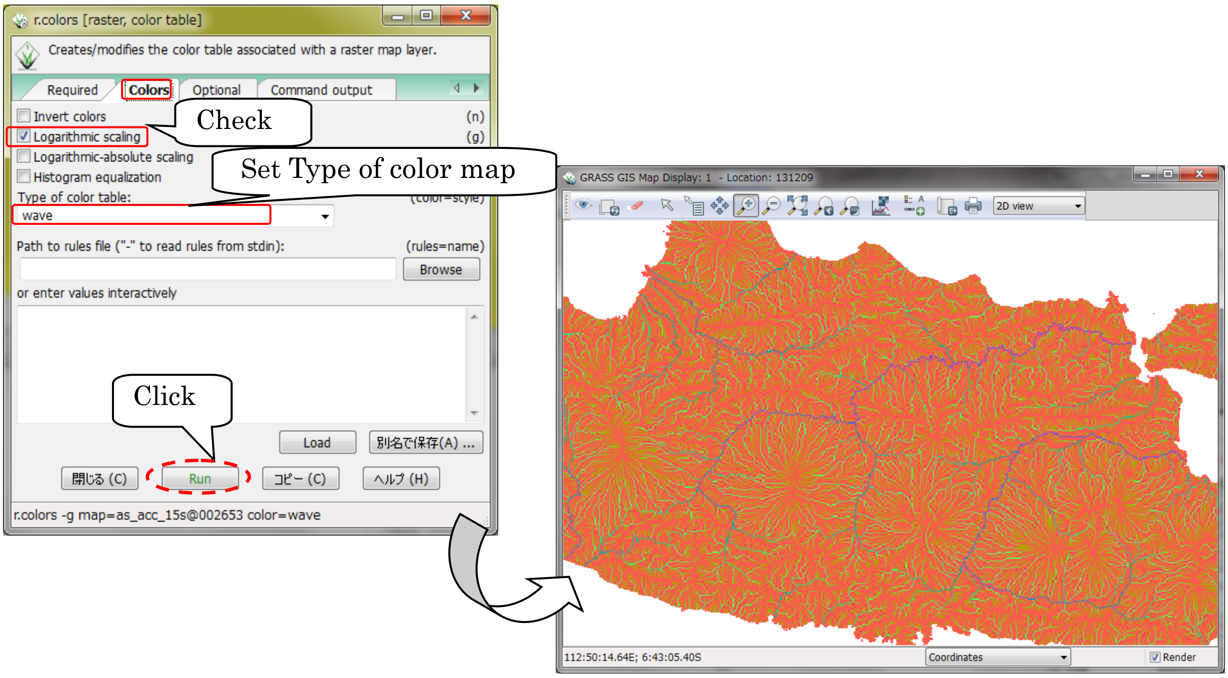
⑫ After importing three types of topography data, check the layer and right click on it and "select zoom to selected map(s)", then the raster file will be displayed in the window. (the following figure shows the example of "dem" display)



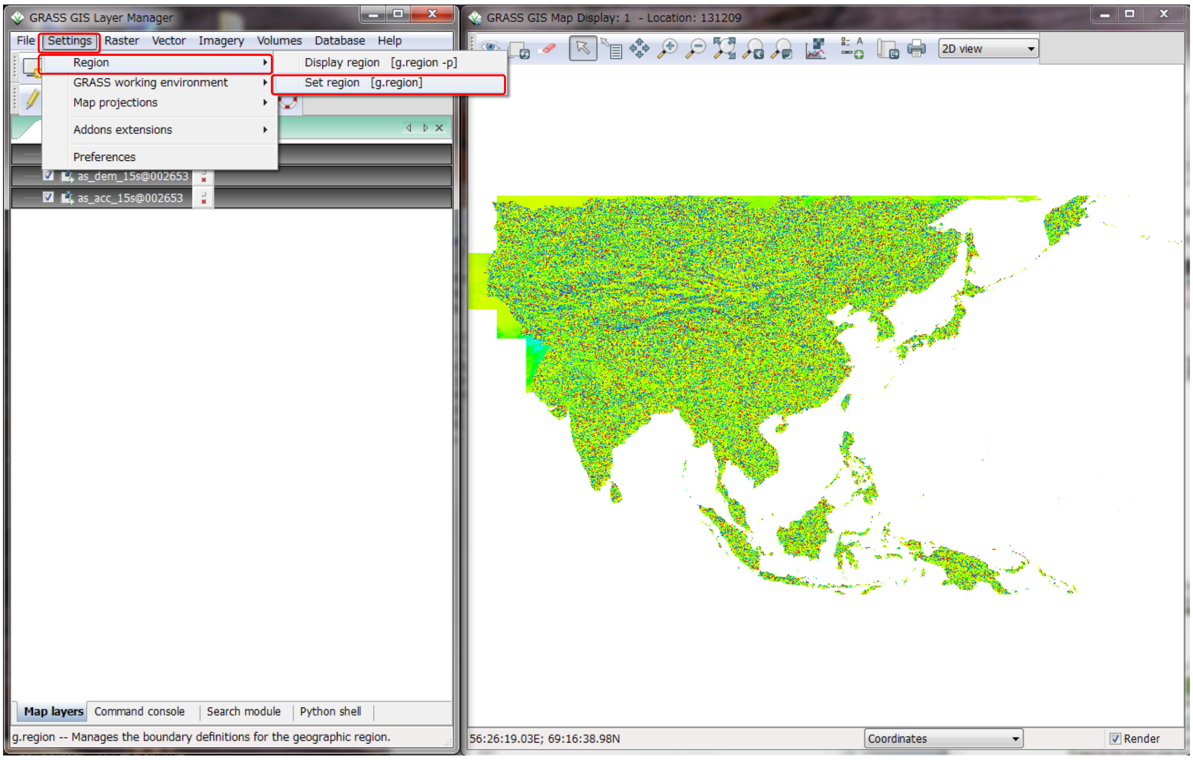
To show the flow accumulation (acc) clearly, right-click the filename of "acc" and select "Set color table".



⑬ 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".

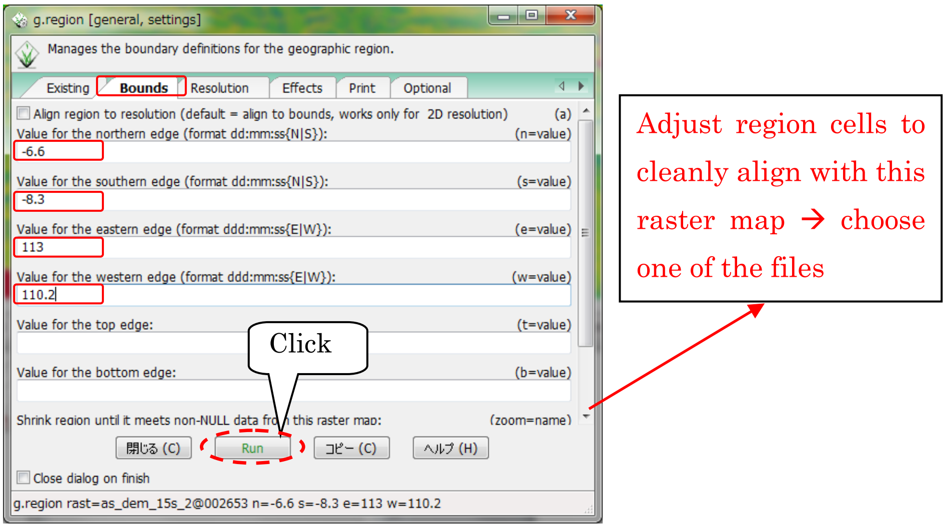


⑭ To set the delineation range, select [Settings] > [Region] > [Set Region].

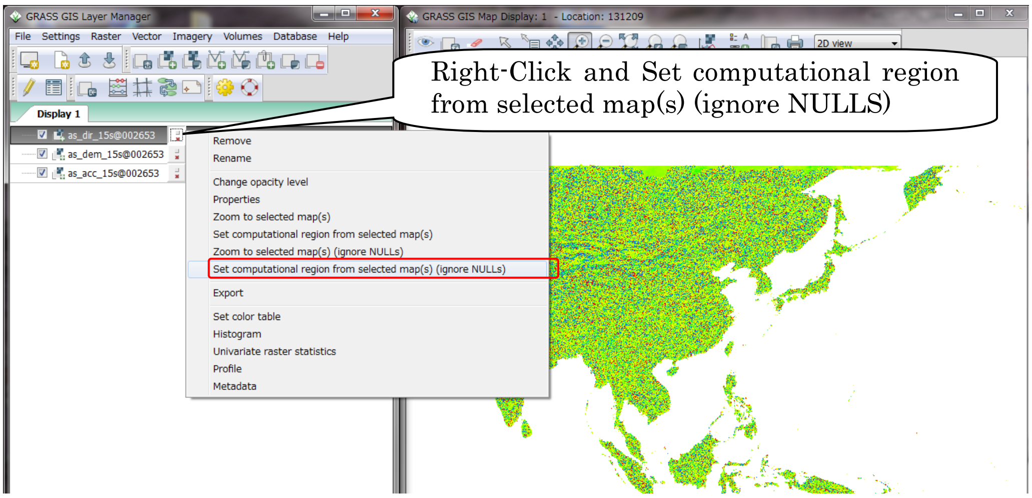


⑮ Input values for edges of the target area (coordinates) and set a file for adjusting region cells to cleanly align with a raster map, then click "Run".

(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.)

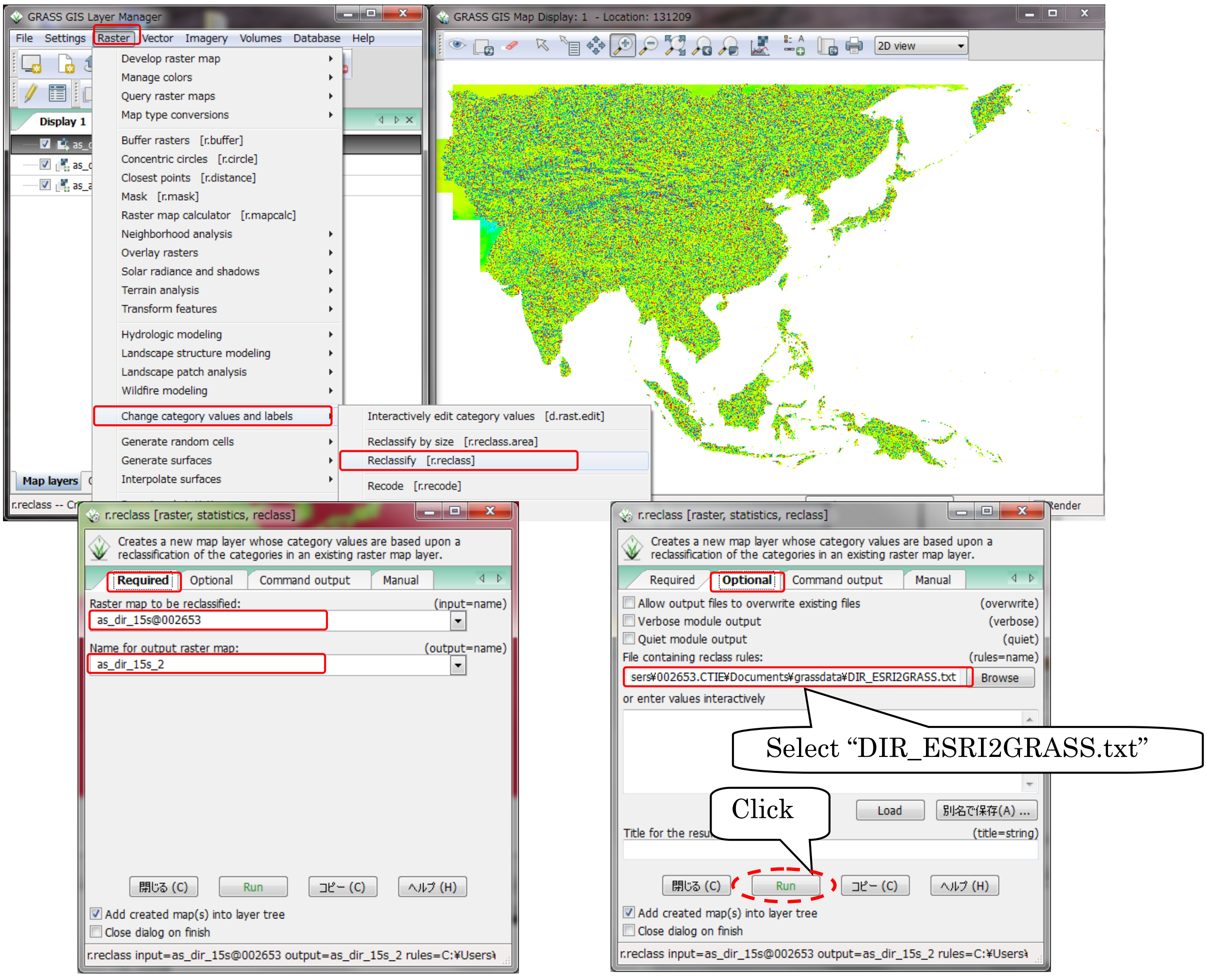


⑯ Right-click the filename of “dir” file and select “Set computational region from selected map(s) (ignore NULLS)”. Perform the same operation for all the three types (dem, acc and dir) of topography data.

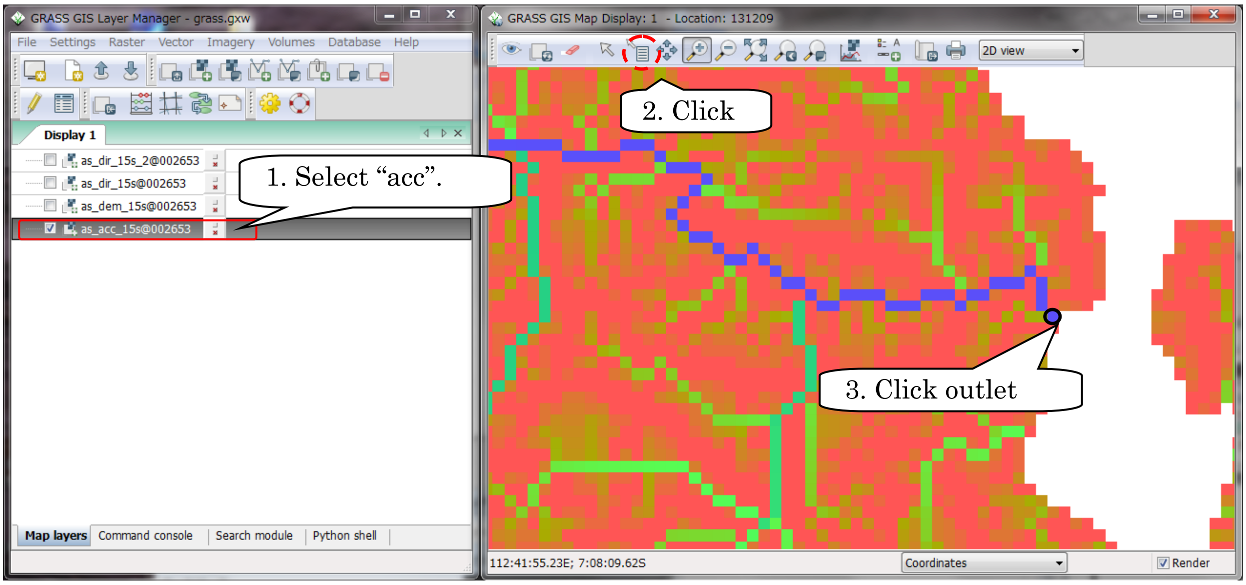


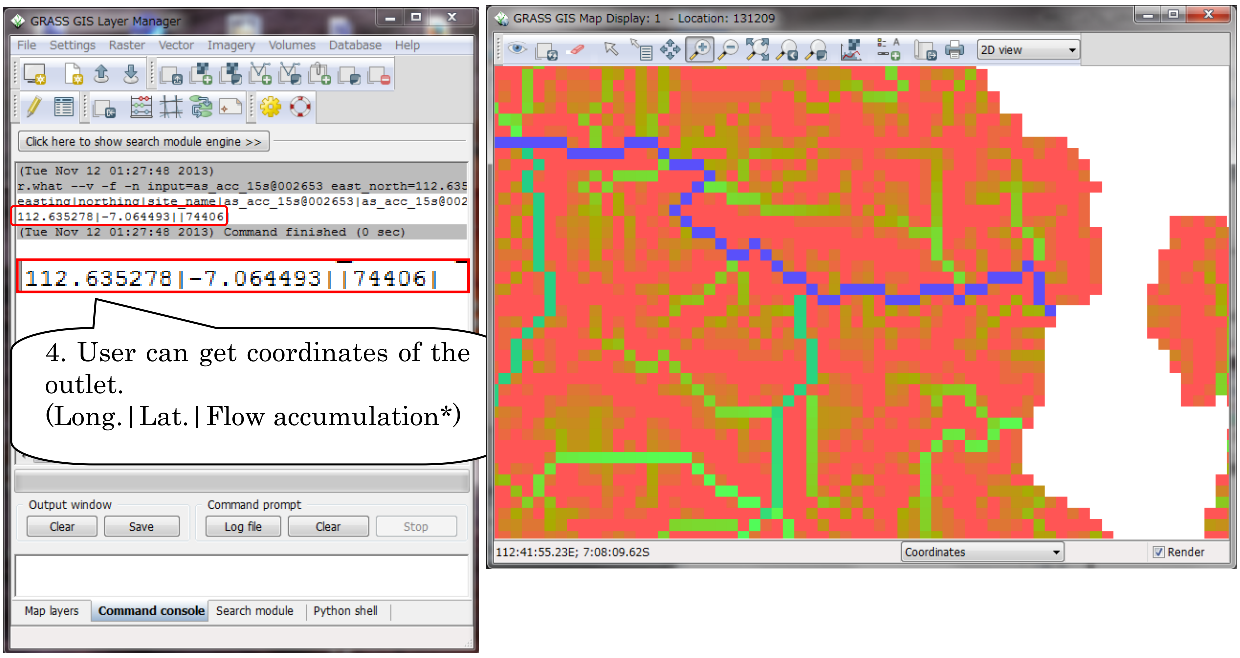
⑰ Only for flow direction, change category values (definition of river flow direction in DIR file), from ESRI type (1, 2, 4, 8, 16, 32, 64, 128) to GRASS type (1, 2, 3, 4, 5, 6, 7, 8).

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

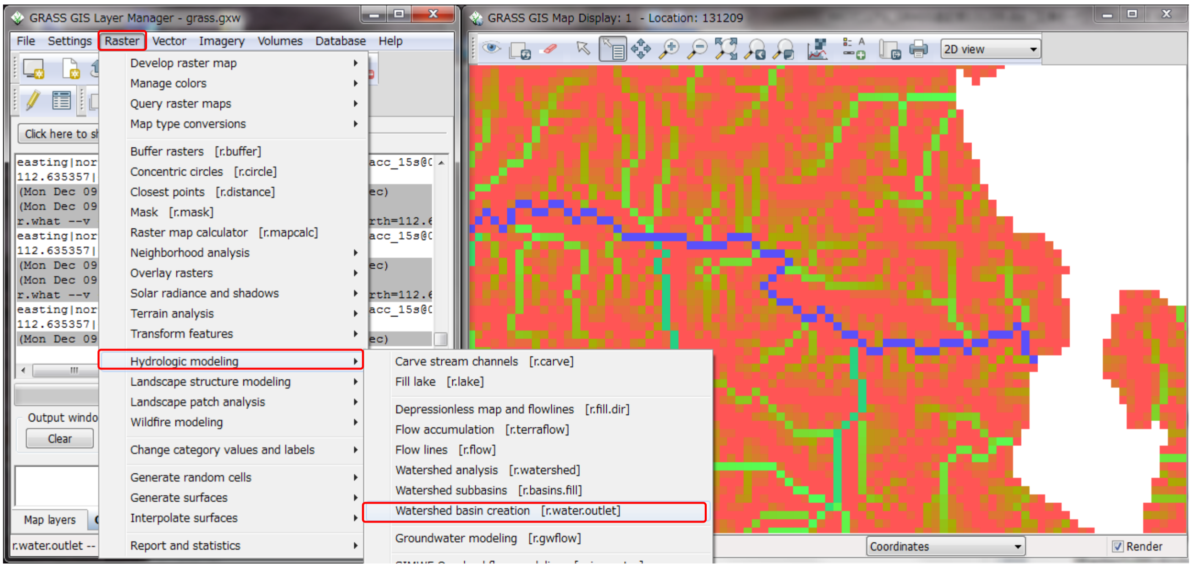


⑱ User needs to know the coordinates of the outlet (long./lat.) of target river basin to clip. Select “acc” file and perform 1, 2, 3 and 4 as shown in following figures.



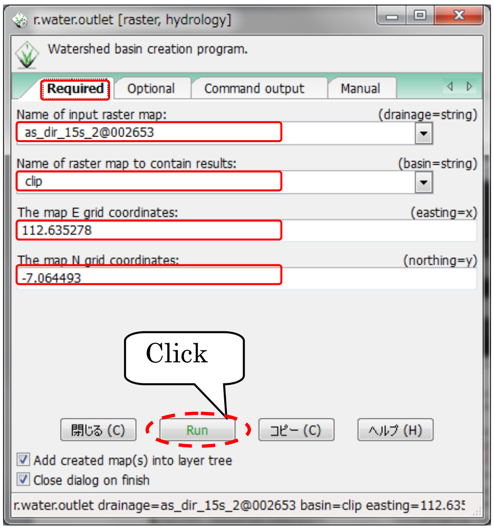


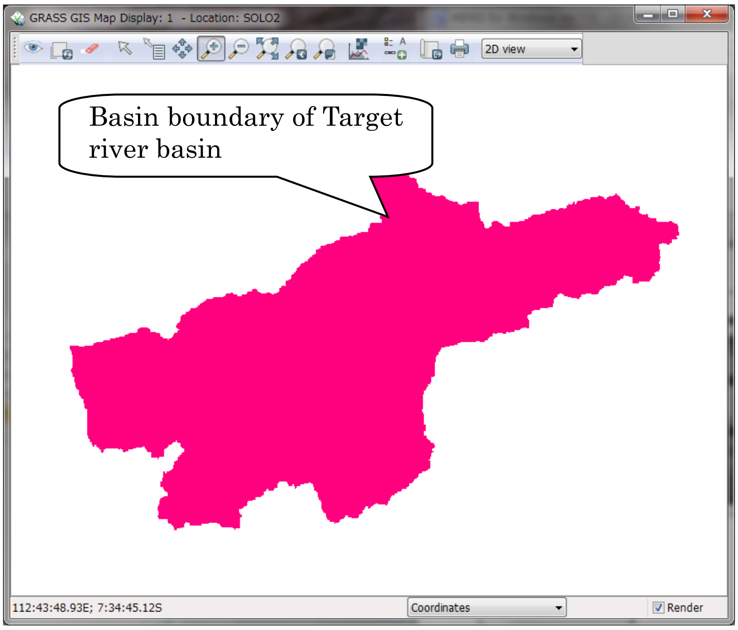
⑲ To create basin boundary, select the outlet of target river basin.
[Raster]>[Hydrologic modeling]>[Watershed basin creation (r.water.outlet)]



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".

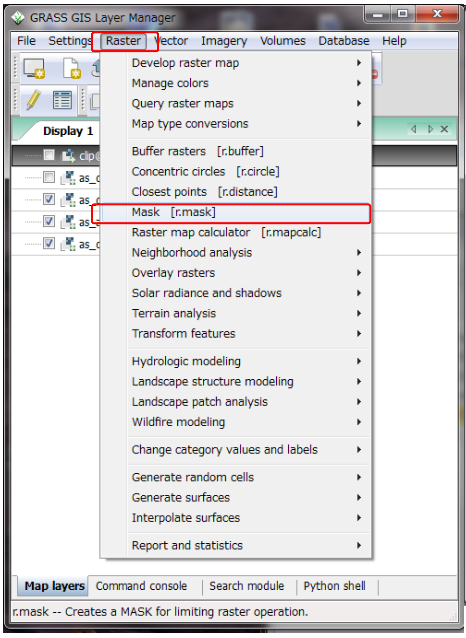
㉑ Input x-coordinate(long.) of the outlet in "The map E gird coordinates" and input y-coordinate(lat.) of the outlet in "The map N grid coordinates" and click "Run". Then, basin boundary layer of target river basin will be shown.

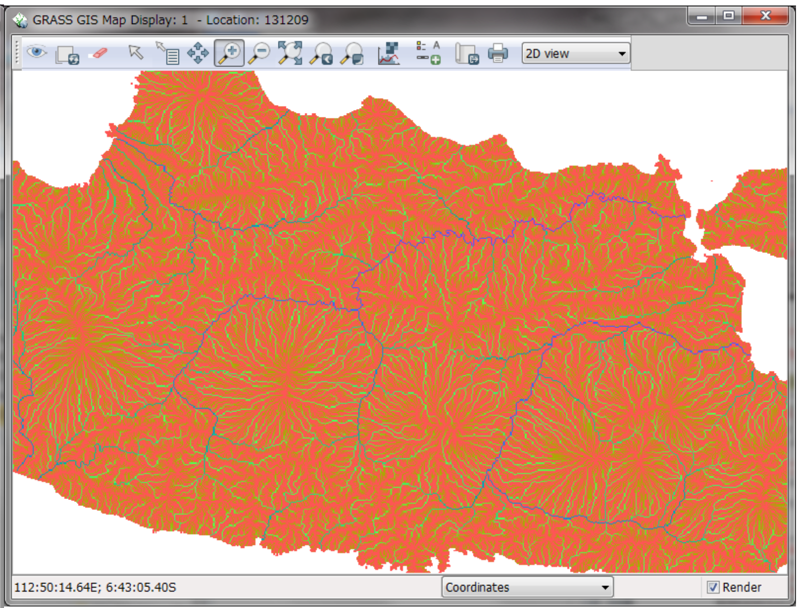




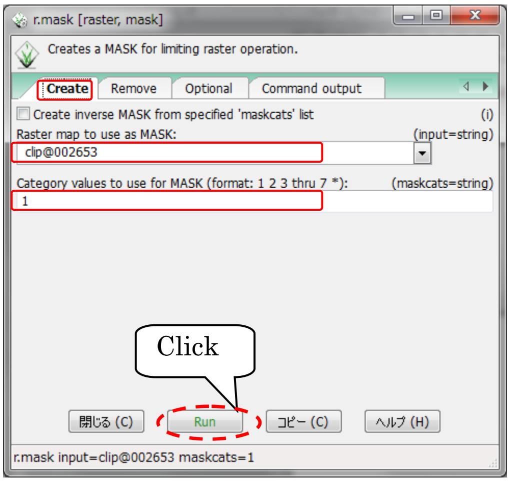
㉒ Clip target river basin by using basin boundary layer.

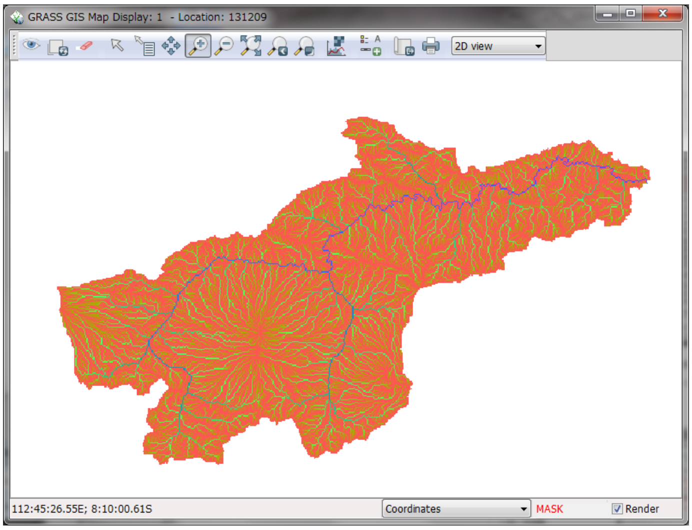
[Raster]>[Mask(r.mask)]





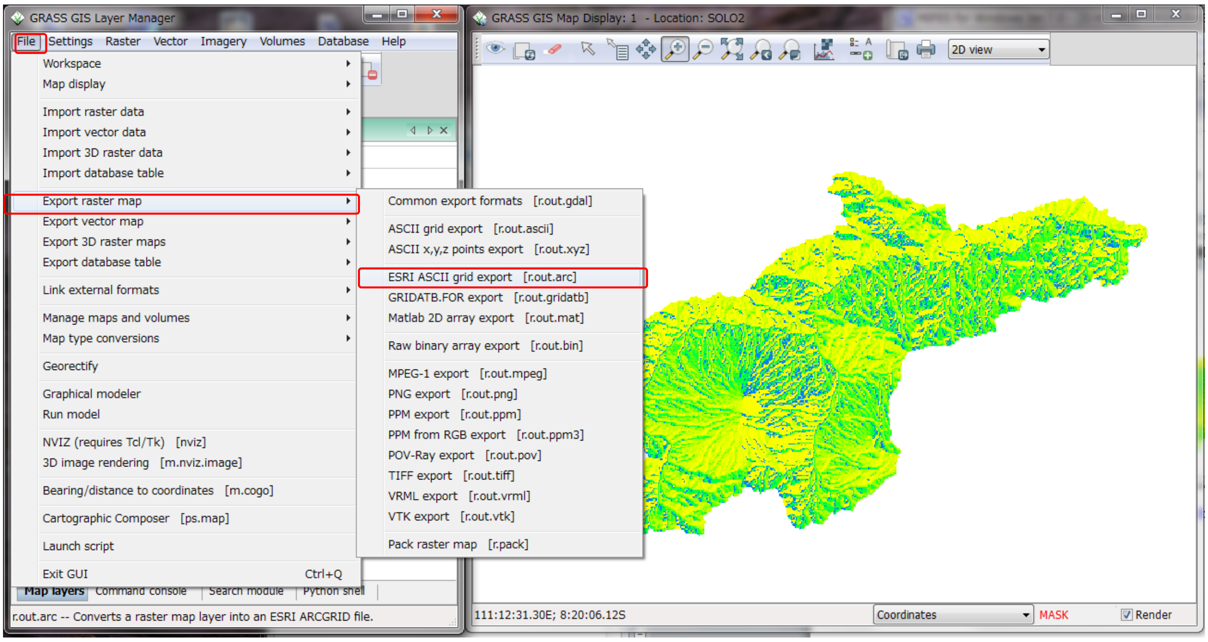
㉓ Select basin boundary layer as “Raster map to use as MASK” and input “1” in “Category values to use for MASK” on “Create” tab and click “Run”. Then, clipped target river basin will be shown.





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

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



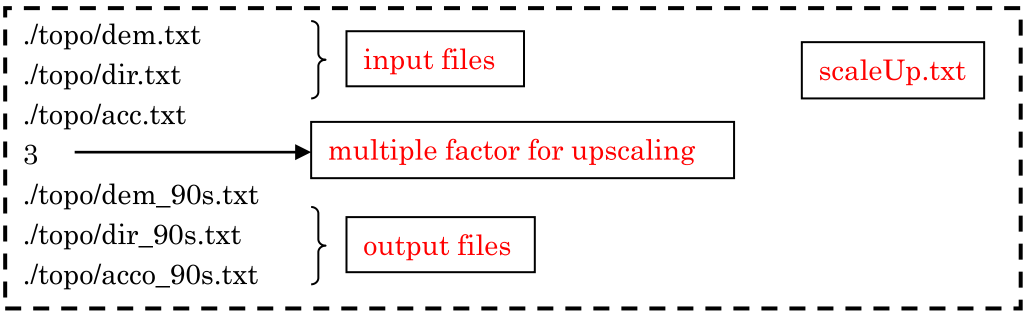
㉕ Select three layer data (dem, dir, acc) and input output file name in "Name for output ARC-GRID map" and click "Run".

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 "scaleUp.exe". 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 Copy "scaleUp.txt" file from "RRI-CUI/etc/scaleUp/" and save it under your project folder (e.g. RRI-CUI/Project/solo30s/)



② Type in "scaleUp.exe" and return to execute scaleUp.exe program 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 (demAdjust2.exe) 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.

Edit demAdjust2.txt if necessary and run demAdjustment2 program by typing in "demAdjustment2.exe" on Command Prompt under the project folder (e.g. solo15s or solo30s).

> 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),

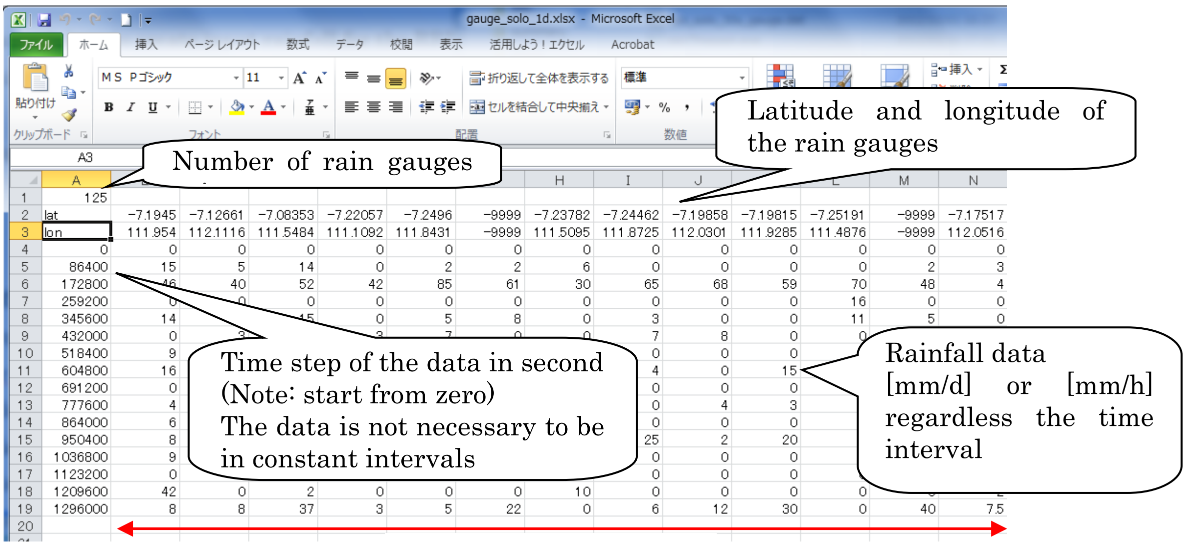
2) GSMaP satellite based rainfall (/etc/GSMaP) and

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 rainThiessen.exe (./RRI-CUI/etc/rainThiessen/rainThiessen.f90) program.

① First, prepare rain gauge data in Excel (e.g. /solo30s/rain/gauge_solo_1d.xlsx).



Number of rain gauges

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_1d.txt)

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

```
./rain/gauge_1d.txt
24
./rain/rain.dat
./rain/gauge_map.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 On command prompt under the project file (e.g. RRI/Project/solo30s/), type in "rainThiessen.exe" to execute the program.

⑤ Confirm an input rainfall file for RRI Model (e.g. "rain.dat") is newly created.

4.2 Prepare Input Rainfall Data from GSMaP

GSMaP products were updated on September 2014. Now the products include GSMaP_NRT (realtime), GSMaP_MVK (standard ver.5 or ver.6) and GSMaP_Gauge (gauge composite). Refer to the following website for the latest information and the registration to download the data. (http://sharaku.eorc.jaxa.jp/GSMaP/index.htm)

4.2.1 Download GSMaP Data

① First, create "gsmap" folder under your project folder (e.g. solo30s).

② Under "gsmap" folder, create "infile" and "cutfile" folders.

③ Download all GSMaP rainfall data you would like to proceed and save them in "infile".

4.2.2 Calculate Rainfall Data Range for a Target Catchment

To calculate the range for the data delineation, calc_area_gsmap.exe can be used. Before executing calc_area_gsmap.exe, copy /etc/GSMaP/calc_area_gsmap.txt and paste it under the created "gsmap" folder.

In the copied “calc_area.txt”, specify “horizontal_resolution [d]” and “temporal_resolution [h]” of original GSMaP product you will use. Also specify “ncols” to “cellsize” based on the target catchment, whose parameters can be obtained from the headers of topographic files of “dem”, “acc” or “dir”.

Note: The holizontal resolution of GSMaP product is either 0.1 [deg] or 0.25 [deg], and the temporal resolution is either 1 [h] or 24 [h].

(In case of 0.1 deg → xul = 0.05, yul = 59.95; 0.25 → xul = 0.125, yul = 59.875).

Running calc_area_gsmap.exe creates a file named “out_by_calc_area_gsmap.txt”.

out_by_calc_area_by_gsmap.txt

```
horizontal_resolution [d] :   0.1000000
temporal_resolution [h]   :           1

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
```

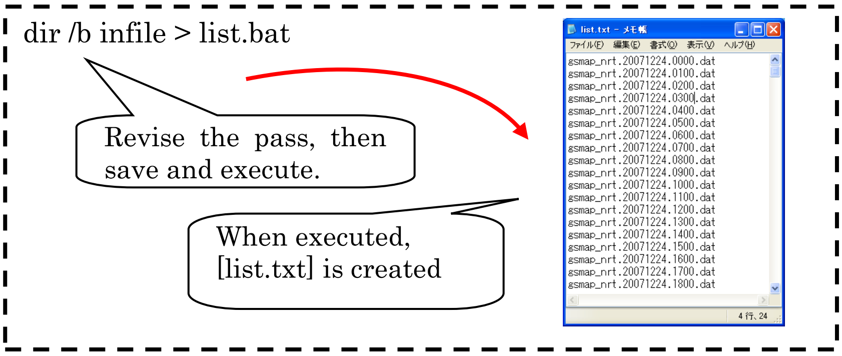
Information that is necessary to delineate GSMaP (jleft, ibottom, iright, itop)

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

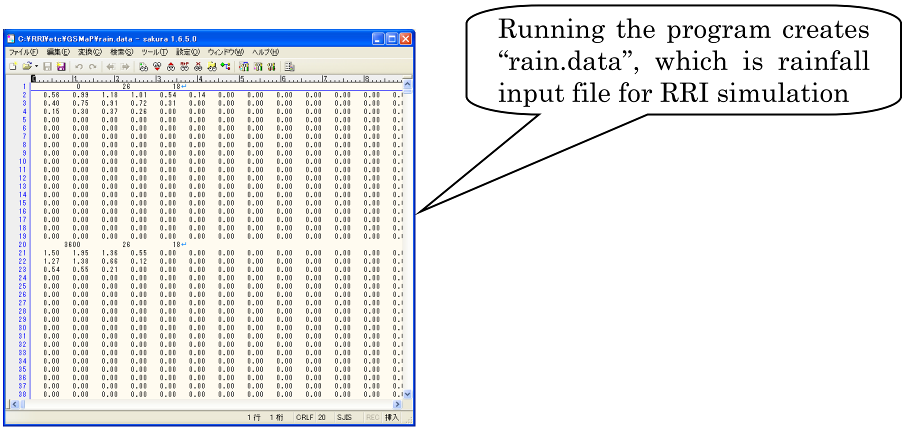
4.2.3 Delineating GSMaP Data for Target Area

④ Unzip all the downloaded files (a linux user may use “gunzip *”, otherwise please find an appropriate program to unzip “.dat.gz” files)

⑤ Copy “/etc/GSMaP/makeList.bat” under “gsmap” folder and execute it to list up all the unziped files as "list.txt”



⑥ Type in “read_gsmap.exe” on command prompt at “gsmap”.



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

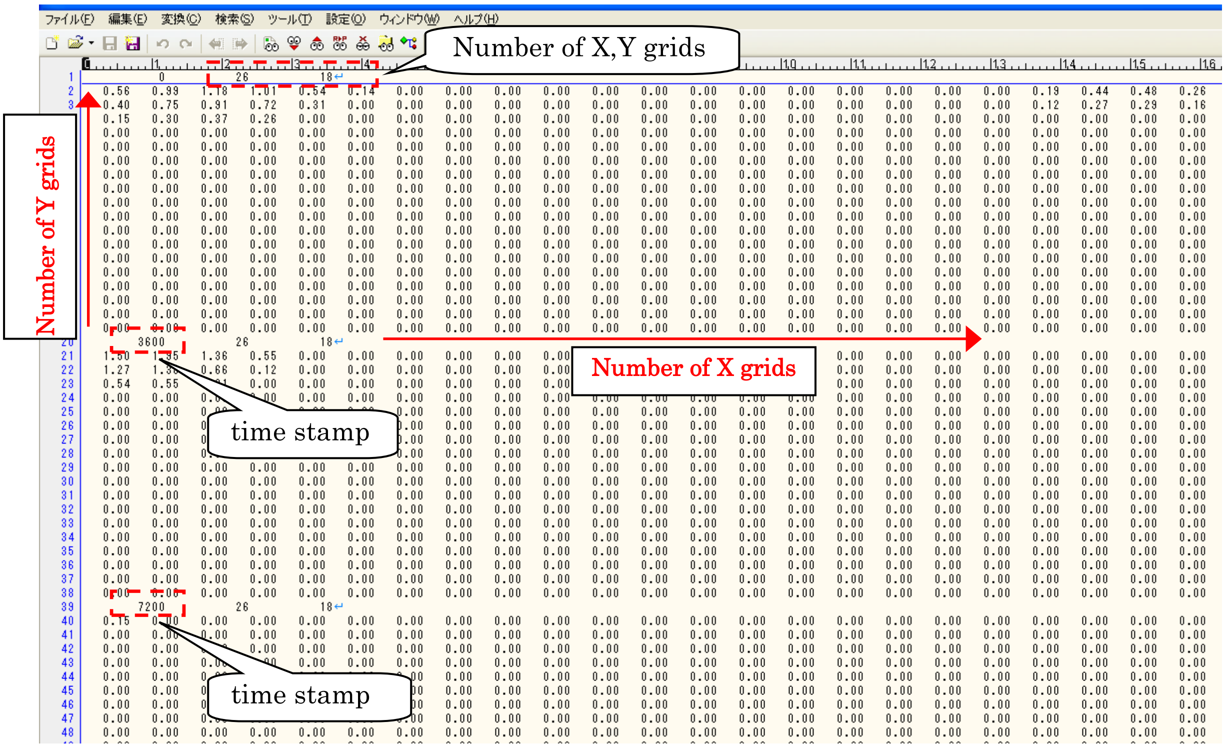
Note: the extract does not run if the same output files already exist

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_2

./rain/rain.dat
./topo/adem.txt
./topo/acc.txt
./topo/adir.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 "rainBasin.exe" program. To run the program, "rainBasin.txt" must be prepared in the following way.

rainBasin.txt

```
./rain/rain.dat
./topo/adem.txt
110.2d0
-8.3d0
0.00833333d0 0.00833333d0
./rain/rain_hyeto.txt
./rain/rain_dist.txt
./rain/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]

On command prompt, type in "rainBasin.exe" to create three output files identified in L6, L7 and L8 to show hyetograph, total rainfall distribution map and cumulative rainfall, respectively.

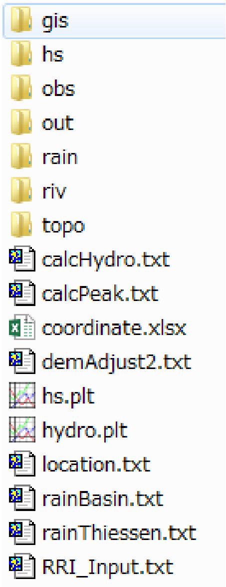
Note: To calculate average rainfall over a sub-catchment, one can replace the file indicated in L2. First, one can use GIS to delineate the sub-catchment and convert the mask into ASCII GIS format. For areas having pixel values greater than -10 will be considered as a sub-catchment area.

5. Conditions Setting for RRI Simulation

5.1 Folder Configuration

The following shows the folder configuration inside ./RRI-CUI/Project/solo30s

[./**RRI-CUI/Project/solo30s**]



Folders

topo : Stores following sets of topographic data

- Digital elevation model (dem.txt)
- Adjusted digital elevation model (adem.txt)
- Flow accumulation (acc.txt)
- Flow direction (dir.txt)
- Adjusted flow direction (adir.txt)
- (optional) Land use data (landuse.txt)

rain : Stores following sets of input rainfall data

- Rainfall data (rain.dat)
- (optional) Evapotranspiration data (PET.txt)

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

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

gis : Stores GIS related data

obs: Stores observation related data

riv: Stores river section related data

【Control files】

- RRI_Input.txt : RRI model control file for 0_rri_1_4_2.exe
- demAdjust2.txt : demAdjustment program (demAdjust2.exe) [pre-processing]
- rainThiessen.txt : Rainfall processing program (rainThiessen.exe) [pre-processing]
- calcHydro.txt : Hydrograph calculation program (calcHydro.exe) [post-processing]
- calcPeak.txt : Peak inundation depth calculation program (calcPeak.exe) [post-processing]
- rainBasin.txt : Rainfall analysis program (rainBasin.exe) [post-processing]

【Input Files Other Programs and Files】

- hydro.plt : gnuplot script to draw hydrograph
- hs.plt : gnuplot script to create inundation depths figures (prepared by /etc/prepHsPlt)
- ciirdubate.xlsx: excel file to covert from (i, j) to (x, y) or (x, y) to (i, j)
- location.txt : Location list to draw hydrographs

5.2 RRI Model Control File (RRI_Input.txt)

RRI_Input.txt

L1	RRI_Input_Format_Ver1_4_2	
L2		
L3	./rain/rain.dat	
L4	./topo/adem.txt	
L5	./topo/acc.txt	
L6	./topo/adir.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 | 1.0d0       # soildepth
L23 | 0.475d0     # gammaa
L24 |
L25 | 0.d0        # kv (m/s)
L26 | 0.3163d0    # Sf (m)
L27 |
L28 | 0.0d0       # ka (m/s)
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. "ka" 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 "gammaa" within the program.

RRI_Input.txt

L32	0.d0	# ksg (m/s) – set zero for no bedrock gw (1.5d-5)
L33	0.037d0	# gammag (-)
L34	5.7d-5	# kg0 (m/s)
L35	0.1d0	# fg(-)
L36	0.d0	# rgl (m/s)

L32 – L36

Set "ksg = 0.d0" to avoid deep groundwater component. L33-L36 become inactive when ksg = 0.d0. In case to activate the deep groundwater component, make sure to set appropriate initial condition after the sufficient spin-up computation. Otherwise the model will start with hg = 0, which means the deep groundwater is saturated within the bedrock. Note hg is the distance from the soil-bedrock interface to the groundwater table within the bedrock.

L32 : ksg: vertical hydraulic conductivity at the top of bedrock [m/s]

L33 : gammag: porosity of the bedrock [-]

L34 : kg0: lateral hydraulic conductivity at the surface of the bedrock [m/s]

L35: fg: exponential decay of the hydraulic conductivity of the bedrock

L36: rgl (m/s): vertical loss of groundwater from the bottom of the bedrock (typically set to be zero to maintain the water balance inside the model.

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 [km²] 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.

RRI_Input.txt

L38	100	# riv_thresh
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	./riv/width.txt	
L48	./riv/depth.txt	
L49	./riv/height.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.

RRI_Input.txt

```
L51  0 0 0 0
L52  ./init/hs_init_dummy.out
L53  ./init/hr_init_dummy.out
L54  ./init/hg_init_dummy.out
L55  ./init/gamptff_init_dummy.out
L56
L57  0 0
L58  ./bound/hs_bound.txt
L59  ./bound/hr_bound.txt
L60
L61  0 0
L62  ./bound/qs_bound.txt
L63  ./bound/qr_bound.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 ./topo/landuse.txt
L67
L68 0
L69 ./dam.txt
L70
L71 0
L72 ./div.txt
L73
L74 0
L75 ./infile/PET.txt
L76 110.2d0                 # xllcorner_evp
L77 -8.3d0                  # yllcorner_evp
L78 0.00833333d0 0.00833333d0     # cellsize_rain
L79
L80 0
L81 ./riv/length.txt
L82
L83 0
L84 ./riv/sec_map.txt
L85 ./riv/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 L75. 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	1
L100	./location.txt

L87 – L97 : Output file settings

Change the settings of L87 to "1" to output different sets of simulation results listed in the same order between L88 and L97

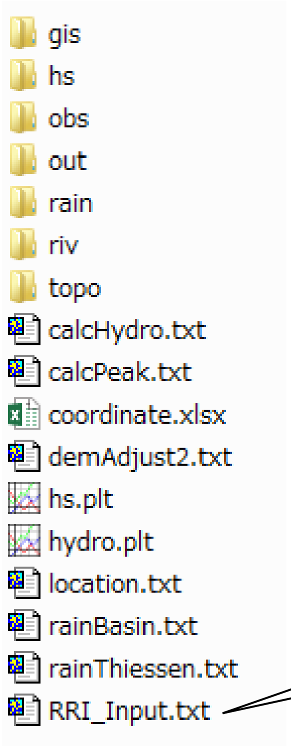
L99 – L100 : Output hydrographs at specified locations (Optional)

Set 1 in L99 to read the location file and output hydrographs at the specified locations.

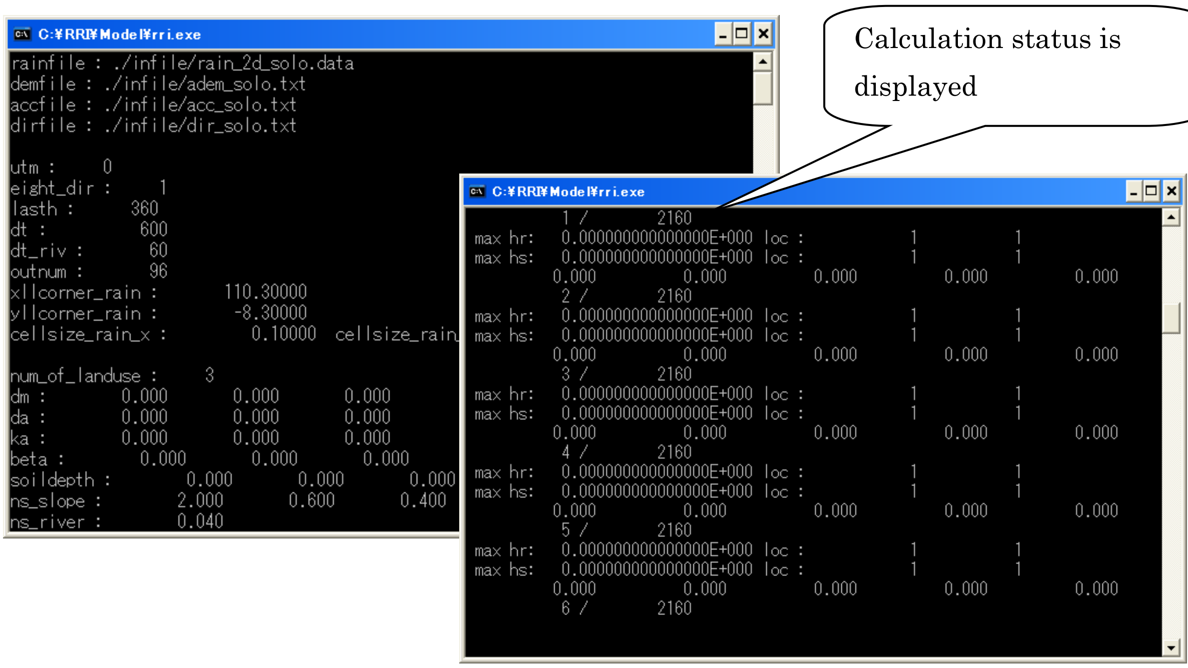
6. Running RRI Model

① Prepare “RRI_Input.txt” under your project folder (e.g. “./RRI-CUI/Project/solo30s)”

② Move current folder to your project folder and type in “0_rri_1_4_2.exe” and return.



Prepare “RRI_Input.txt” before executing 0_rri_1_4_2.exe



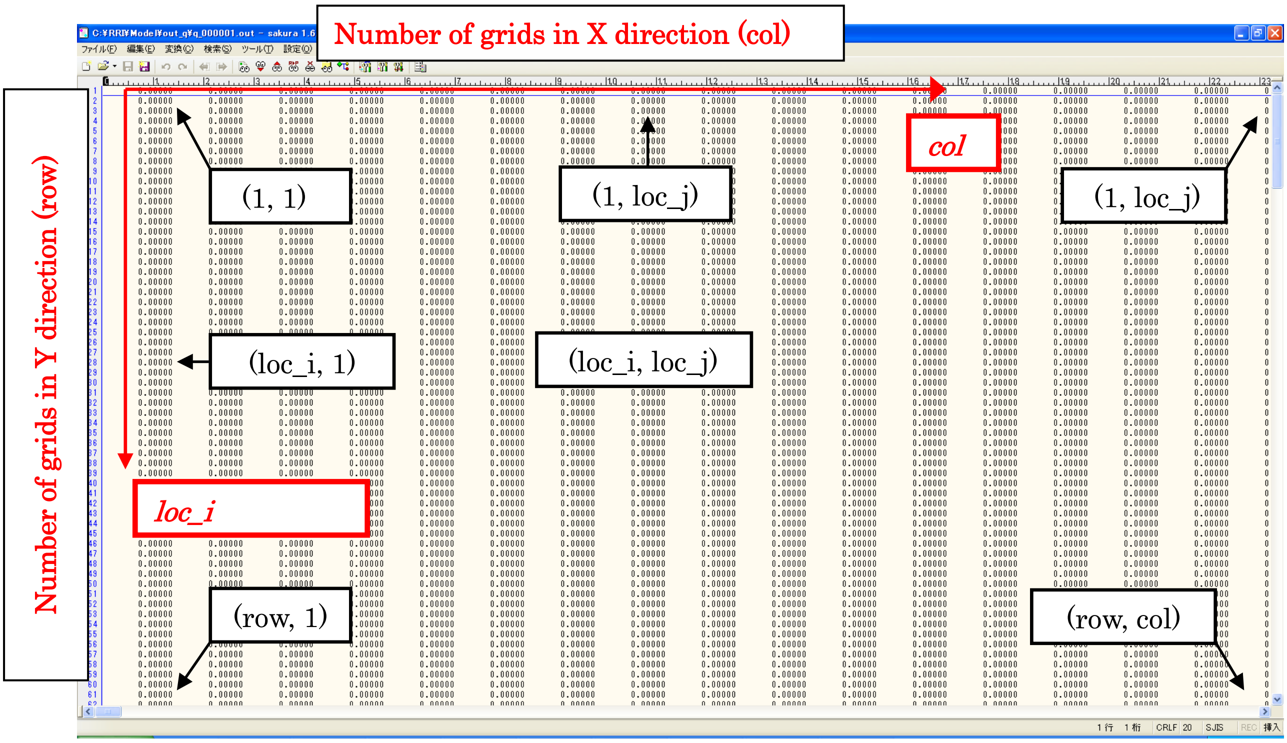
Calculation status is displayed

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 [m^3/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. Inside the project folder, 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"
```

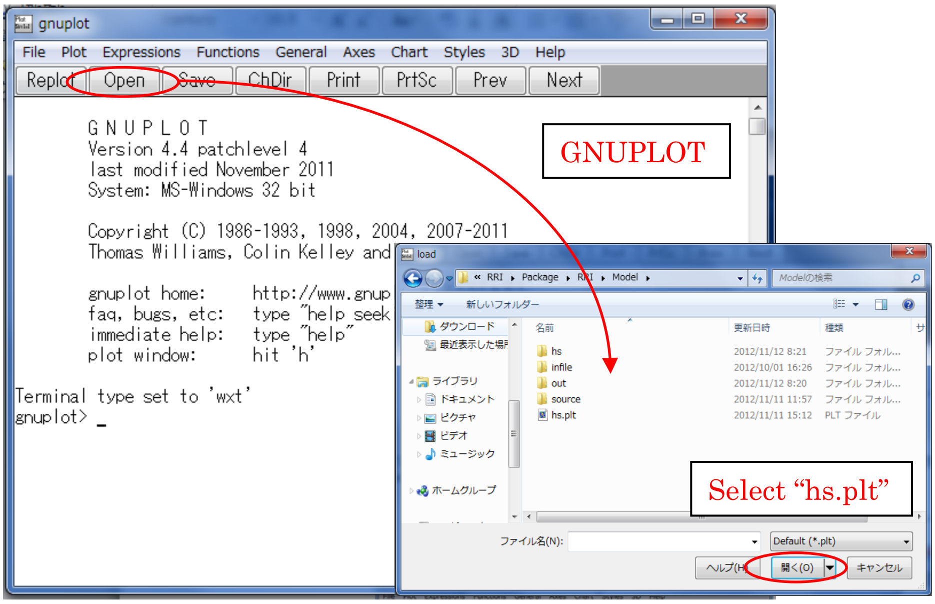
The size of output GIF file, X and Y direction. Use the same X and Y ratio as DEM's col and row.

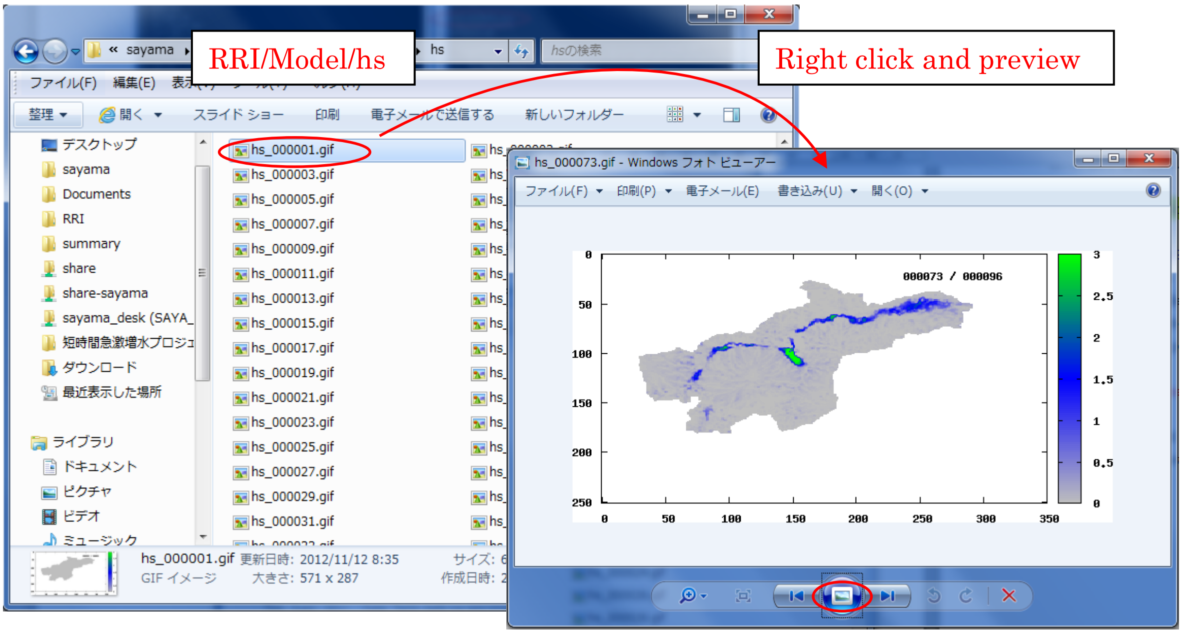
Color pattern settings

Color range

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

② Start GNUPLOT program by clicking “/RRI-CUI/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-CUI/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_)

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

calcHydro.txt

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

location_30s_solo.txt

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

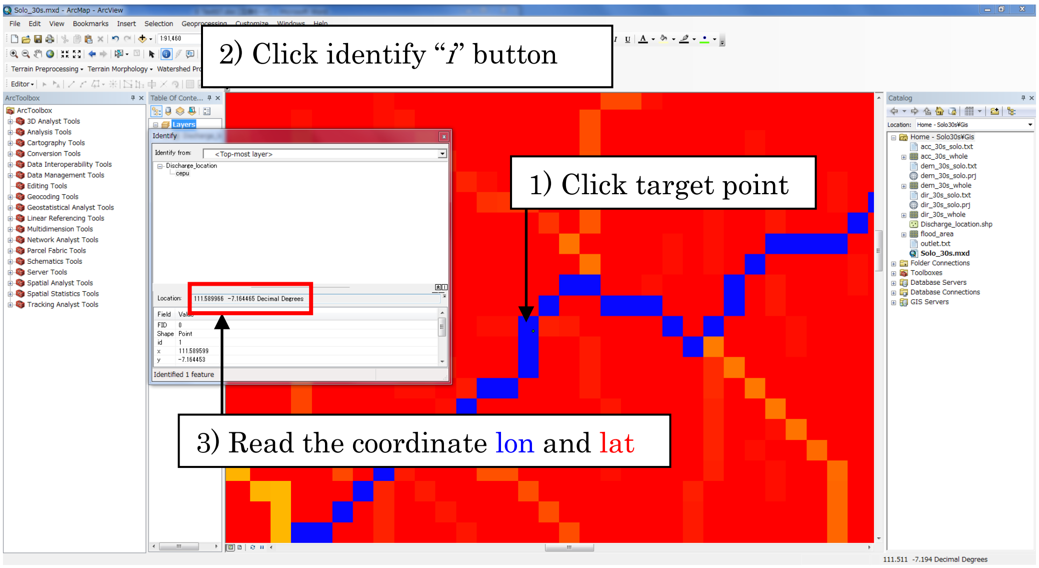
③ Check the created files specified in L3 of "calcHydro.txt". (e.g. ./infile/solo30s/disc_)

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.



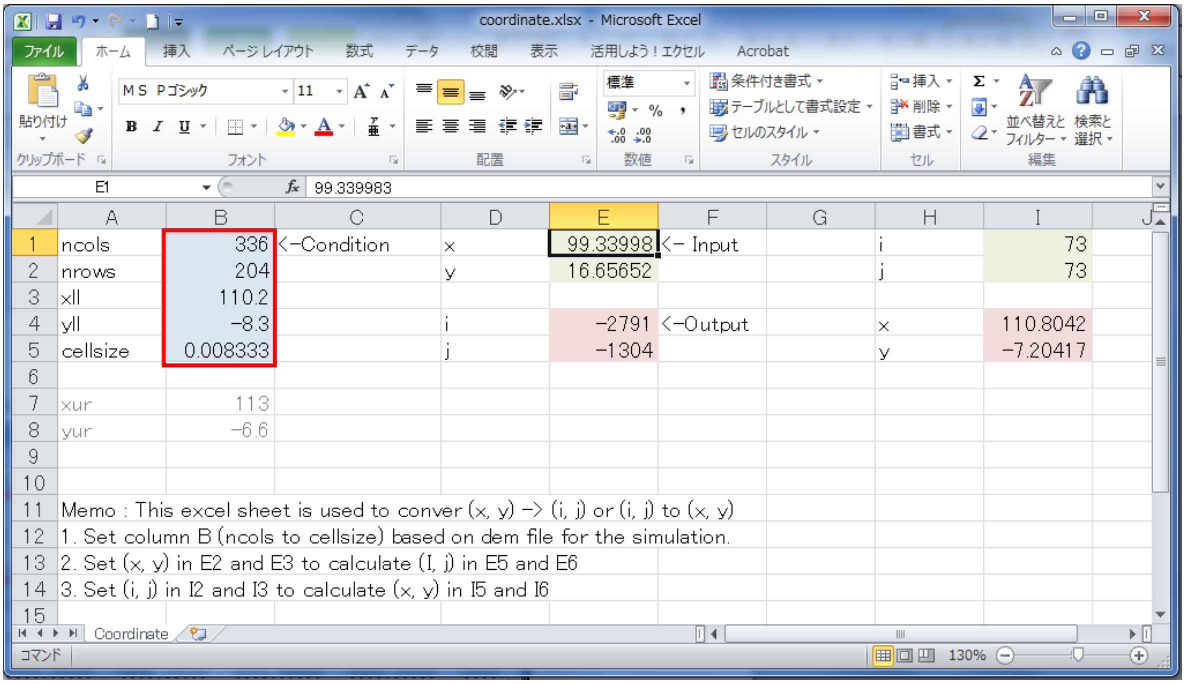
(Displaying "acc" on top to make sure the selected point is on a river grid cell.)

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-CUI/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

modify

add

```
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"
…
```

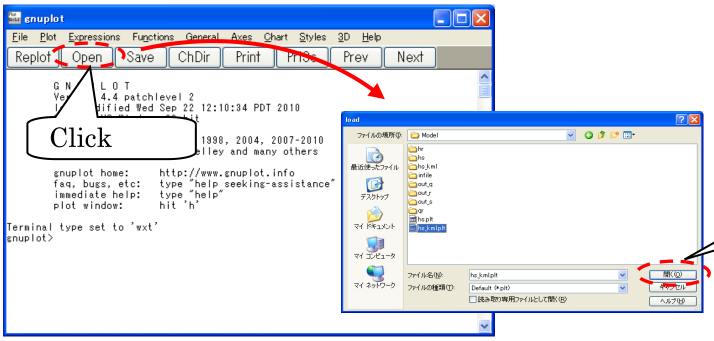
Designate size so that the aspect ratio of size and ratio of number of meshes match.

This part must be added to the original hs.plt file.

The folder name, “hs_kml” should be input here.

modify

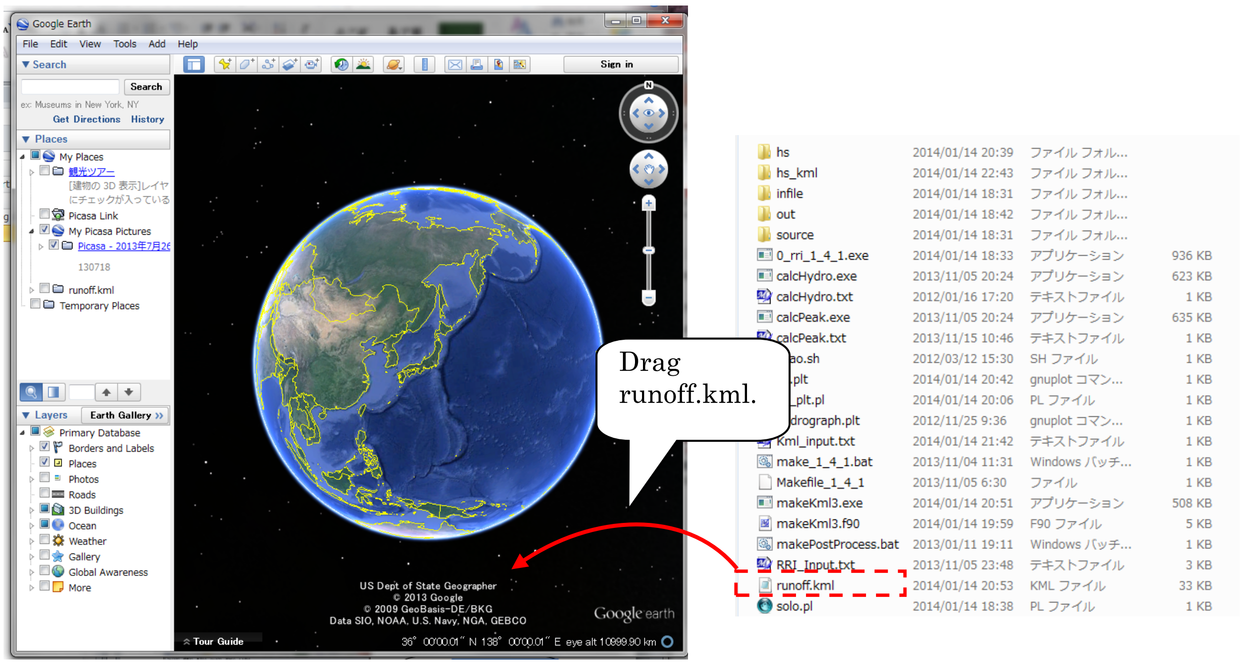
② Start “GNUPLOT” and run “RRI/Model/ hs_kml.plt”.

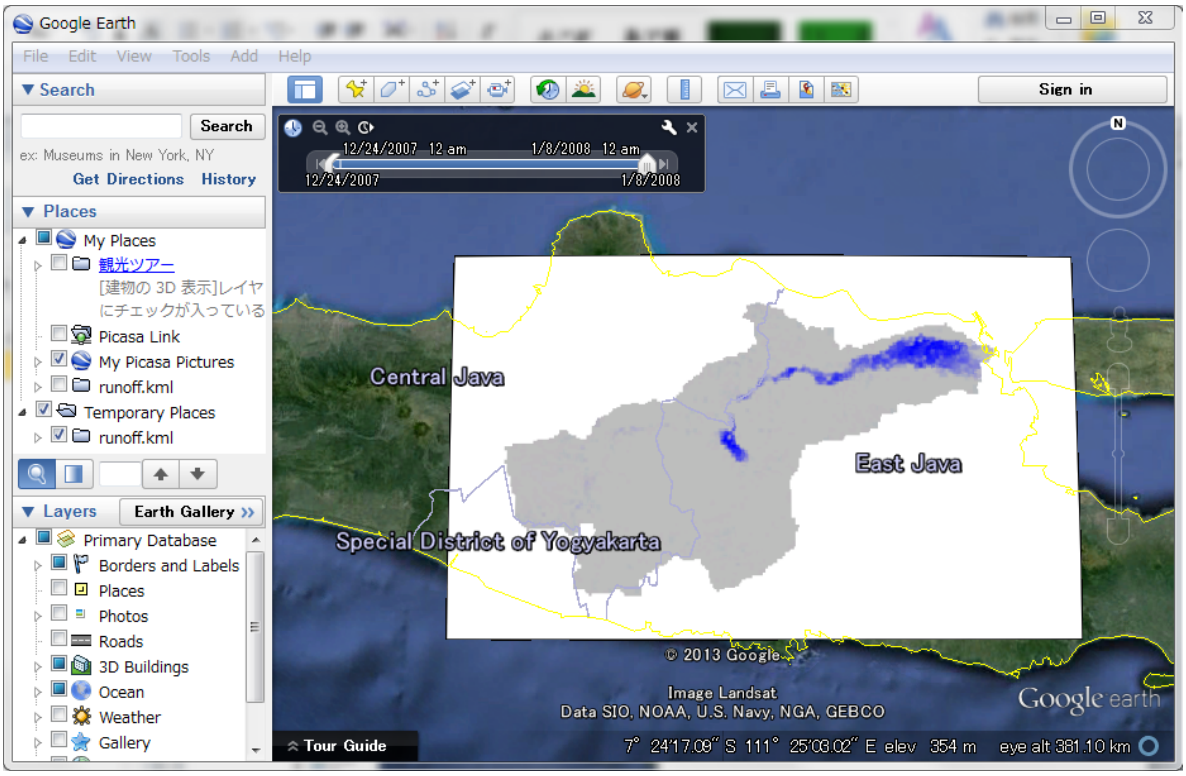


③ An image file is prepared in the “RRI/Model/hs_kml” folder. (Note that a new folder hs_kml must be created in advance.)

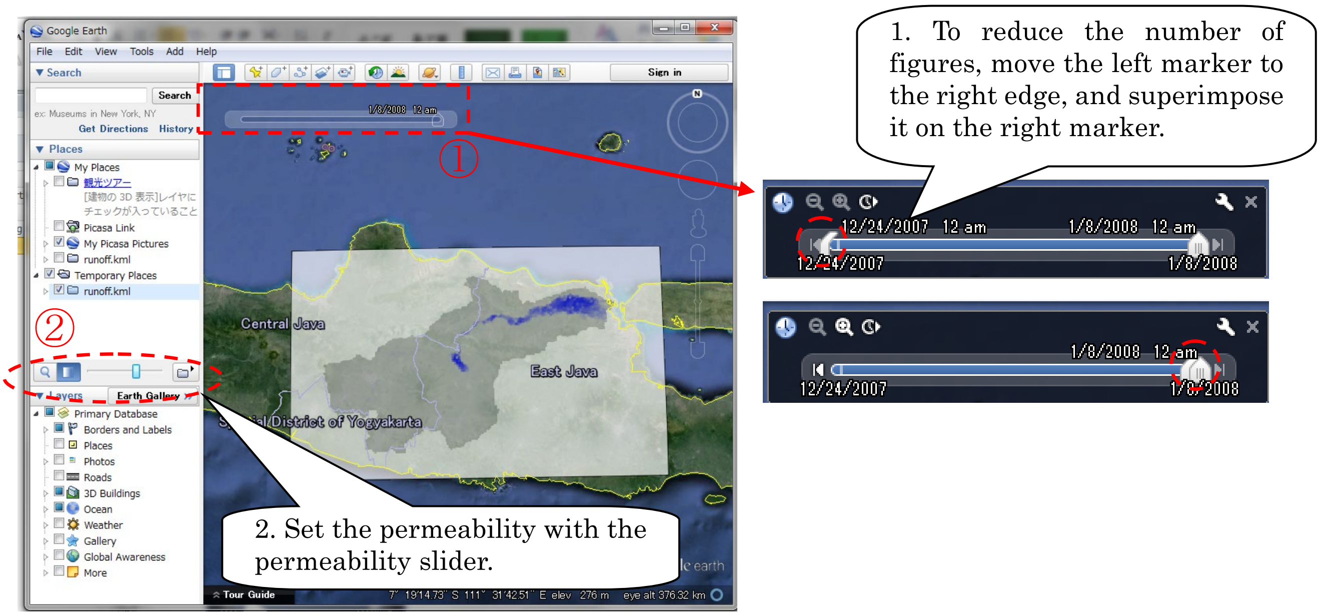
7.5.3 Visualize GIF files with Google Earth

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



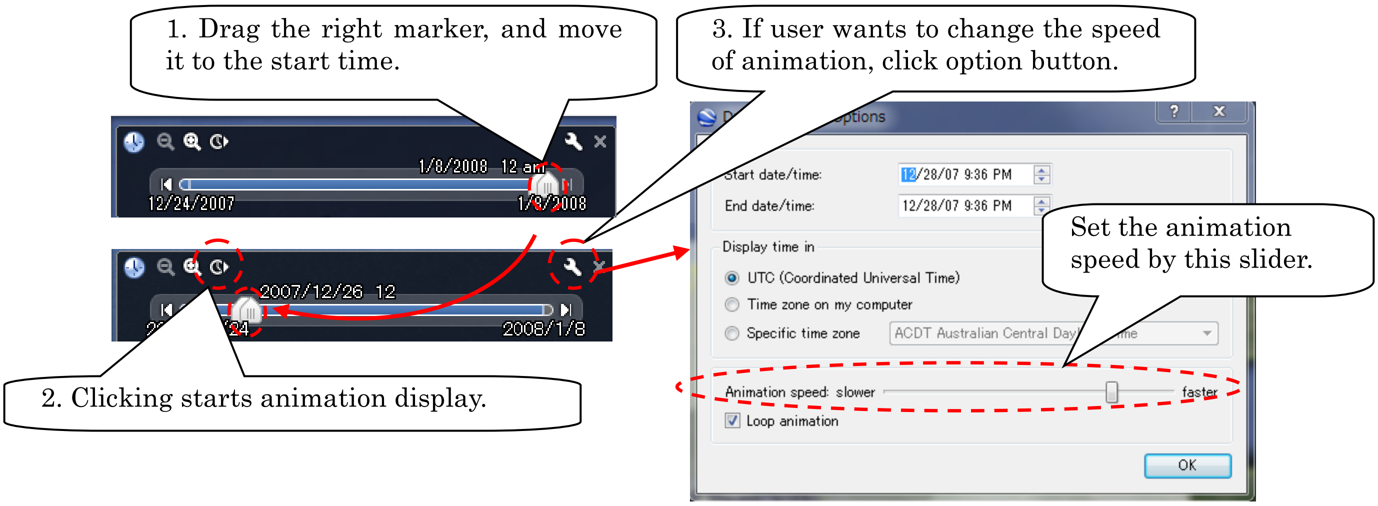


② Designate the number of figures to display at once and their transparency.



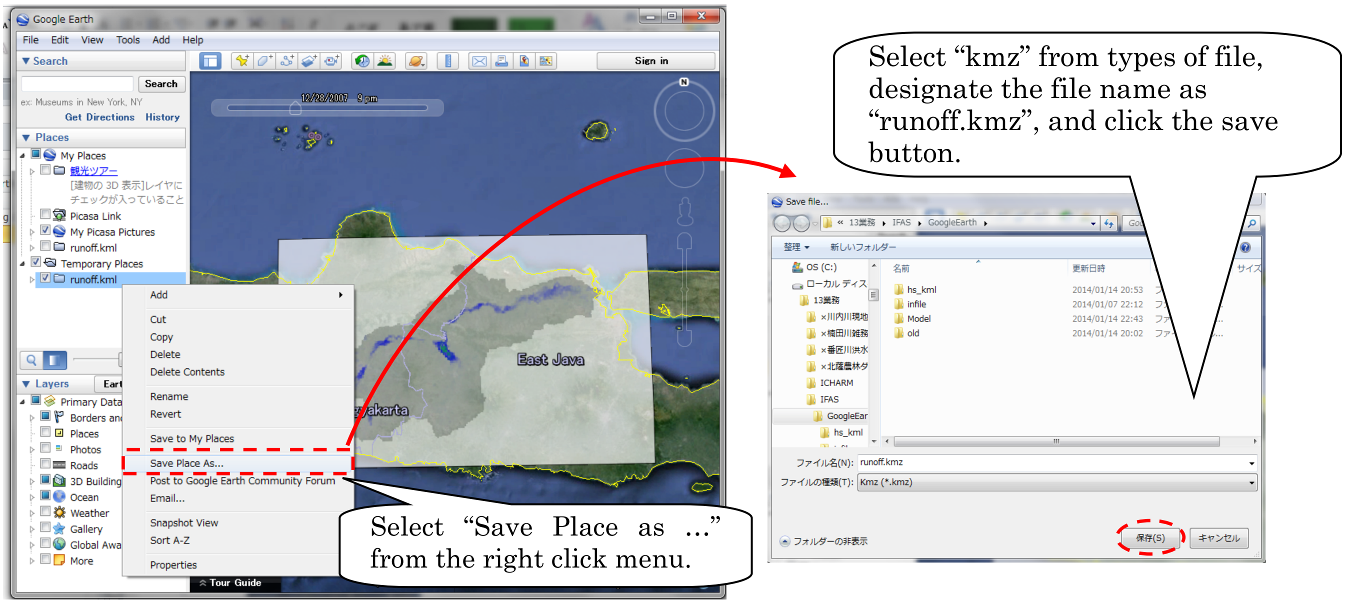
※ On time slider: The right marker represents the present time, while the left marker is used for the number of figures to overlay. Figures in the period between two markers are displayed.

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.

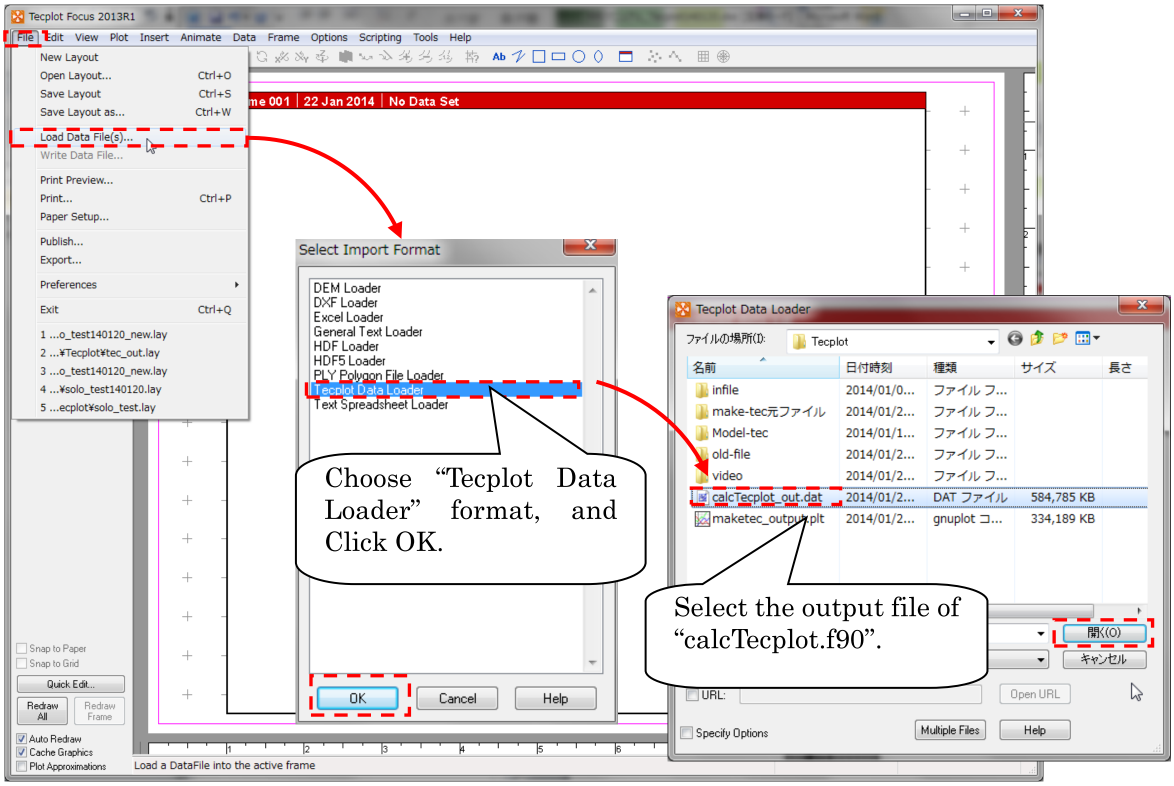
Use "RRI/etc/calcTecplot.f90" to prepare an input file for Tecplot (e.g. "calcTecplot_out.dat"). Prior to running calcTecplot.exe, edit "RRI/etc/calcTecplot.txt", which sets the condition for generating the input file.

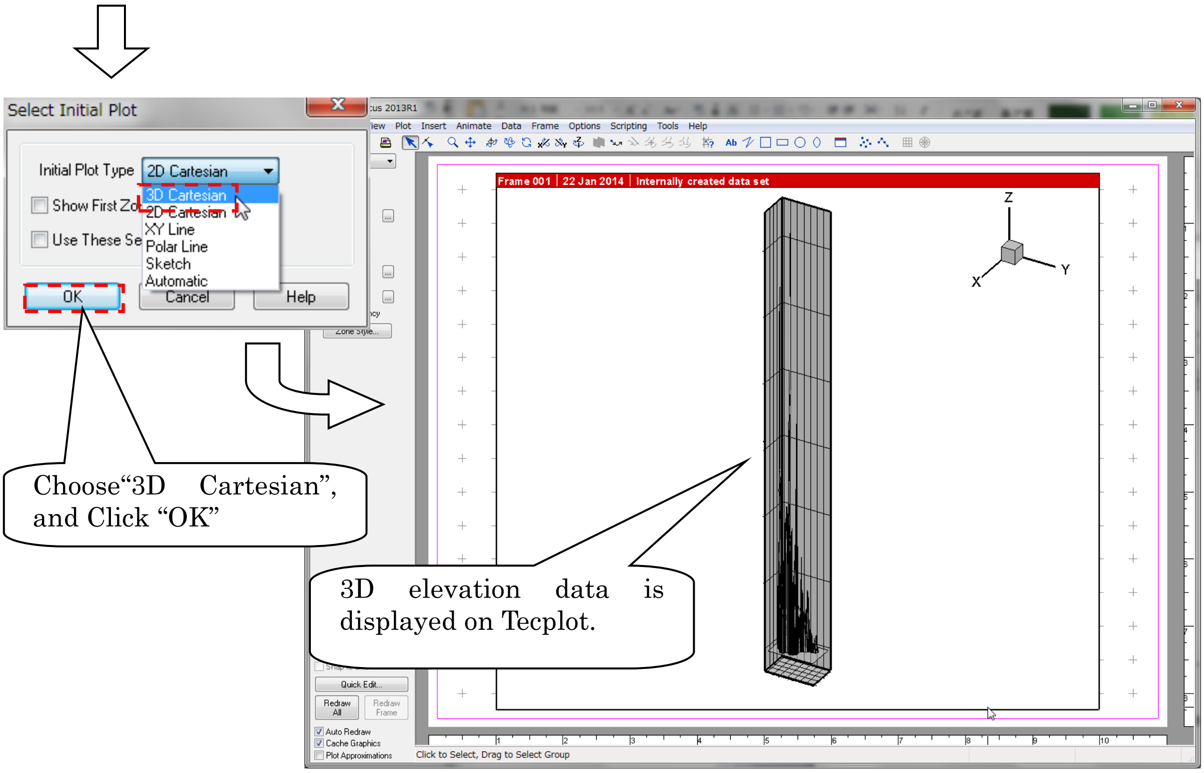
7.6.2 Displaying on Tecplot

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.



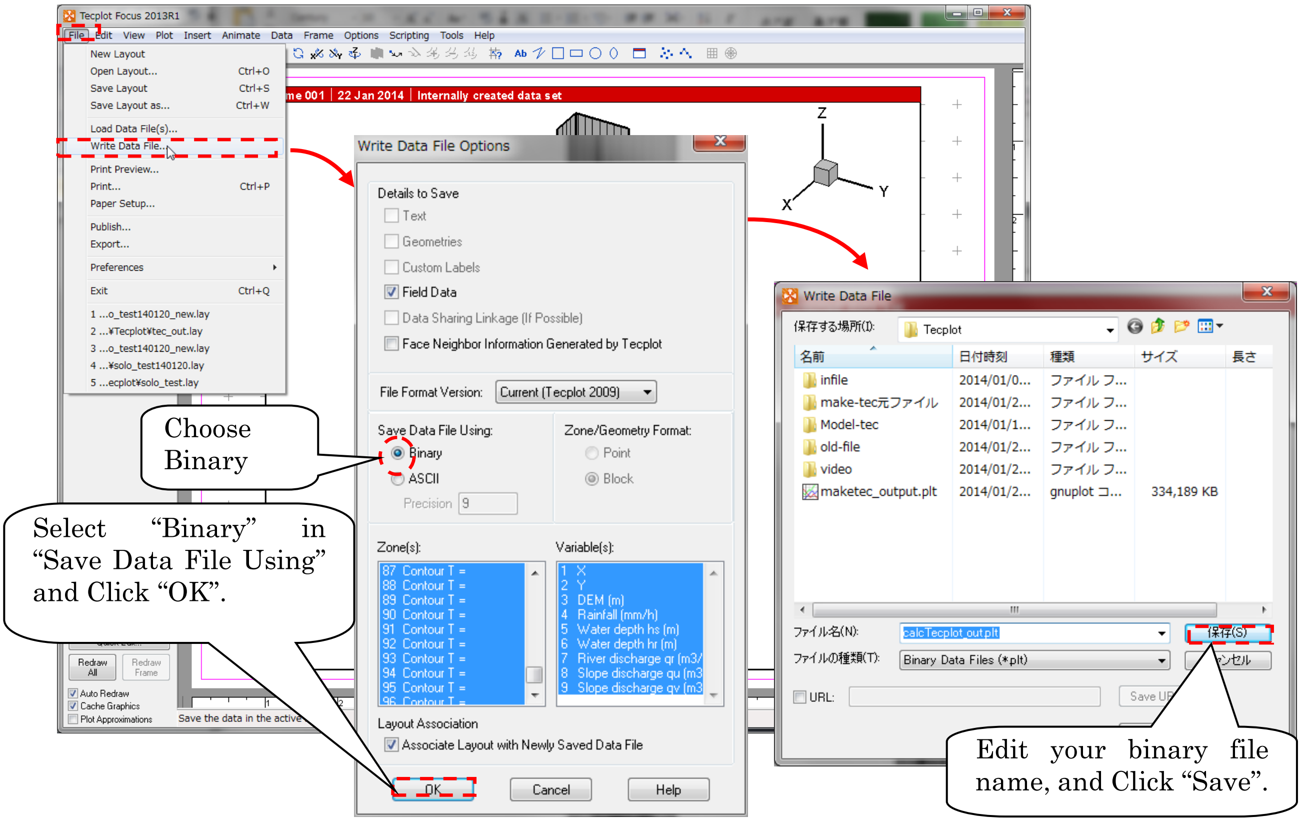


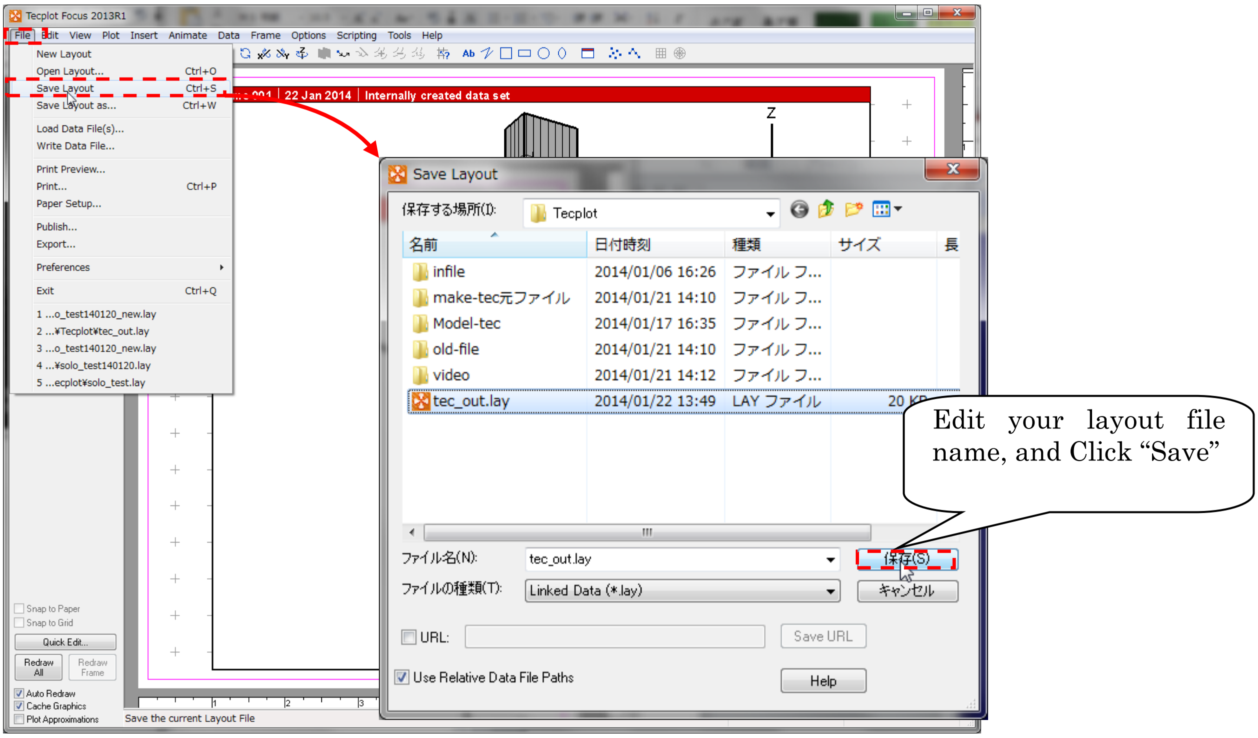
② Write data file (changing input data to binary data), and also save as layout file.

[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.

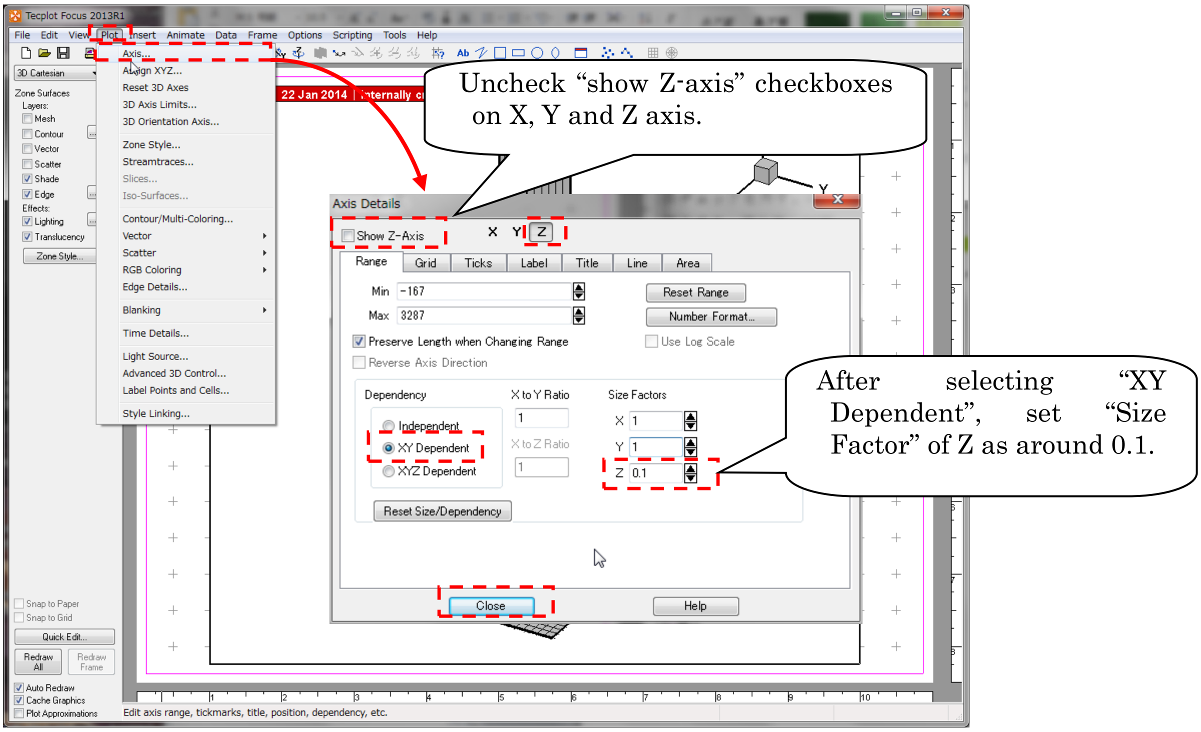




7.6.3 Edit display options on Tecplot

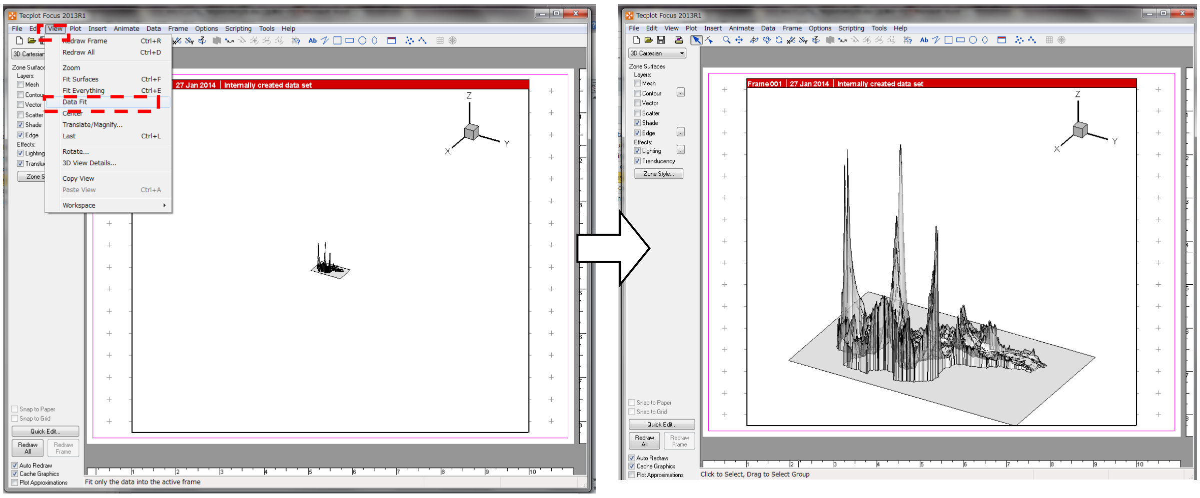
① Edit the ratio of XYZ and hide the axes.

[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.



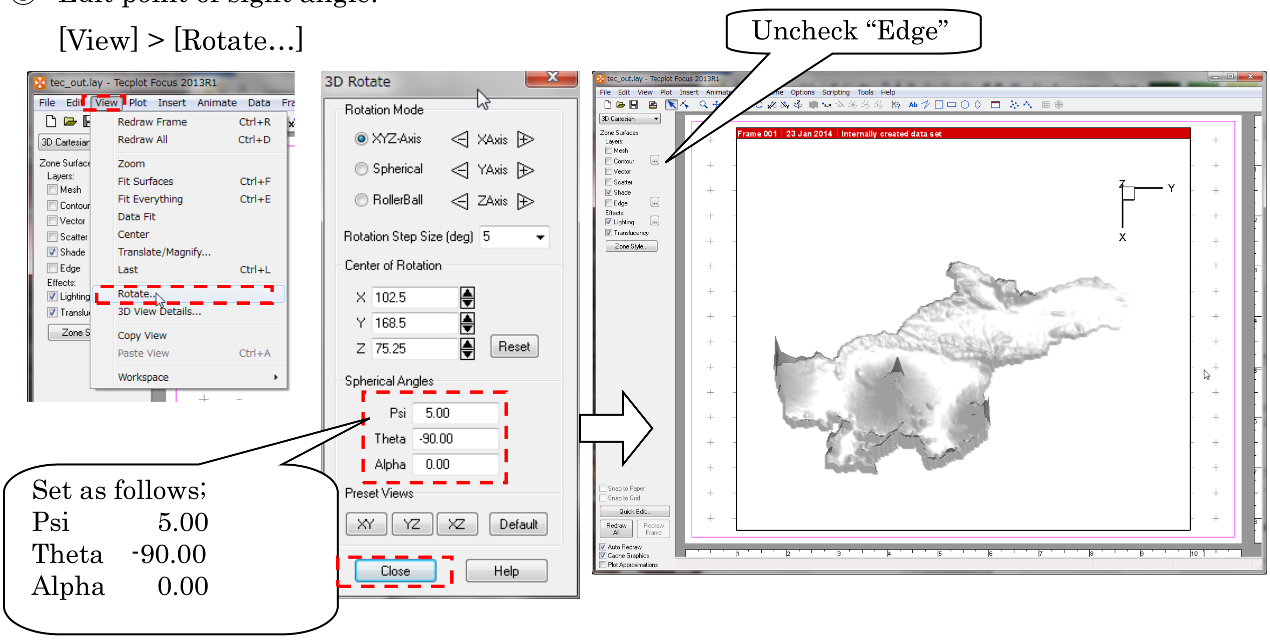
② Fit the data display range to the target range.

[View] > [Data fit]



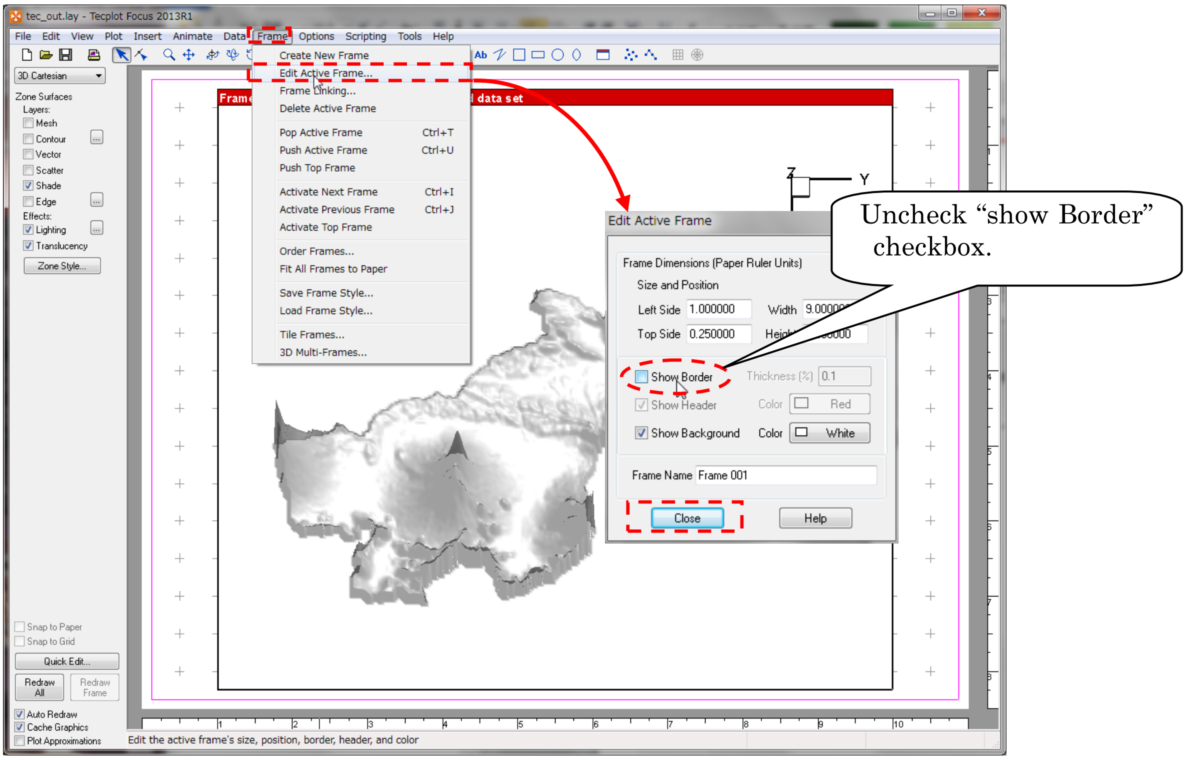
③ Edit point of sight angle.

[View] > [Rotate…]



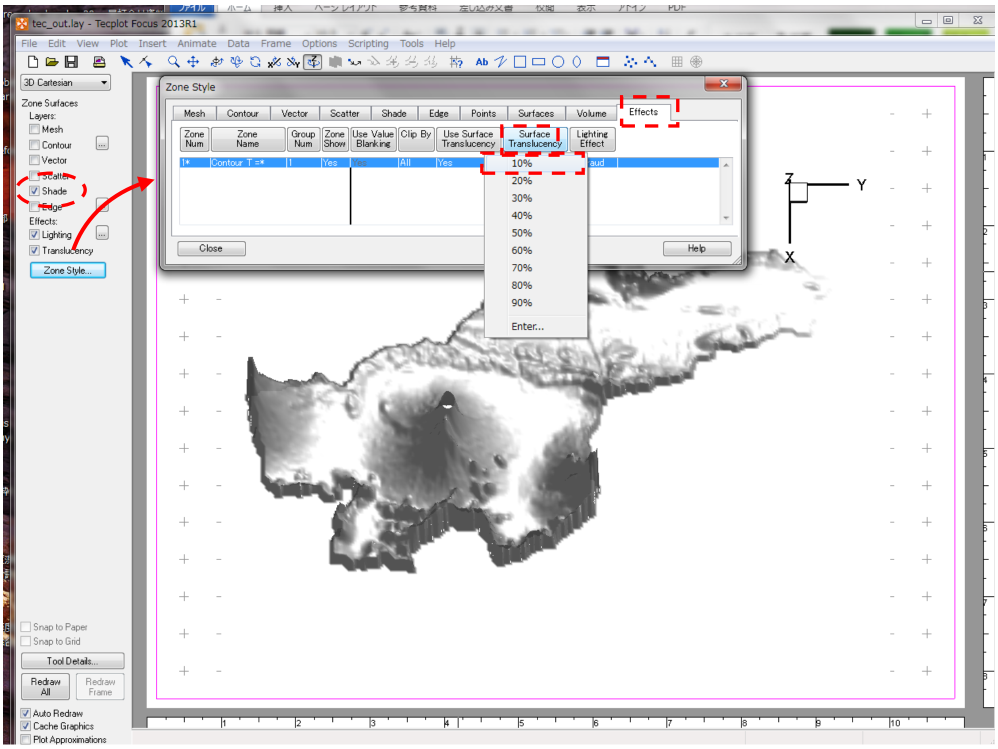
④ Delete unnecessary frame

[Frame] > [Edit Active Frame…]



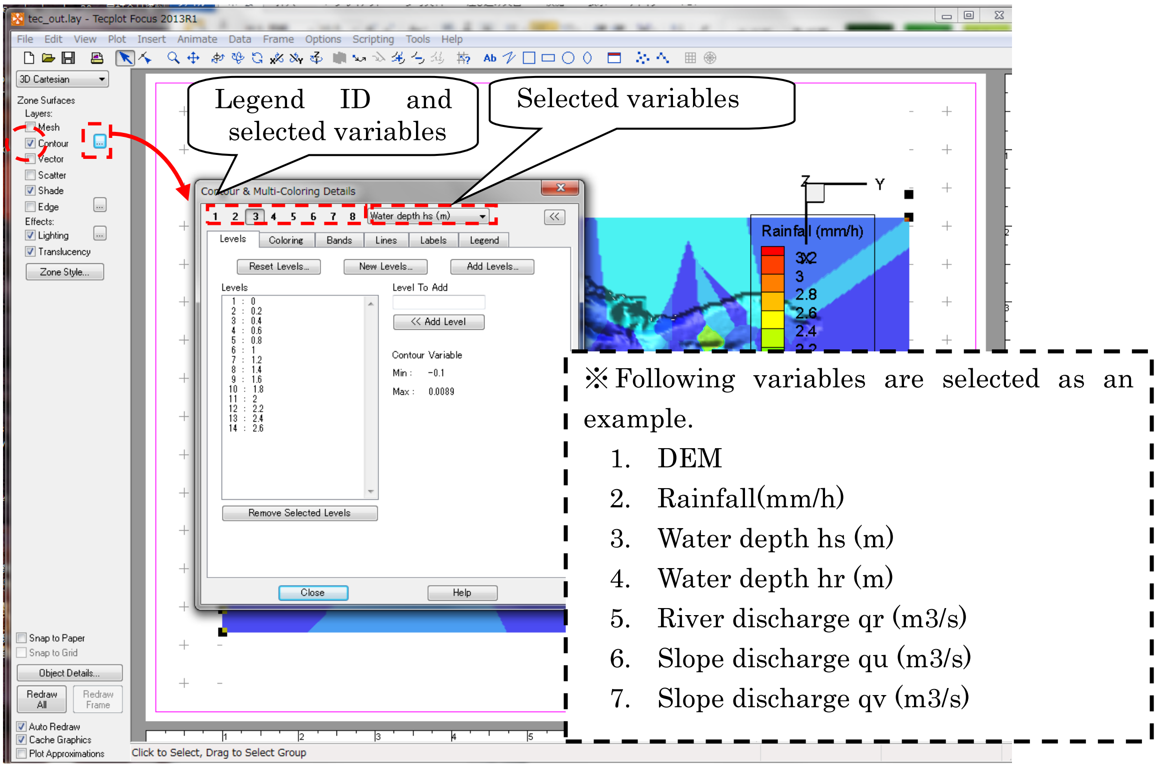
⑤ Edit translucency of shade

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

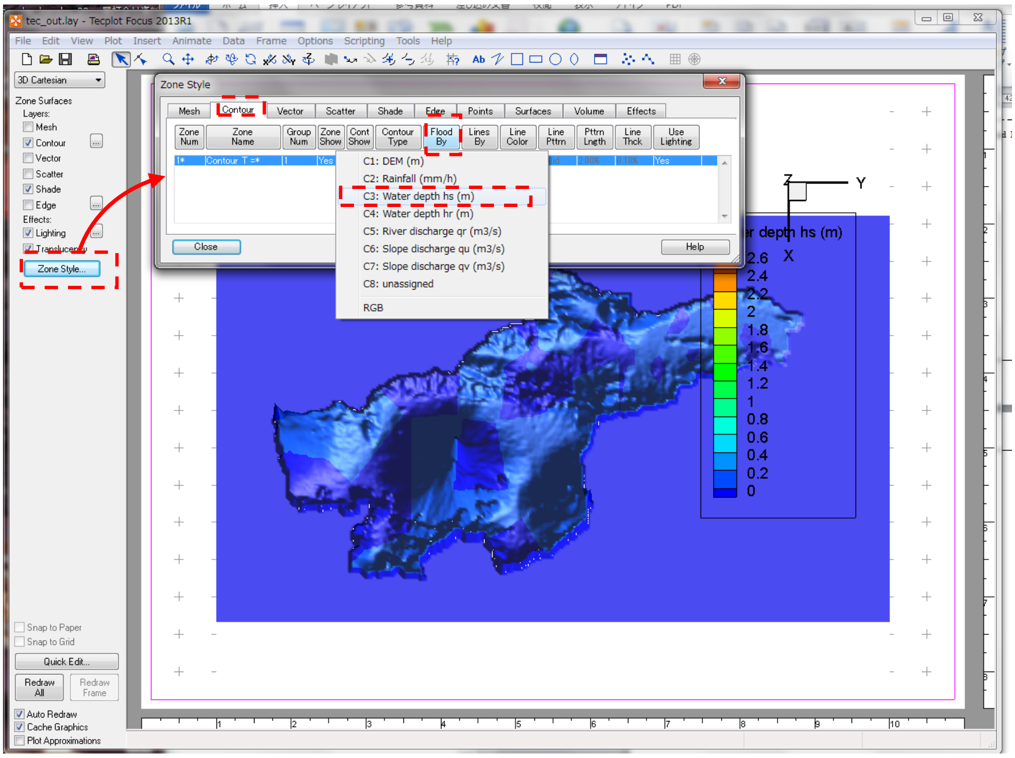


7.6.4 Draw contour figure on Tecplot

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.

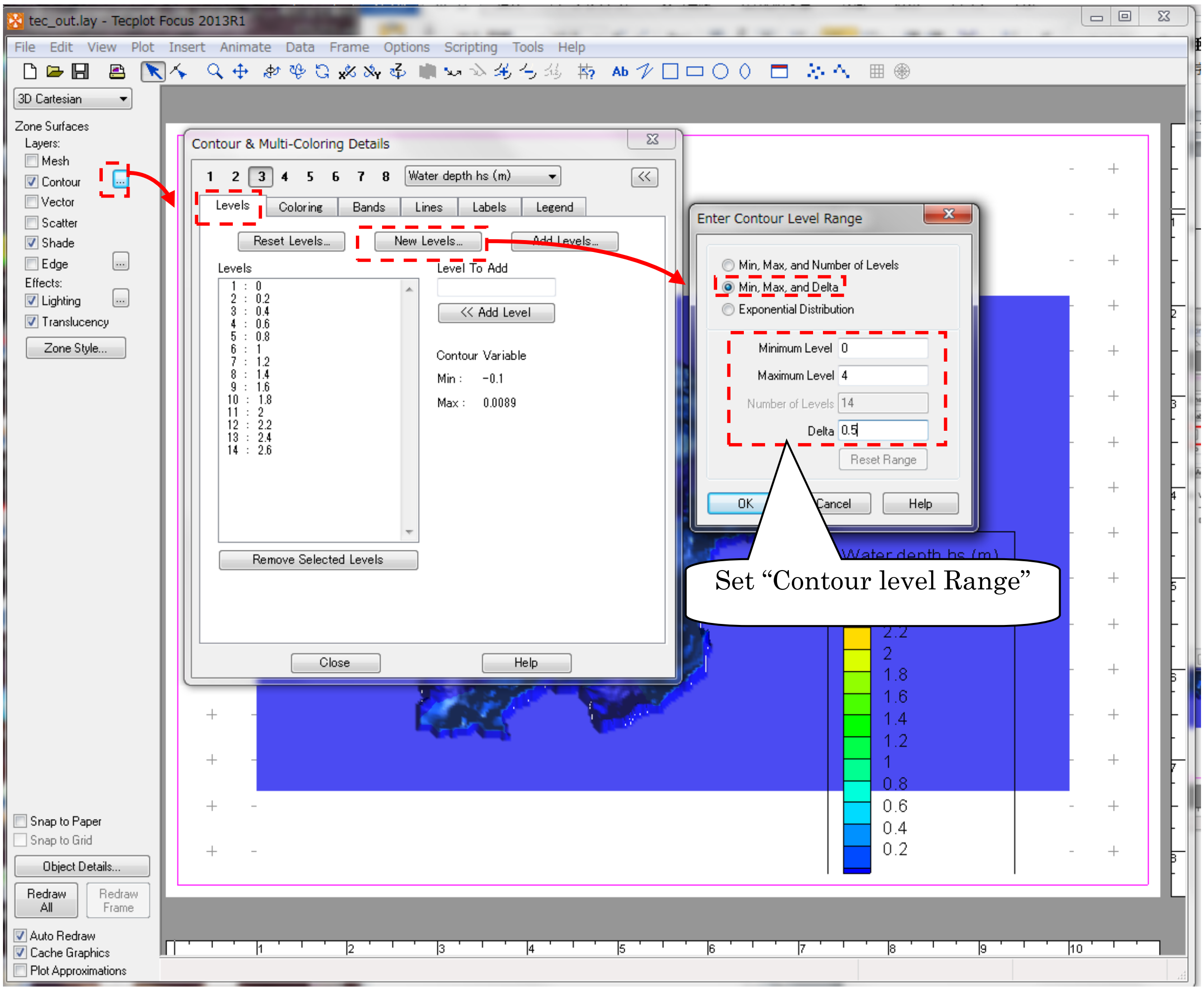


② Select variable to display. User can select variable from variables identified in ①. Click "Zone Style" and edit "Flood By" on "Contour" tab to edit target variable and its legend. "Water depth hs (m)" is selected in the following figure as an example.

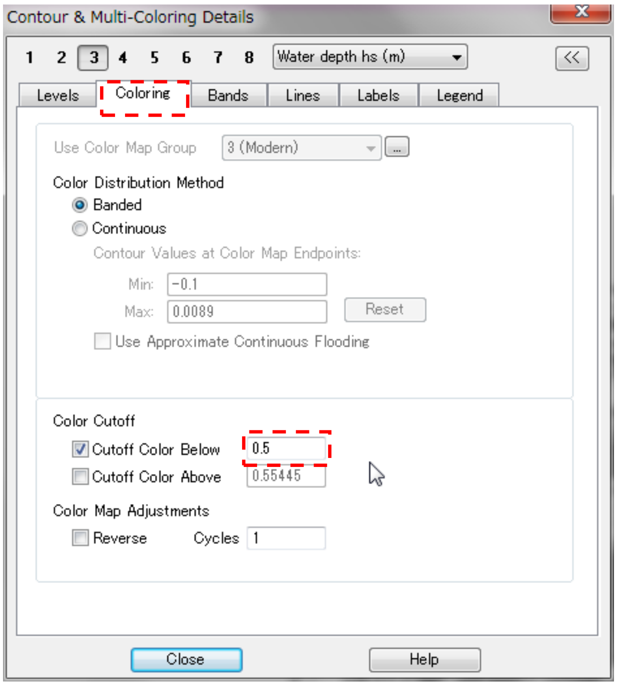


③ Edit legend.

User can edit color legend of contour as follows;

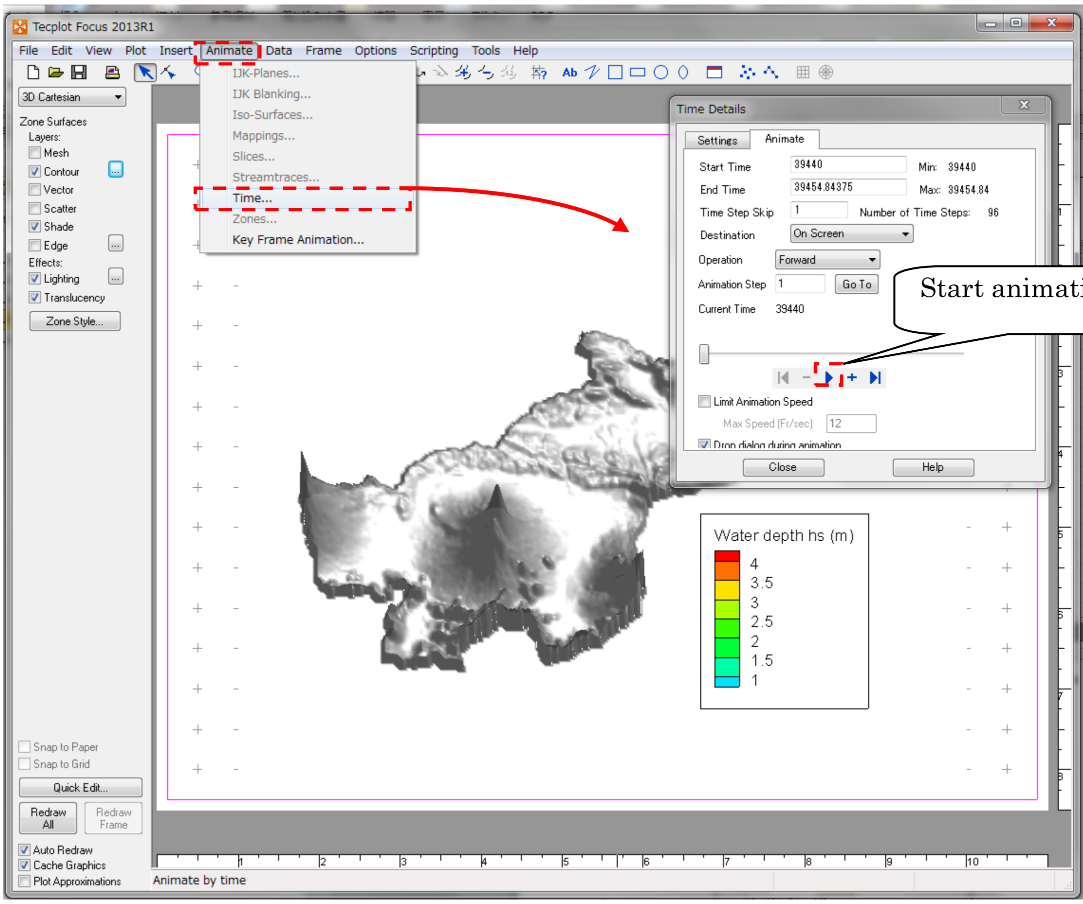


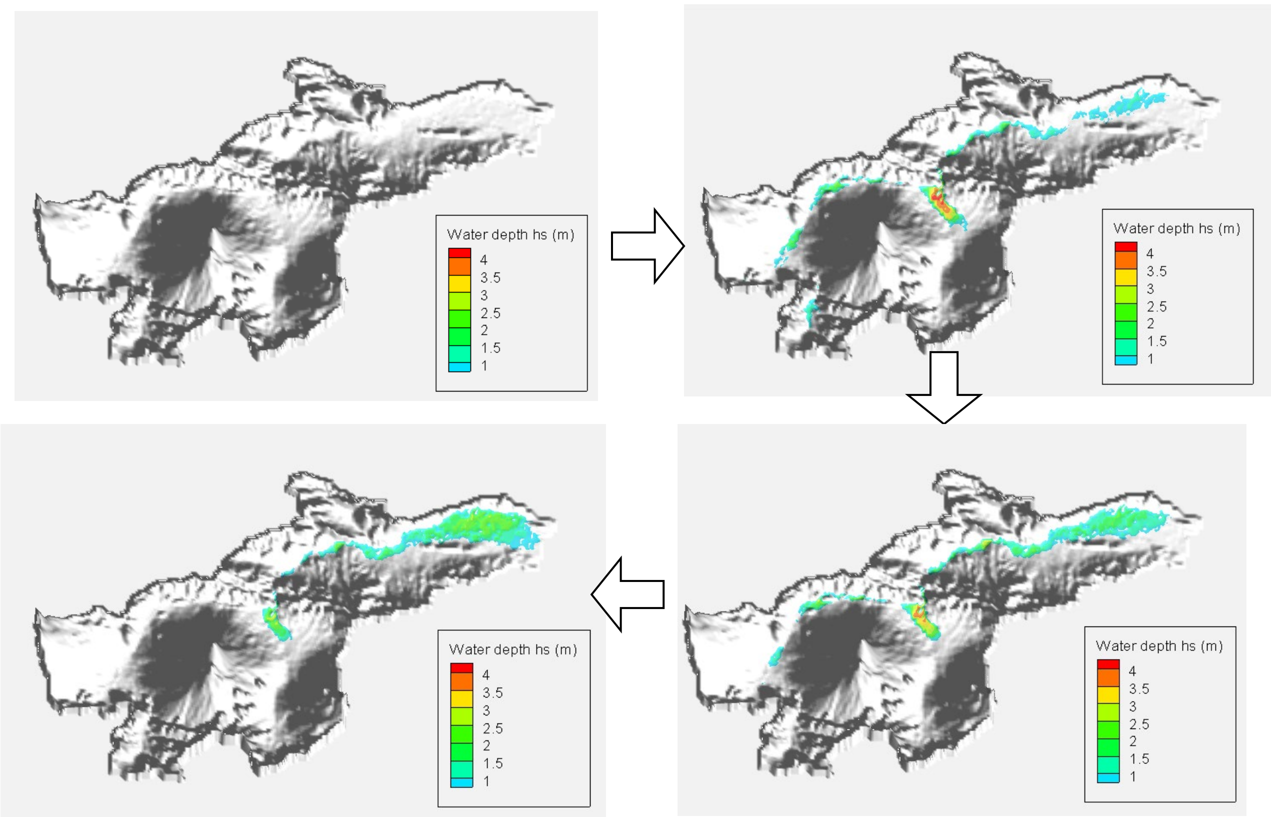
④ User also can edit "cut off" to display upper and lower color limits. Color up to 0.5m is cut in the following figure as an example.



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

[Animate] > [Time…]



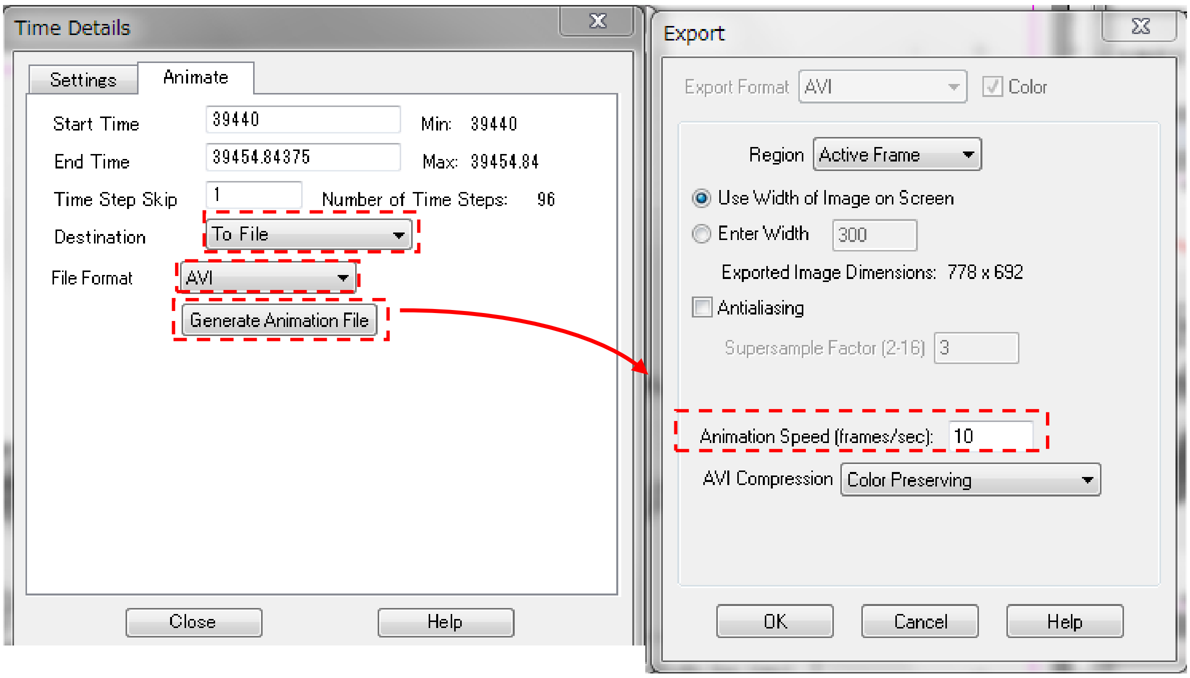


⑥ Export animation file

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

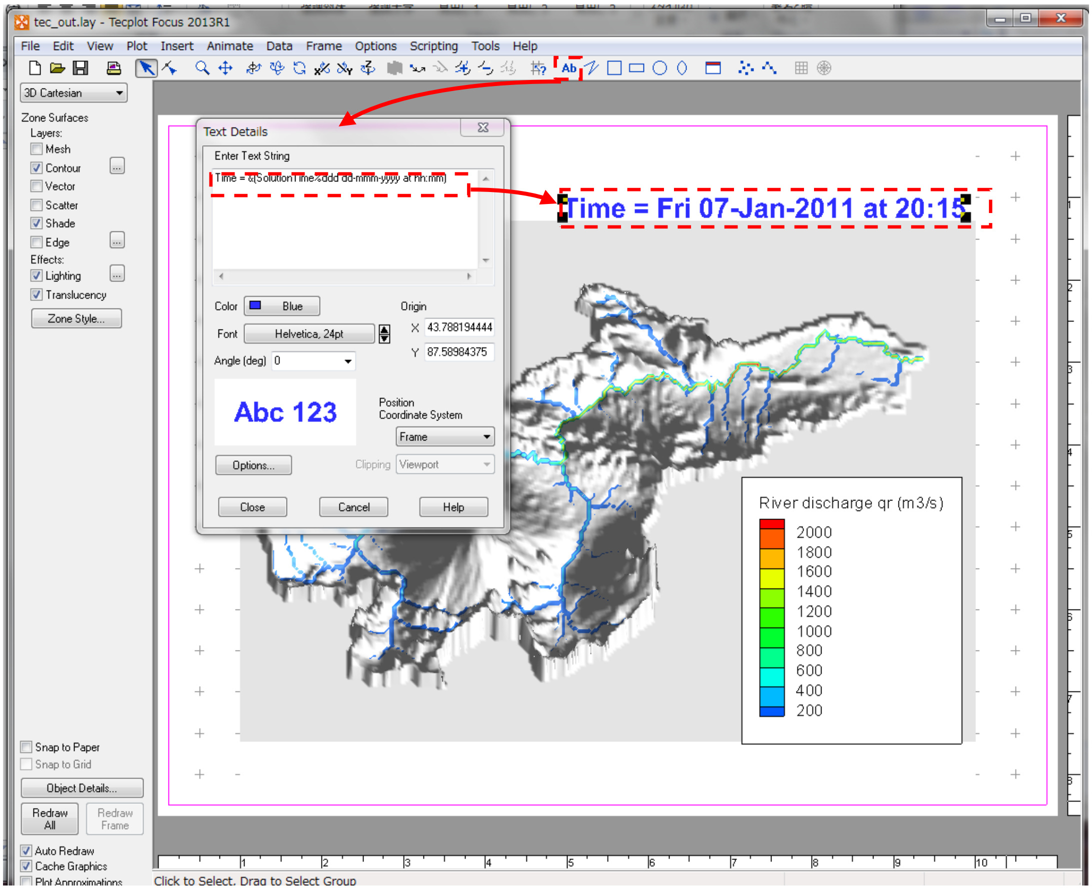
① Display timestamp information on animation

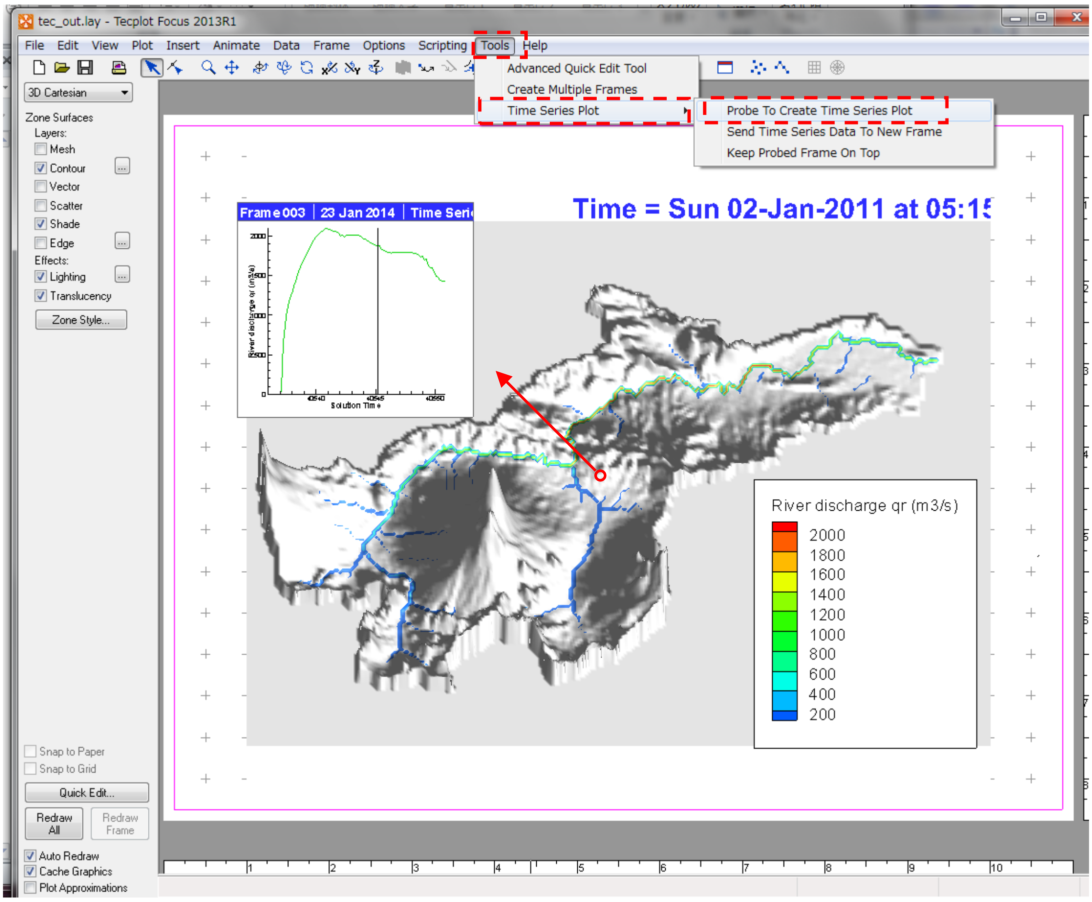
If user needs to display timestamp information on the contour figure, add textbox and input as follows;

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

② Display time series graph on plane view.

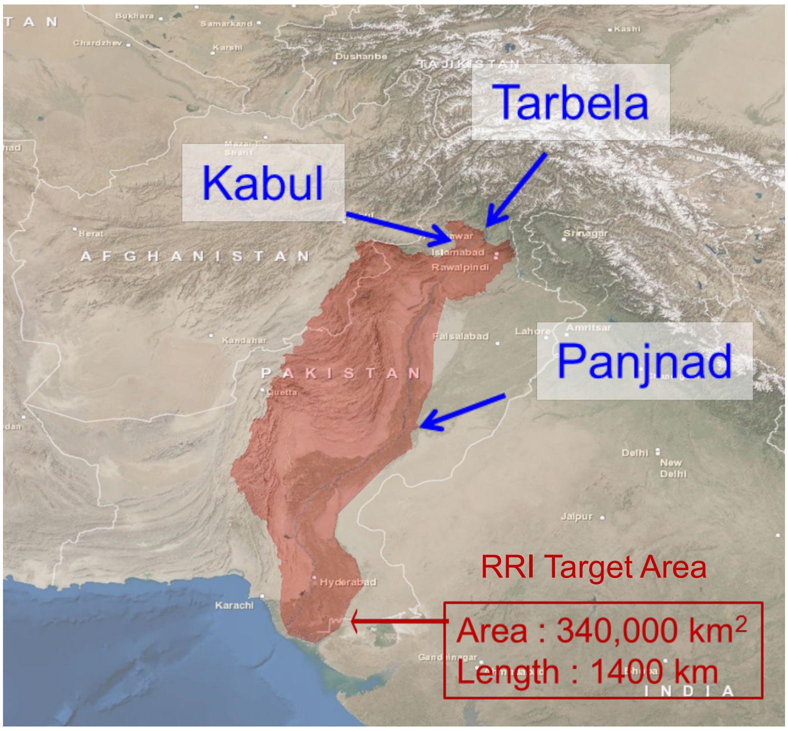
Select [Tools] > [Time Series Plot] > [Probe To Create Time Series Plot] and identify the position by left click with the pointer “+” to display the time-series. Note that the variable selected as “Flood By” will be shown on the time series graph. Hence user needs to change the setting of Zone Style 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
3D Cartesian
Zone Surfaces
Layers:
Mesh
Contour
Vector
Scatter
Shade
Edge
Effects:
Lighting
Translucency
Zone Style...
Text Details
Enter Text String
Time = &(SolutionTime%ddd dd-mmm-yyyy at hh:mm)
Color Blue
Font Helvetica, 24pt
Angle (deg) 0
Origin
X 43.788194444
Y 87.58984375
Abc 123
Position Coordinate System
Frame
Options...
Clipping Viewport
Close
Cancel
Help
Time = Fri 07-Jan-2011 at 20:15
River discharge qr (m3/s)
2000
1800
1600
1400
1200
1000
800
600
400
200
Snap to Paper
Snap to Grid
Object Details...
Redraw All
Redraw Frame
Auto Redraw
Cache Graphics
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
3D Cartesian
Zone Surfaces
Layers:
Mesh
Contour
Vector
Scatter
Shade
Edge
Effects:
Lighting
Translucency
Zone Style...
Frame003 | 23 Jan 2014 | Time Seri
Solution Time
Time = Sun 02-Jan-2011 at 05:15
River discharge qr (m3/s)
2000
1800
1600
1400
1200
1000
800
600
400
200
Snap to Paper
Snap to Grid
Quick Edit...
Redraw All
Redraw Frame
Auto Redraw
Cache Graphics
Plot Approximations

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.

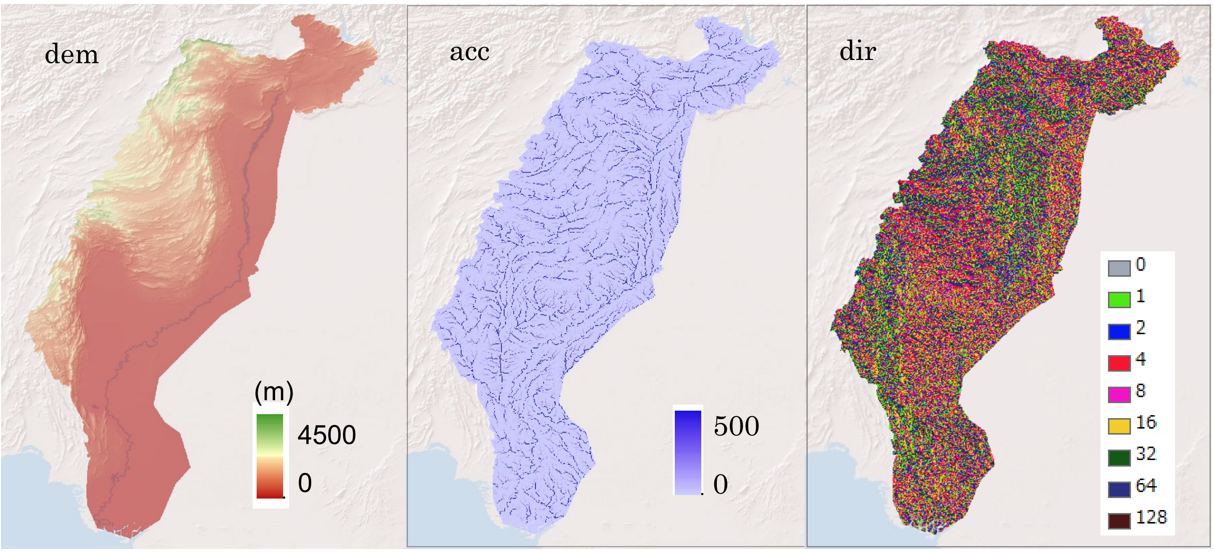


A polygon covering the simulation target (the red mask in the above figure) was prepared first. The flow direction data in HydroSHEDS (30sec) was used to identify the entire Indus River basin. Then the upstream areas above Tarbela, Kabul and Panjnad were removed from the entire Indus River basin.

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.

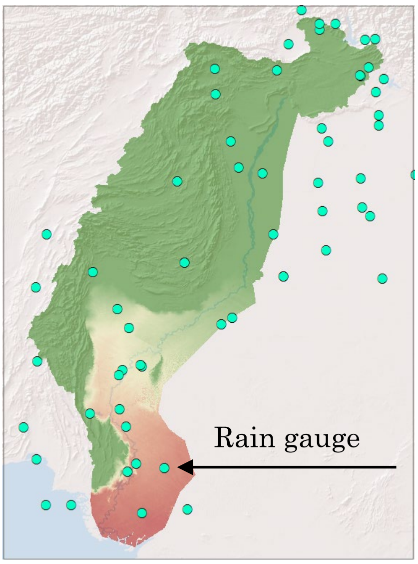
8.1 On Input Topography

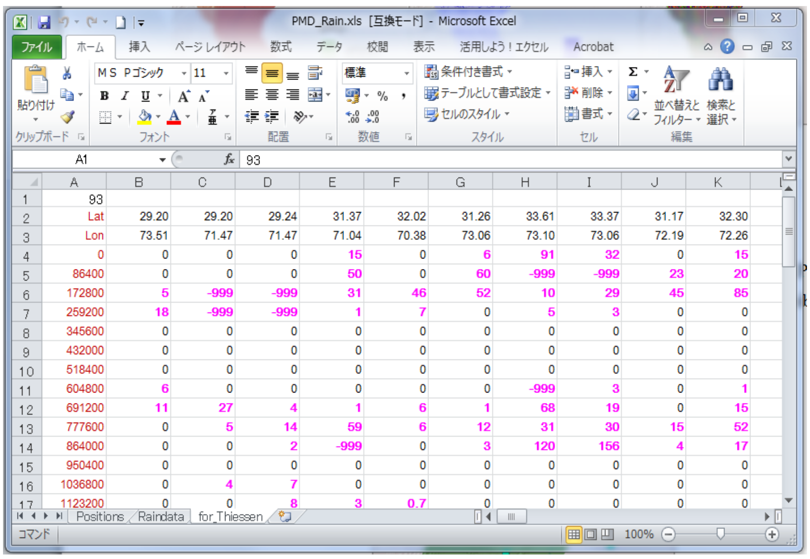
By using the catchment polygon, dem, acc and dir datasets were clipped for the catchment area. The function embedded in ArcGIS ([Spatial Analyst Tools] → [Conditional] → [Con]) was used to mask the target area out of the regional datasets of HydroSHEDS (30 second resolution). Then “demAdjust2” program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

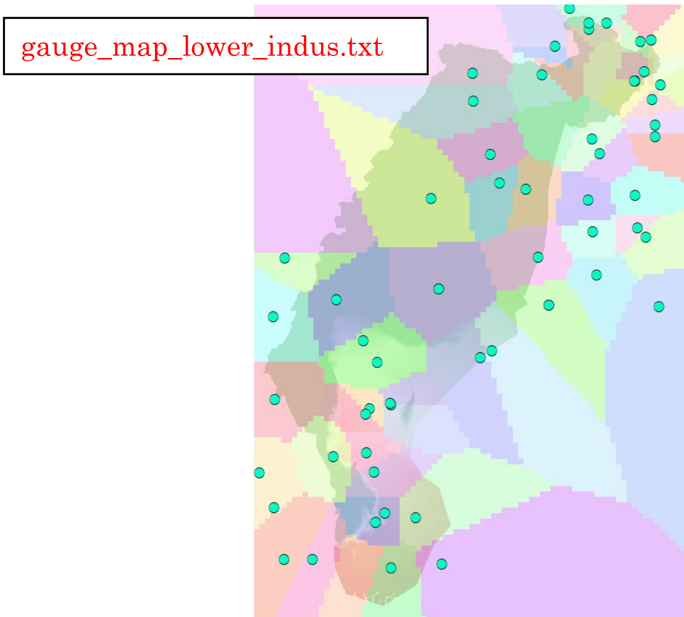




	A	B	C	D	E	F	G	H	I	J	K
1	93										
2	Lat	29.20	29.20	29.24	31.37	32.02	31.26	33.61	33.37	31.17	32.30
3	Lon	73.51	71.47	71.47	71.04	70.38	73.06	73.10	73.06	72.19	72.26
4	0	0	0	0	15	0	6	91	32	0	15
5	86400	0	0	0	50	0	60	-999	-999	23	20
6	172800	5	-999	-999	31	46	52	10	29	45	85
7	259200	18	-999	-999	1	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).

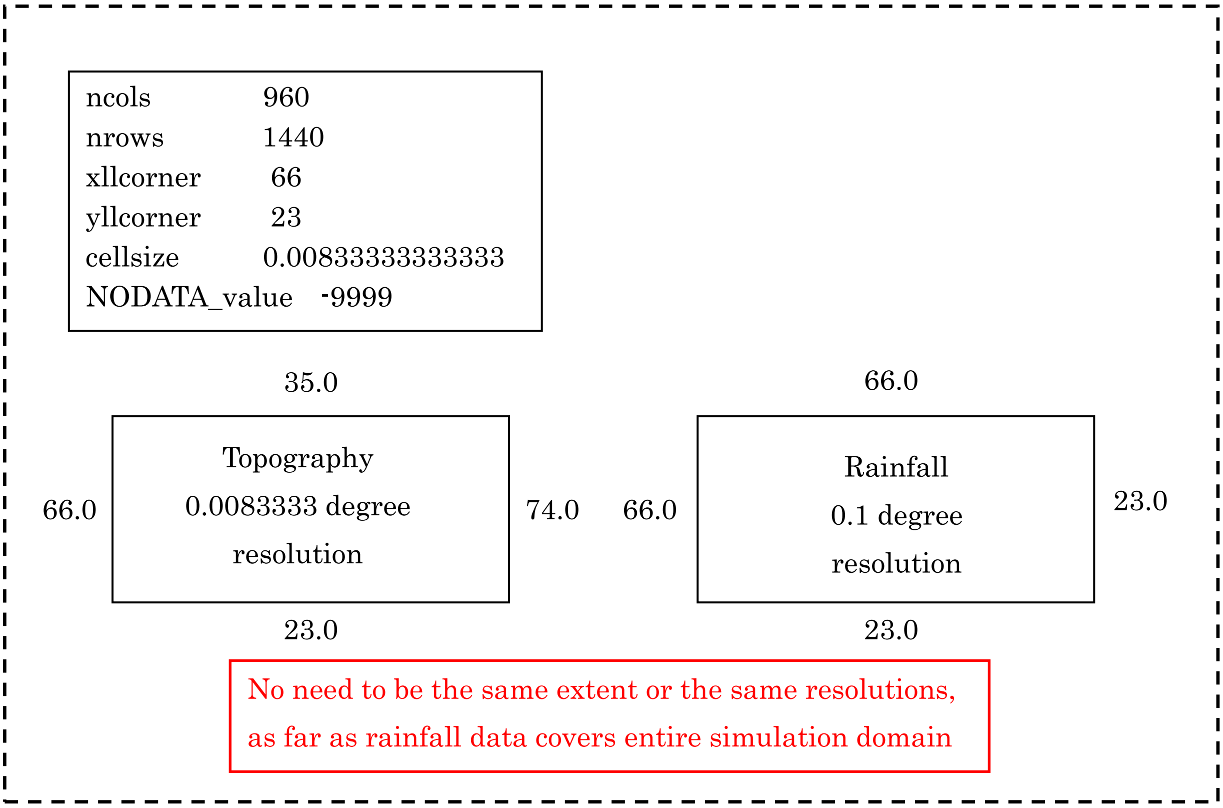


Here is the sample of the rainThiessen program input file (rainThiessen.txt).

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
```

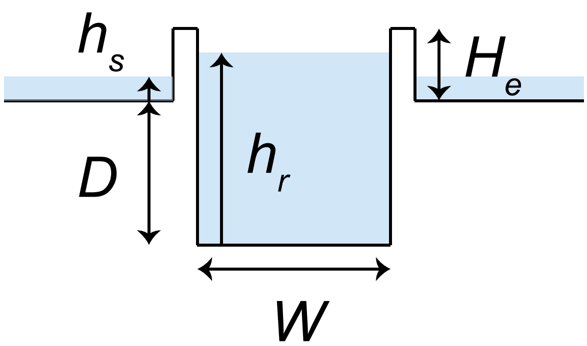
RRI_Input.txt

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

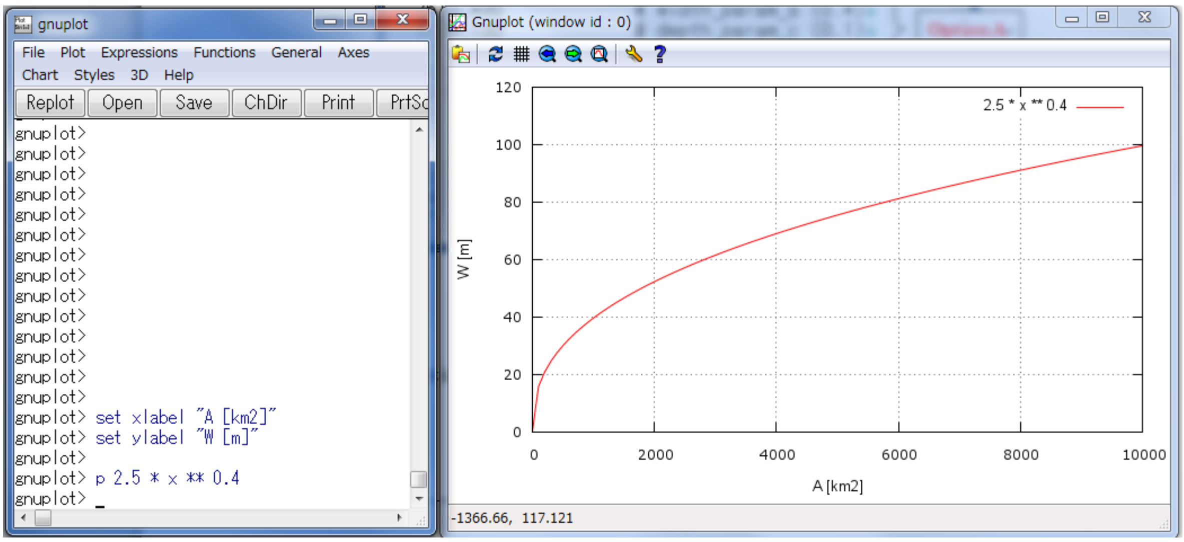
L38	100	# riv_thresh	RRI_Input.txt
L39	2.5d0	# width_param_c	Option A
L40	0.4d0	# width_param_s	Option A
L41	0.1d0	# depth_param_c	Option A
L42	0.4d0	# depth_param_s	Option A
L43	0.d0	# height_param	Option A
L44	20	# height_limit_param	Option A
L45			
L46	1 ← 0 : Option A / 1 : Option B (Read from files)		
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			

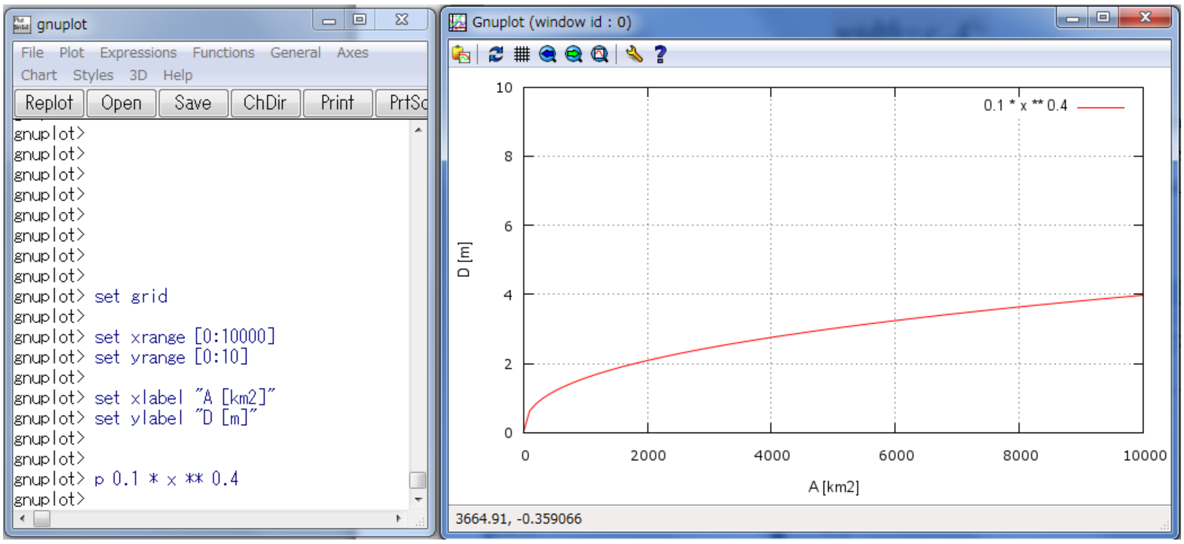
<u>A) For the first option,</u> the parameters of the following empirical equations must be appropriately set to represent target catchment condition (L38 – L44 of RRI_Input.txt).

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

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

where A in the equations is the upstream catchment area [km^2] 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.





<u>B) For the second option,</u> 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.

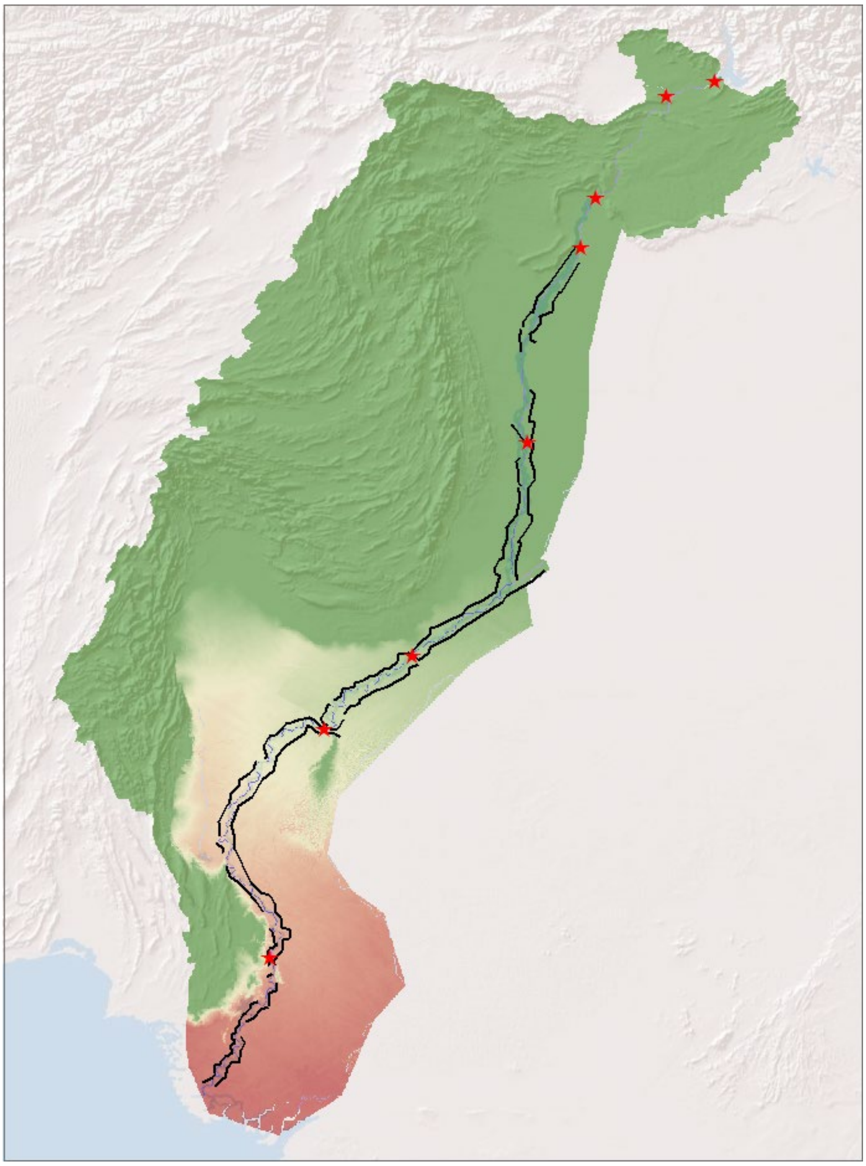
8.5 On Embankment Setting

There are two kinds of embankment settings in RRI simulation.

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

<u>A) The first type of embankment</u> 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.

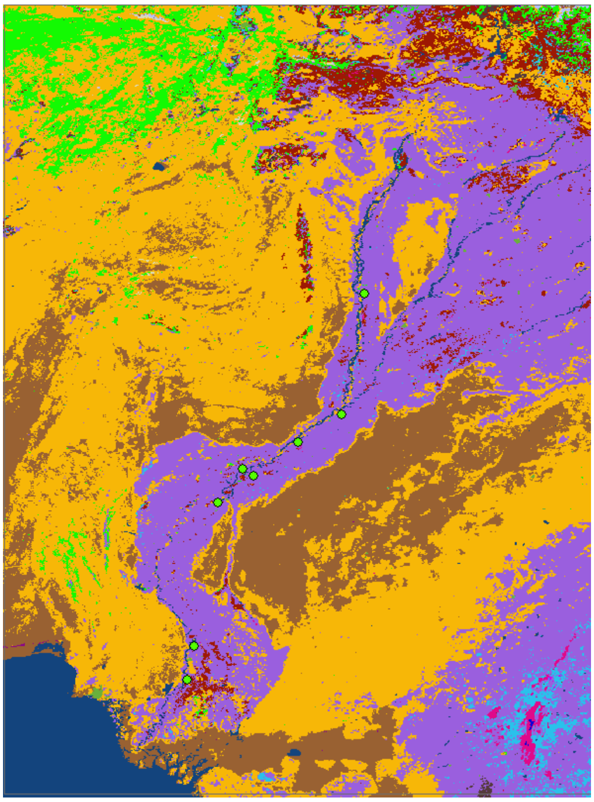
Note that even if a user intends to set a continuous embankment apart from a main river, if a tributary joins into the river and if the "*height*" value is set on a river grid cell where *width* >

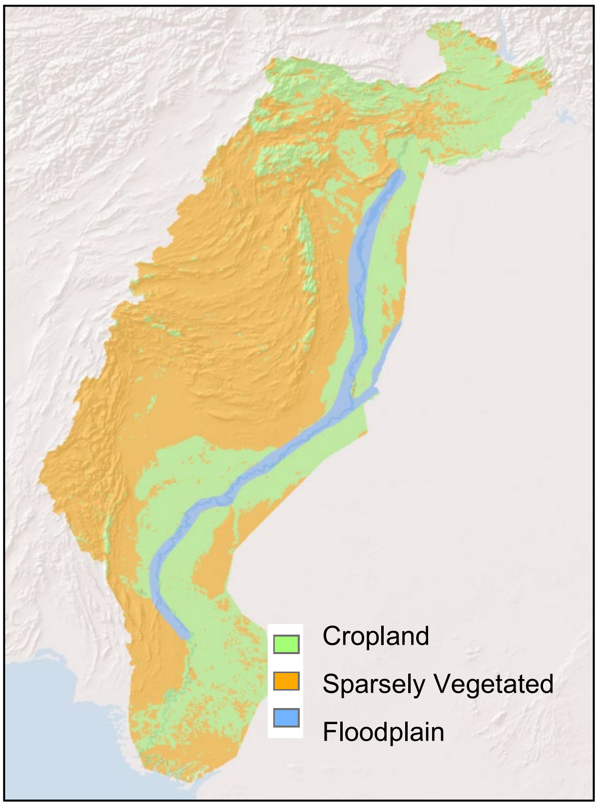
0, the embankment would be regarded as the embankment of Type A. As a result, the set embankment will be discontinuous at the location.

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

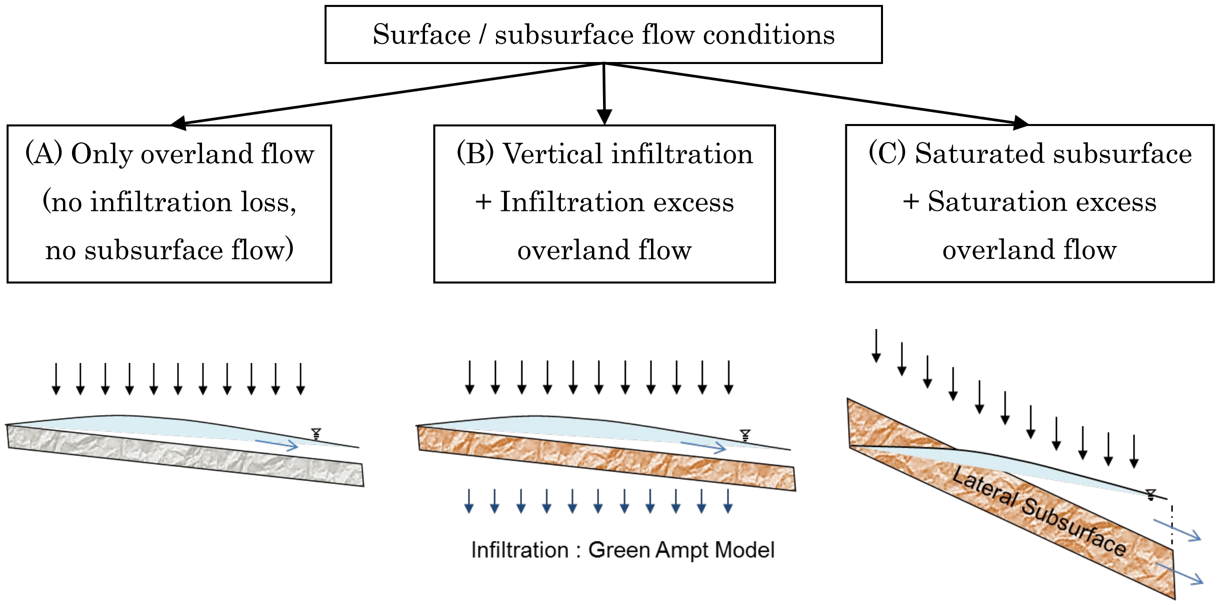
USGS Land Use/Land Cover System Legend (Modified Level 2)

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 (-)	gammaa		0.471d0 (0.3 ~ 0.5)	
k_v (m/s)	kv	0.d0	5.56d-7	0.d0
S_f	Sf	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 kv 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 kv = 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 "ksg = 0.d0" to avoid bedrock groundwater computation, whose initial condition must be carefully set (only for experienced users).
** If both ka and kv 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          # kv
L26 | 0.273d0 0.1101d0 0.d0           # Sf
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                  # ksg
L33 - L36 are inactive under ksg = 0.d0
```

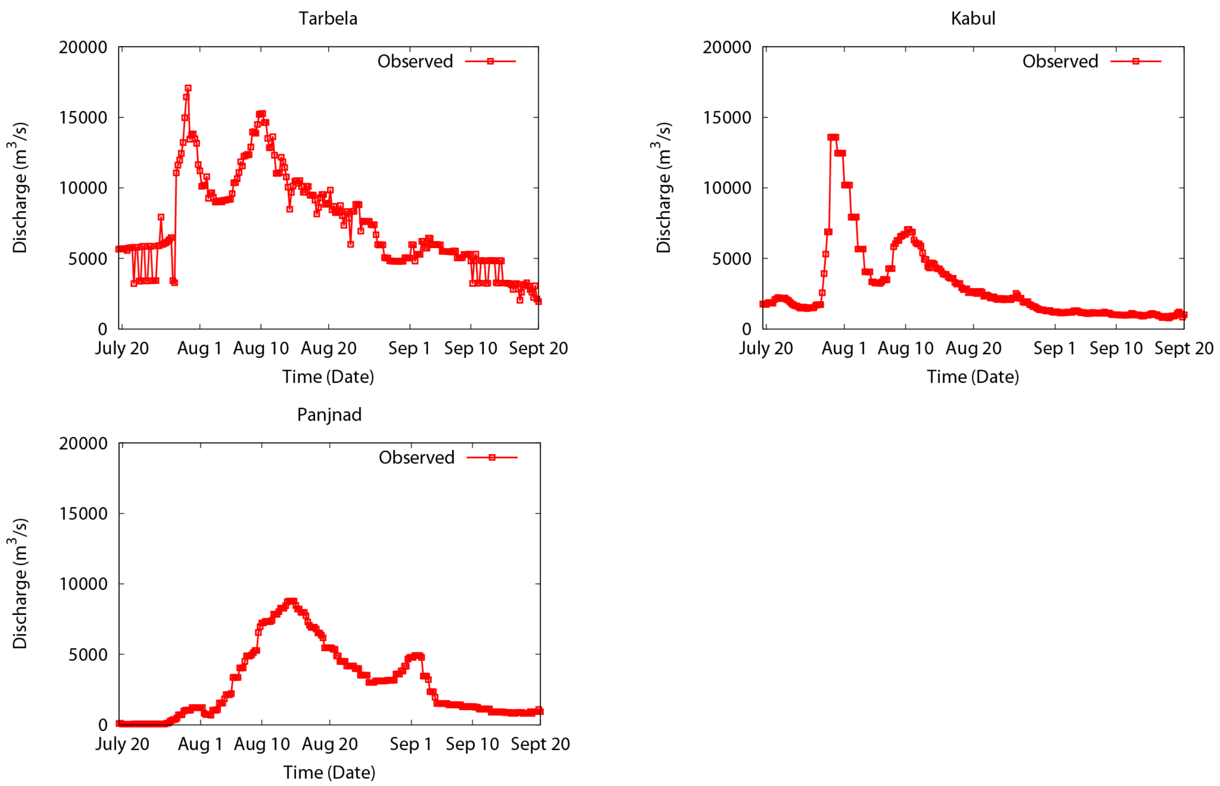
Reference Table : Green-Ampt Infiltration Parameters for different soil texture

Soil texture class	k_v (m/s)	ϕ [gammaa]	S_f (m)
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.

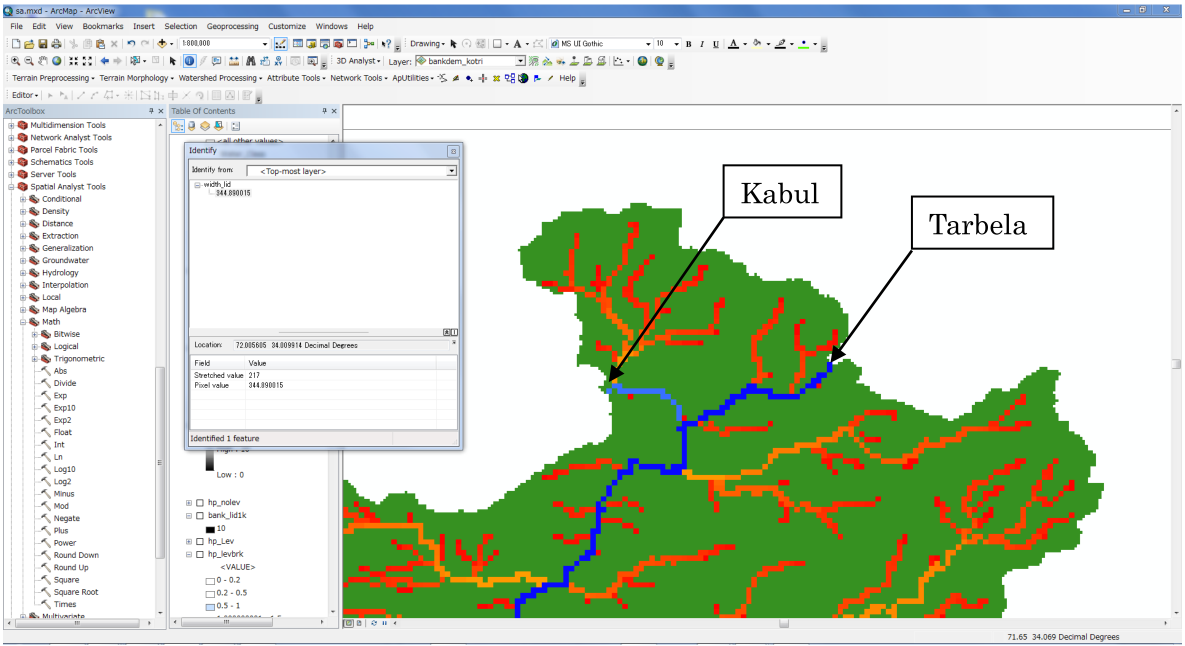


The steps to set river discharge boundary conditions are described below.

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: [m^3/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.

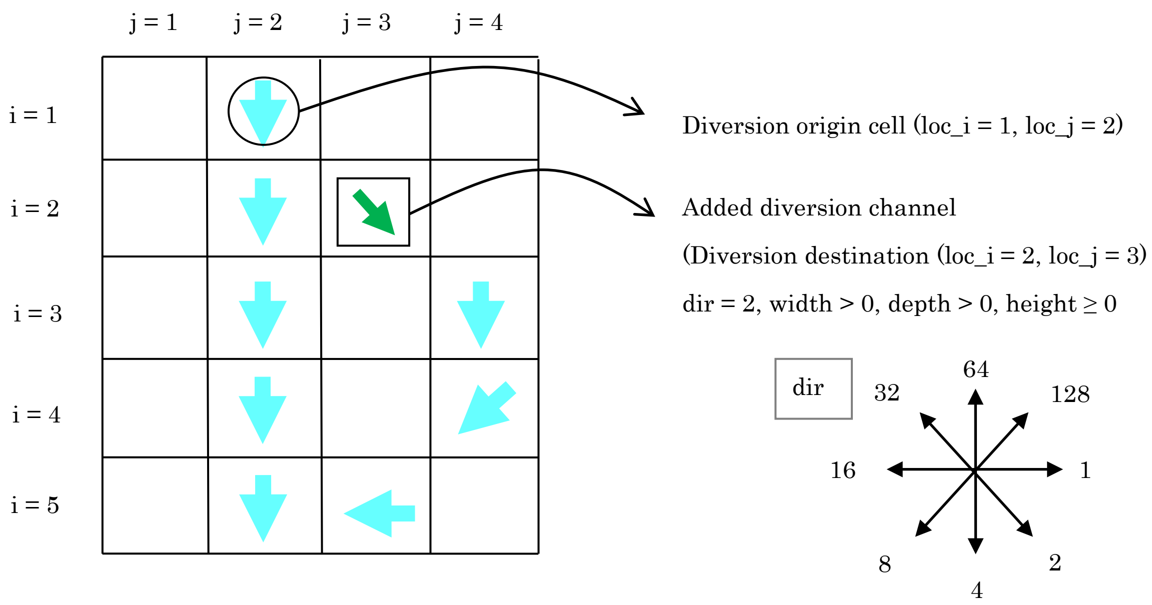
This feature enables the continuous long-term simulation. In order to read the initial conditions, L49 to L52 in the RRI_Input.txt must be edited in a same manner as the example of the boundary condition setting.

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_rate” specifies the ratio of discharge diverted from the main river to the diversion..

```
1   2   2   3   0.2
```

div_sample.txt

loc_i_org	loc_j_org	loc_i_dest	loc_j_dest	div_rate

④ Activate this option by setting flag 1 on L70 and specify the diversion file name (e.g. div_sample.txt) 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.

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

dam_sample.txt

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.

8.12 Arbitrary cross sections (for advanced users)

In addition to the rectangular river cross sections, the RRI Model can incorporate also arbitrary cross section directly. The followings explain the procedure to reflect cross section information to the model.

① Use "section.f90" program in the "etc" folder to prepare cross section files (e.g. sec_000001.txt) based on cross section coordinate information files (e.g. CS99C2.txt).
② Prepare a map file (e.g. sec_map.txt) to define which river grid cells should reflect which cross section files (e.g. sec_000001.txt).
③ Change RRI_Input.txt setting to read sec_map.txt and the cross section files.
④ (Optional) prepare and read river length file from the RRI_Input.txt

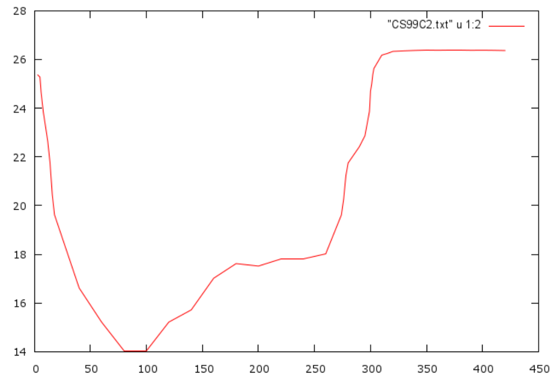
The followings explain the detailed step to reflect the cross section information.

① The following steps use "RRI-CUI/etc/section/section.f90" program. Suppose cross section coordinate information files are available in the following format.

```
3.000000    25.35900
5.000000    25.27600
6.000000    24.65500
8.000000    23.83900
…           …
```

CS99C2.txt

(First column: x, second column: z)

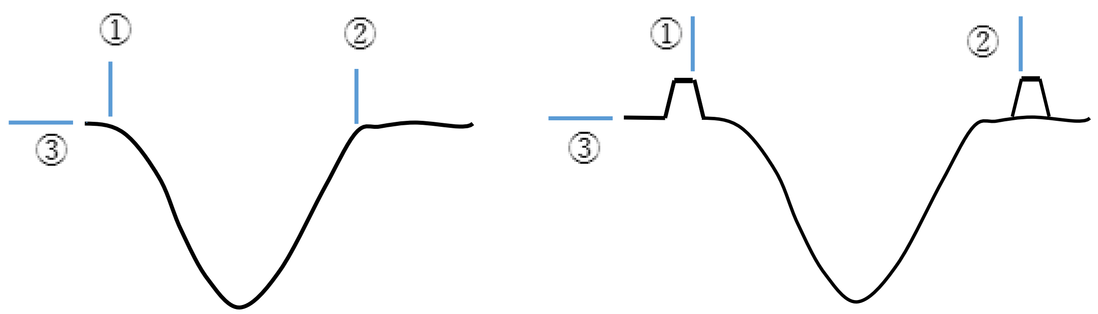


Prepare “section.txt” file accordingly to be inputted to “section.f90” program.

section.tx

```
CS99C2.txt
sec_000001.txt
0.03   ! Manning's roughness in river
100     ! div
25.     ! datum level
1        ! startx
30      ! endx
```

where the first two lines are input and output file names and the third line sets the Mannings’s roughness in the river. The fourth line defines “div”, which determines the number of divisions in depth directions, with 100 divisions in the above example. The fifth line is a datum level (m), which is the elevation of floodplain level identified from the cross section information (e.g. CS99C2.txt) as shown in the figure below. The RRI Model sets the defined cross section in a way that this datum level will be equal to the DEM elevation at each grid-cell.



For example, suppose the DEM of a grid-cell to set a river cross section is 24 m, while the estimated flood plain level read by the cross section file (i.e. elevation of ③ in the above figure) is 25 m. In this case, a user has the following two options.

a) Regardless the elevation information contained in the cross section data, prioritize the elevation of DEM 24 m and the floodplain level in the cross section will be at 24 m. For this option, please set the above "datum level" as 25 m. This means that the 25 m in the cross section will be regarded as the datum and this level will be the same as the DEM.
b) If both absolute elevation information contained in the DEM and cross section files are reliable and they are generally consistent each other, the user may set "datum level" as 24 m to use the elevation information contained in the cross section file directly. This means that the level of 24 m in the cross section will be equal to the level in DEM.

For L6 (startx) and L7 (endx) in the "section.txt", please set minimum and maximum x coordinates, which should be within a river grid-cell, as shown in the above figure ① and ②.

Based on the above setting, "section.f90" will automatically proceed the following steps.
STEP 1: The program measures depth (d) and levee height (h). The depth (d) is the elevation difference between the deepest position in the cross section and the level of ③. The height (h) is the smaller elevation either ① or ② measured from ③.
STEP 2: It divides a cross section into the number of "div" (i.e. 100 in this case) from the deepest point to the highest elevation between ① and ②.
STEP 3: It outputs perimeters, widths and the Manning's roughness parameters corresponding to each depth.

An example of output files by section.f90 (e.g. sec_000001.txt)

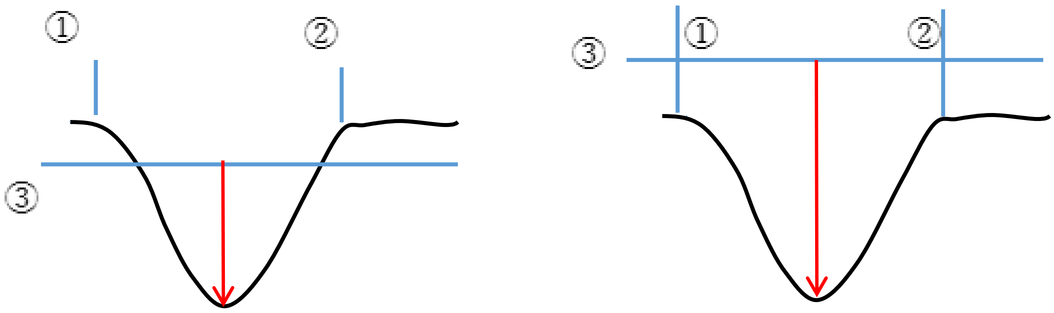
```
div         d                h                    sec_000001.txt

100         10.98000         0.00000
0.11339         23.78646         23.77967         0.03000
0.22678         27.57293         27.55933         0.03000
0.34017         31.35939         31.33900         0.03000
0.45356         35.14586         35.11867         0.03000
…
11.11222        298.35455        295.76846        0.03000
11.22561        298.56949        295.95105        0.03000
11.33900        300.62881        298.00000        0.03000

depth           perimeter        width            roughness
```

* section.f90 reads the Manning's roughness as a single parameter and set the constant value in the output file. However as shown in the above figure, the file can set different composite Manning's roughness values depending on water depths. One may change the roughness values if necessary.

② Prepare a section map file (e.g. sec_map.txt") to define the position of grid-cells to set cross section files. The "sec_map.txt" should have the same format (and size) as the dem.txt and contains numbers (greater than 1) at grid cells, whose numbers corresponding to cross section files. For example, if "sec_map.txt" indicates "5" at a particular grid cell, the RRI Model will find "sec_000005.txt" and reads and assigns the cross section information.

③ By setting 1 in L83 of RRI_Input.txt, the model will read "./riv/sec_map.txt" and corresponding river cross section files (e.g. sec_******.txt).

④ If necessary, a user may set river length at each grid-cell defined by "length.txt", whose format and the matrix size must be the same as "dem.txt". In that case, set 1 in the L80 and reads "./riv/length.txt" from the RRI_Input.txt. This can be activated when rivers are meandering or one would like to reflect distant information contained in cross sections.



(Reference) The method of RRI calculation depending on the datum level ③.

In case of left panel in the upper figure, the model will start exchange water by overtopping from the river to the slope cells once the river water exceeds the lower level of ① or ②.

In case of right panel in the upper figure, the model does not exchange water between the river and slope cells unless the river water exceeds the level of the datum level ③. As a result, the modeled river sections become larger than the actual one indicated by the figure.

9. Use of RRI Graphical User Interface (GUI)

This section explains how to use RRI-GUI to apply the model at a basin and visualize the simulation results.

9.1 Pre-setting

1) Unzip “RRI_1_4_2.zip” and save it under a working folder (e.g. C:/”).

3) Check your PC is 32 or 64 bit. (My Computer → Property)

4) Install two programs saved in “RRI-GUI/Pre-setting”

① w_fcompxe_redist_intel64.msi

② vcredist_x64.exe

(for 32 bit, install vcredist_x86.exe and w_fcompxe_redist_ia32.msi)

For “vcredist_x64.exe”, you may encounter an error message suggesting you have already the newer version of “Microsoft Visual C++ 2010 Redistributable”. In that case, you can just close the error message and cancel to install “vcredist_x64.exe”.

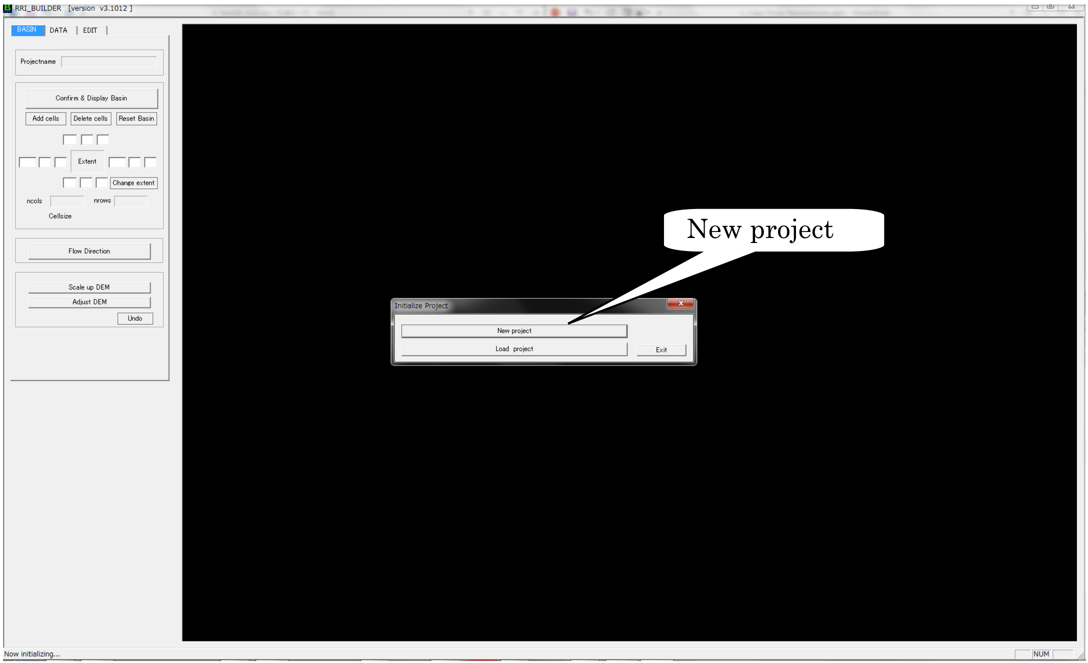
5) Execute RRI_BUILDER_64.exe

(for 32 bit machine, execute RRI_BUILDER_32.exe)

9.2 Model application and running with RRI_BUILDER

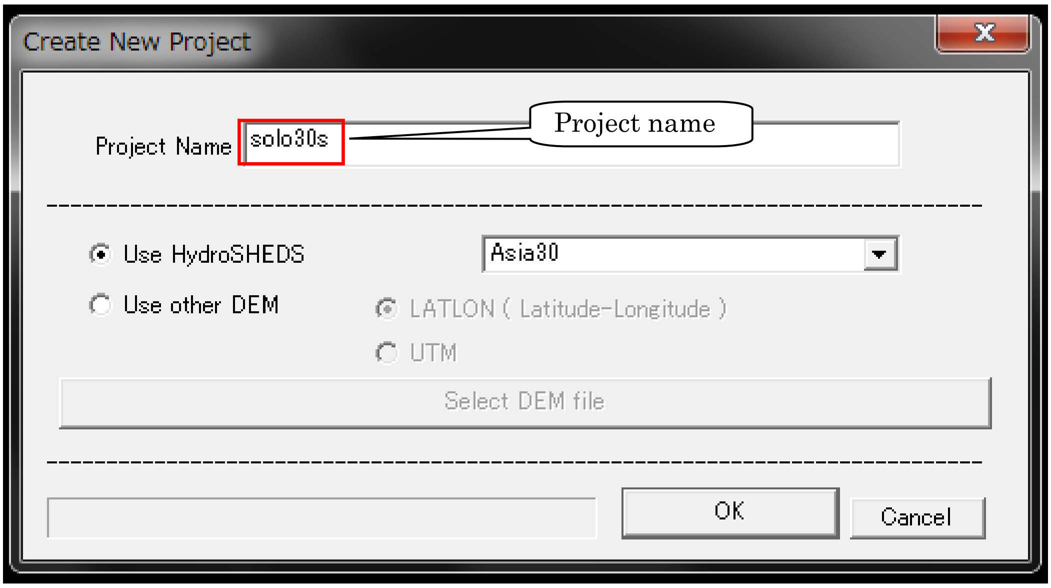
9.2.1 Preparing Input Topography Data

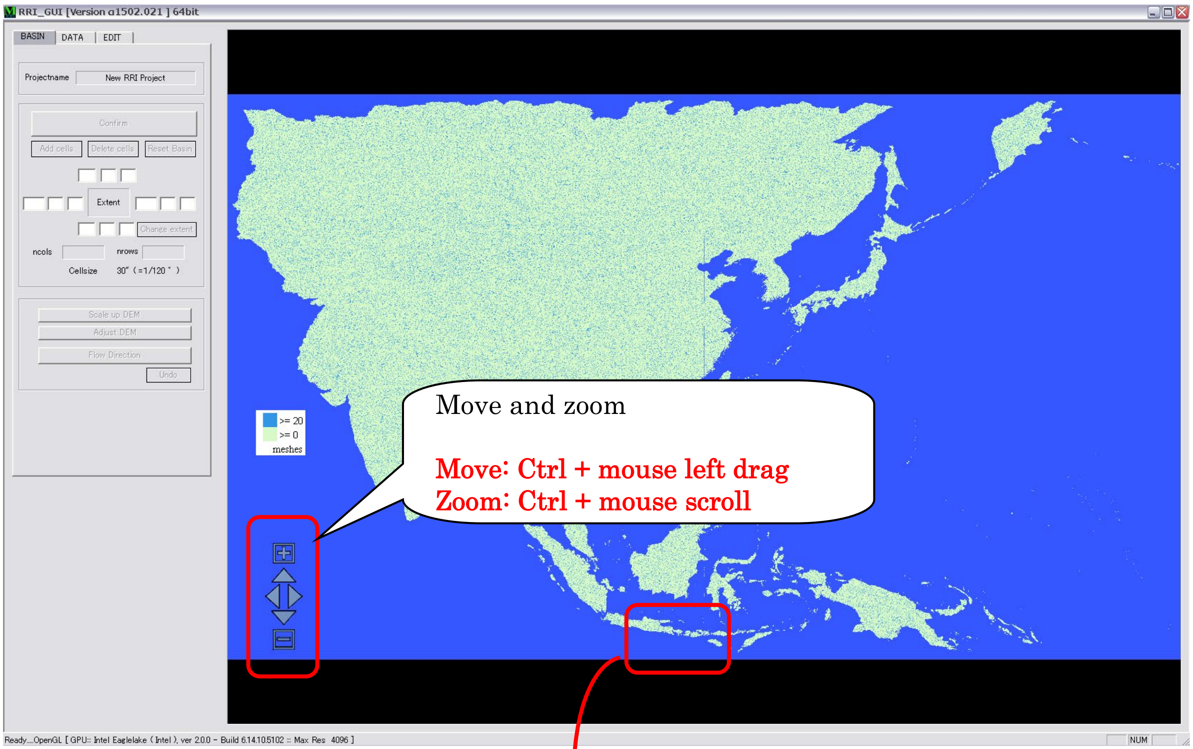
The first screen of the "RRI_BUILDER_64.exe" is to choose "New Project" or "Load Project".



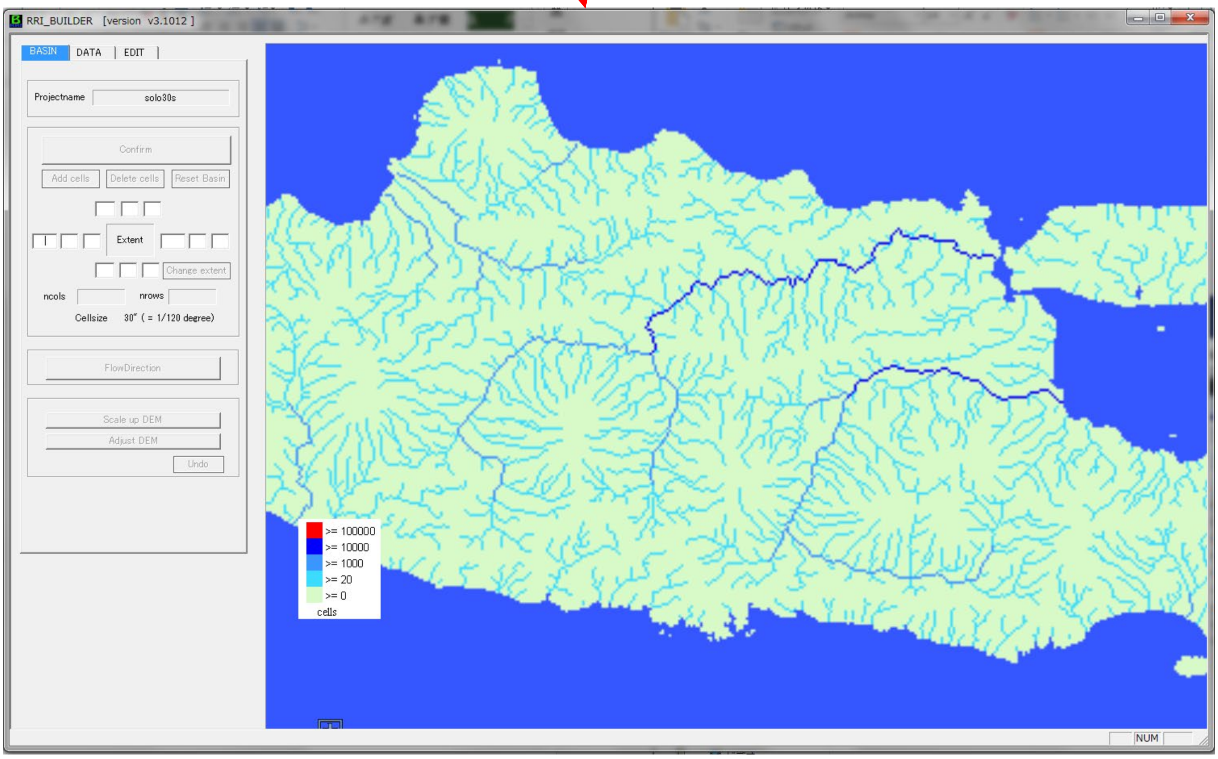
Select "New Project" in this exercise.

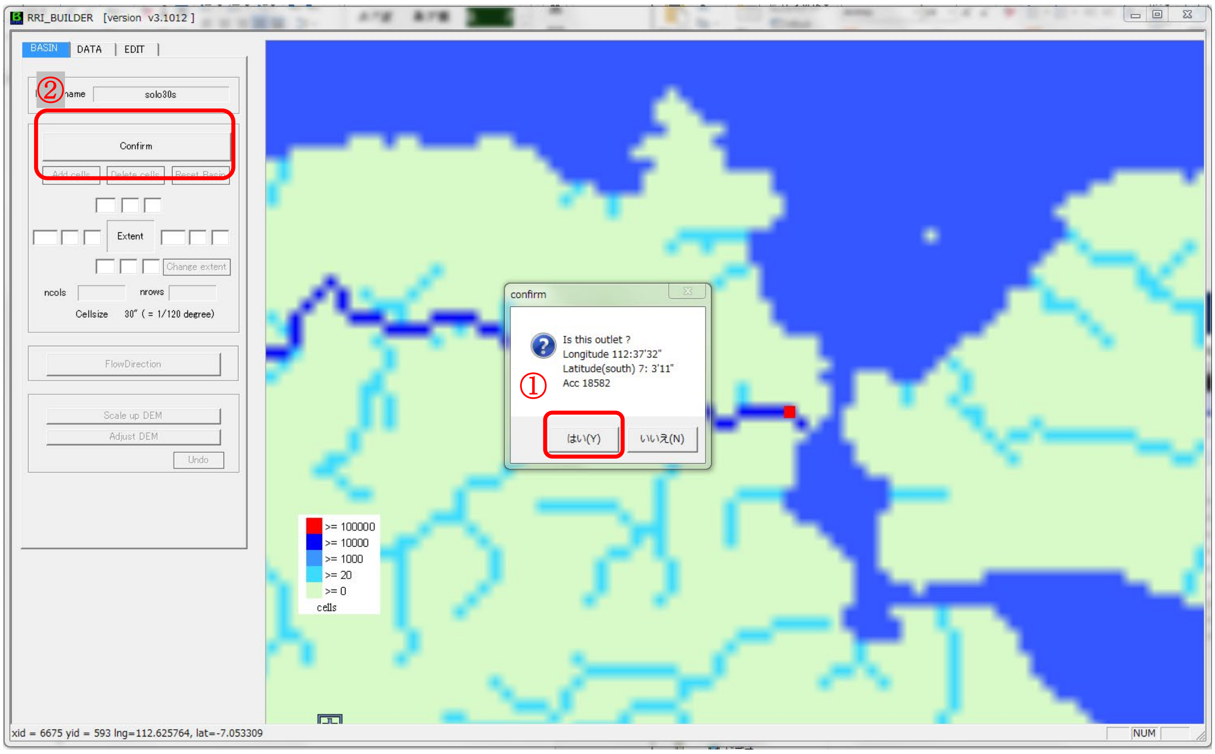
Type in a new project name (e.g. "solo30s") with the selections of "Use HydroSHEDS" and "Asia30", then click "OK".





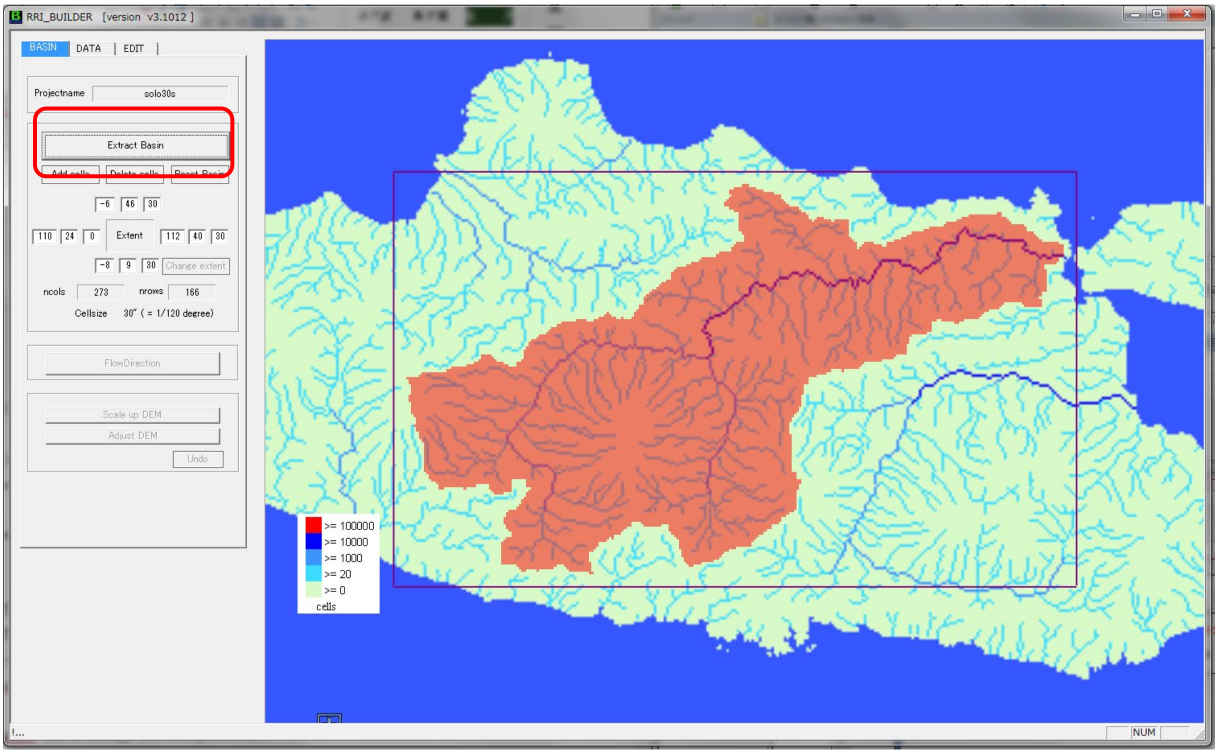
Zoom into Java Island in Indonesia





After zoom into the outlet area of the Solo River basin, click a pixel along the main river near the river mouth (not necessary to be exactly the same as the above example).

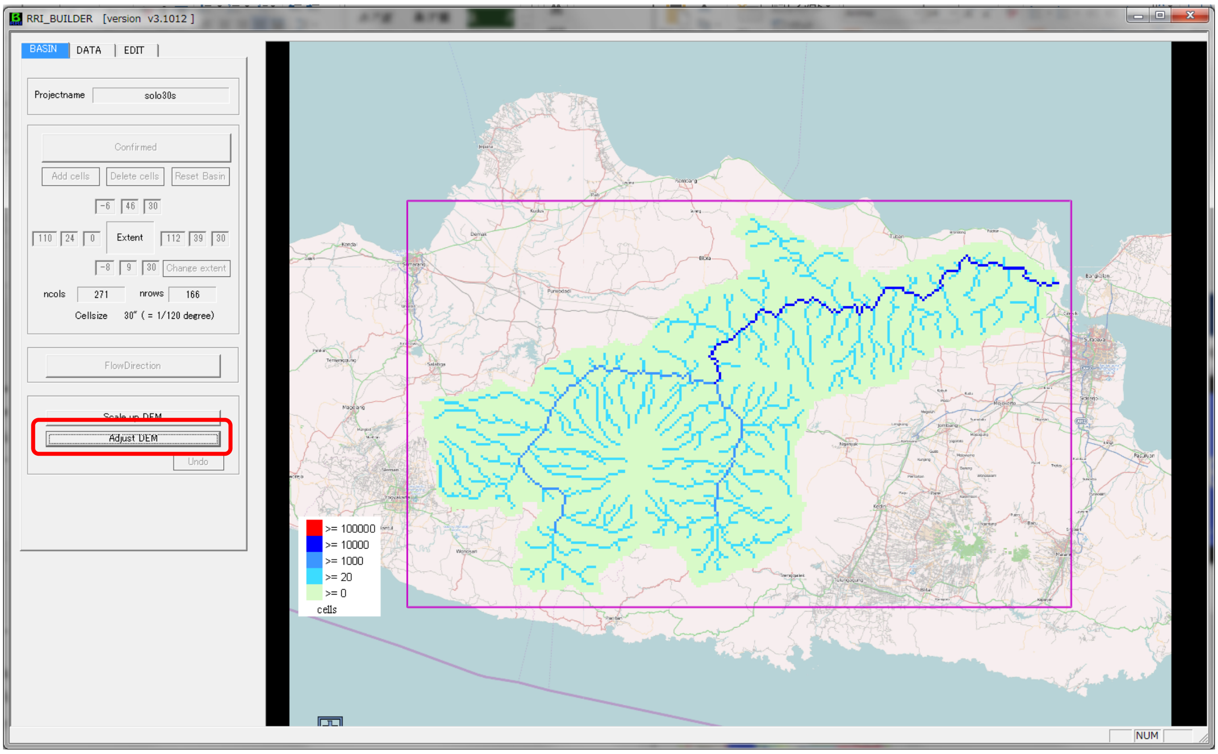
Then choose "Yes" on the window and "Confirm" on the left panel.



Click "Extract Basin" after you confirm the area of the basin.

(If not satisfactory, click "Reset Basin" and retry it.)

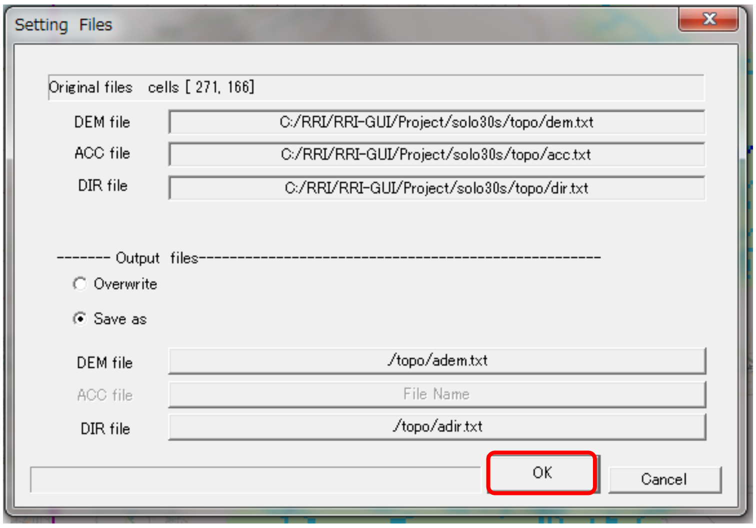
If the background map is available, the following extracted basin will be displayed. (Even the background image is not shown for some reasons, it is essentially no problem for the following simulation).



(Optional)

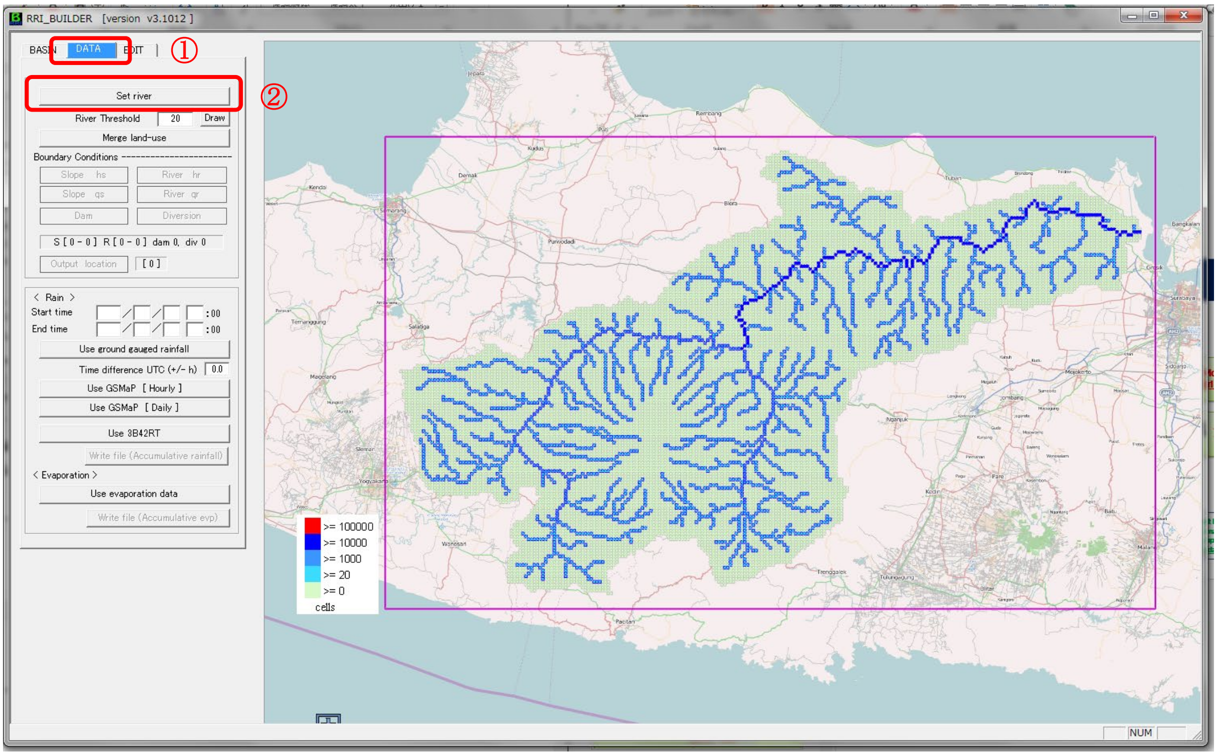
The step of "Scale up DEM" is an optional. Use this option in case you want to scale up the DEM for example changing the model resolution from 30 second to 60 seconds.

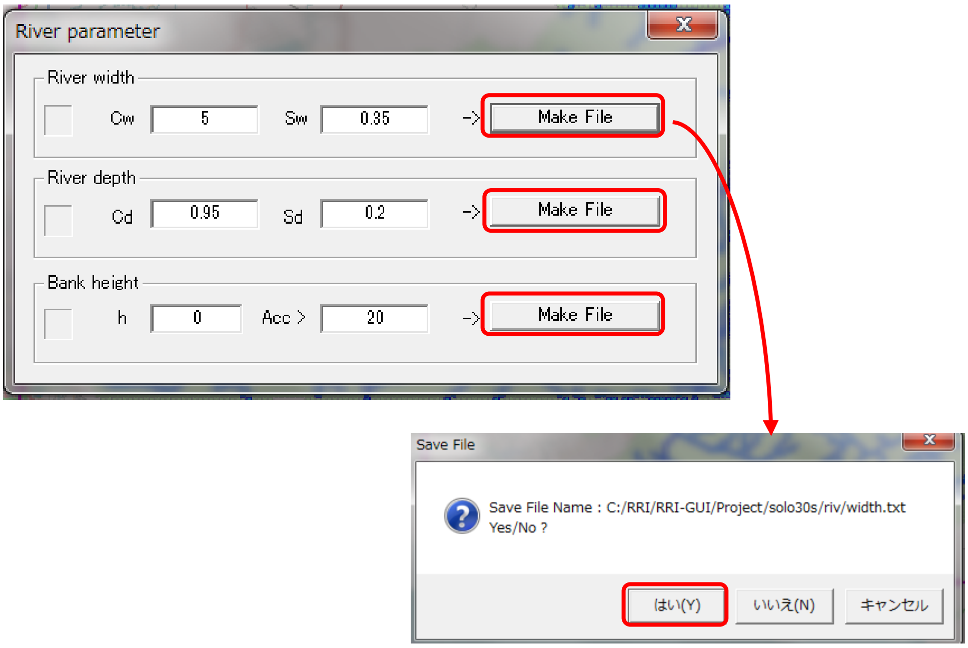
The next step is to execute "AdjustDEM". This procedure is always necessary for the stable simulation.



Choose OK with the default setting. (you will see a command screen running AdjustDEM program).

Now select “DATA” Tab on the left top and click “Set river”.



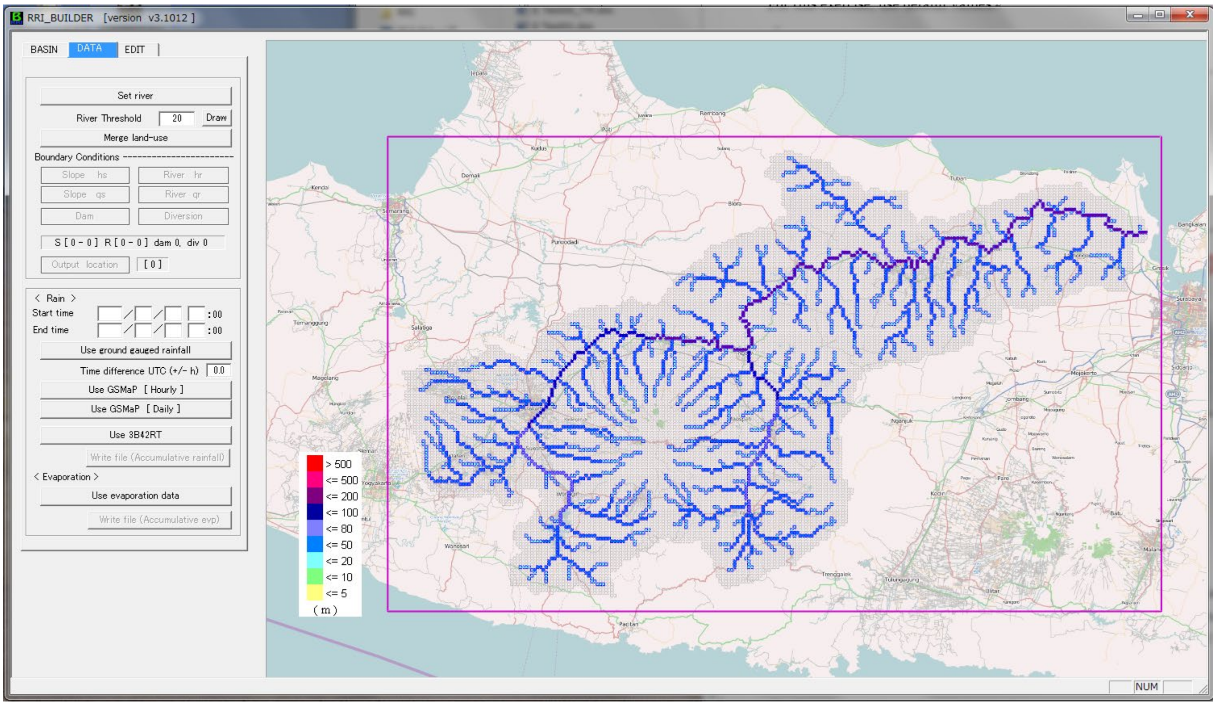


Click all the three “Make File” on River parameter setting.

These values are the parameters to determine the cross sections based on the equations.

For this exercise, use default values.

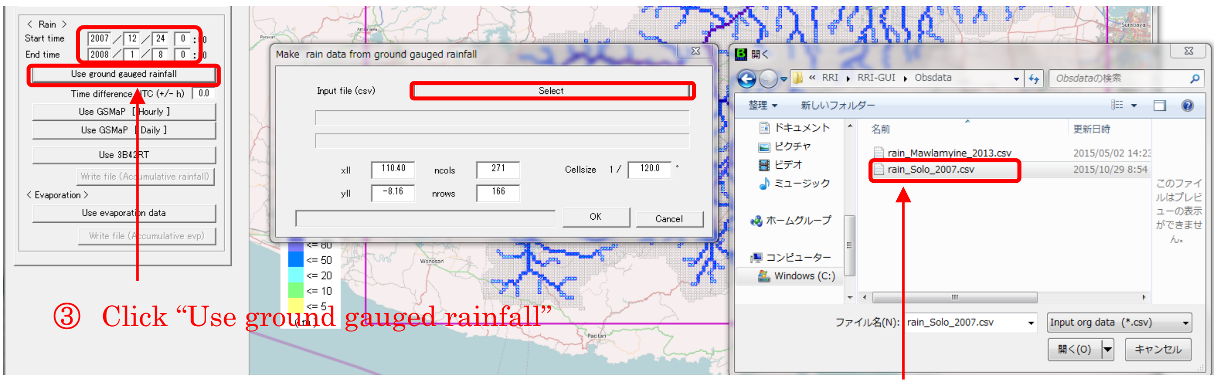
After confirming the three green signs, close this window.



9.2.2 Preparing Input Rainfall Data

① Set “Start time” and “End time” under <Rain>

Start : 2007/12/24 0:00, End : 2008/1/8 0:00

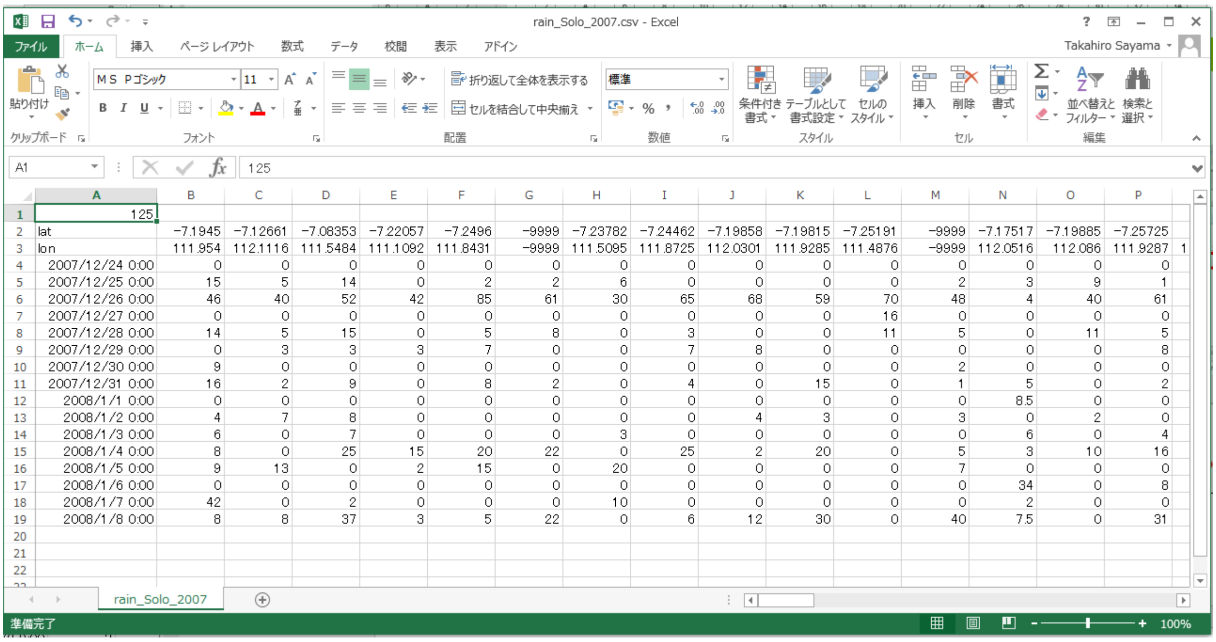


③ Click “Use ground gauged rainfall”

② Click “Select” to find a input rainfall file

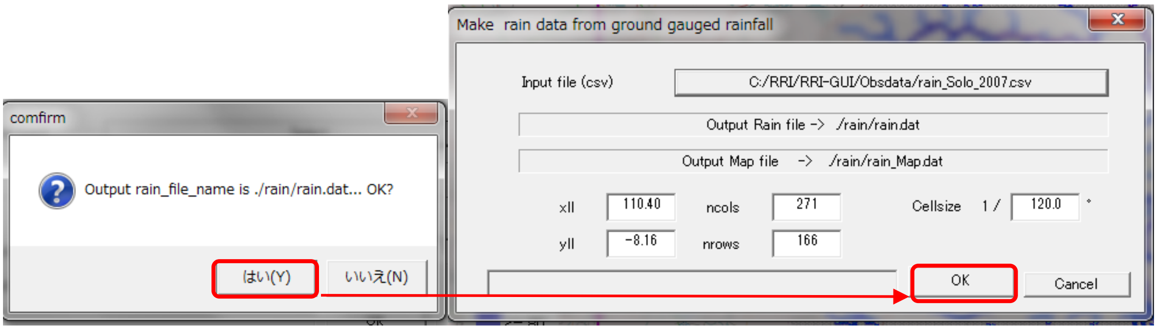
(In this excercise, “RRI-GUI/Obsdata/rain_Solo_2007.csv”)

An input rainfall file must be the following format saved as csv. The file can be prepared by a text editor or Excel (saved as csv).

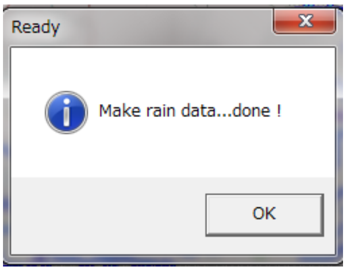


	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1	125															
2	lat	-7.1945	-7.12661	-7.08353	-7.22057	-7.2496	-9999	-7.23782	-7.24462	-7.19858	-7.19815	-7.25191	-9999	-7.17517	-7.19885	-7.25725
3	lon	111.954	112.1116	111.5484	111.1092	111.8431	-9999	111.5095	111.8725	112.0301	111.9285	111.4876	-9999	112.0516	112.086	111.9287
4	2007/12/24 0:00	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
5	2007/12/25 0:00	15	5	14	0	2	2	6	0	0	0	0	2	3	9	1
6	2007/12/26 0:00	46	40	52	42	85	61	30	65	68	59	70	48	4	40	61
7	2007/12/27 0:00	0	0	0	0	0	0	0	0	0	0	16	0	0	0	0
8	2007/12/28 0:00	14	5	15	0	5	8	0	3	0	0	11	5	0	11	5
9	2007/12/29 0:00	0	3	3	3	7	0	0	7	8	0	0	0	0	0	8
10	2007/12/30 0:00	9	0	0	0	0	0	0	0	0	0	0	2	0	0	0
11	2007/12/31 0:00	16	2	9	0	8	2	0	4	0	15	0	1	5	0	2
12	2008/1/1 0:00	0	0	0	0	0	0	0	0	0	0	0	0	8.5	0	0
13	2008/1/2 0:00	4	7	8	0	0	0	0	0	4	3	0	3	0	2	0
14	2008/1/3 0:00	6	0	7	0	0	0	3	0	0	0	0	0	6	0	4
15	2008/1/4 0:00	8	0	25	15	20	22	0	25	2	20	0	5	3	10	16
16	2008/1/5 0:00	9	13	0	2	15	0	20	0	0	0	0	7	0	0	0
17	2008/1/6 0:00	0	0	0	0	0	0	0	0	0	0	0	0	34	0	8
18	2008/1/7 0:00	42	0	2	0	0	0	10	0	0	0	0	0	2	0	0
19	2008/1/8 0:00	8	8	37	3	5	22	0	6	12	30	0	40	7.5	0	31

Please note that the format is slightly different from the one used by the Thiessen Polygon program explained for RRI-CUI (Command User Interface). The first column of the fille (L4-) is date and time. Currently the date and time must be in the form of "yyyy/mm/dd h:mm".



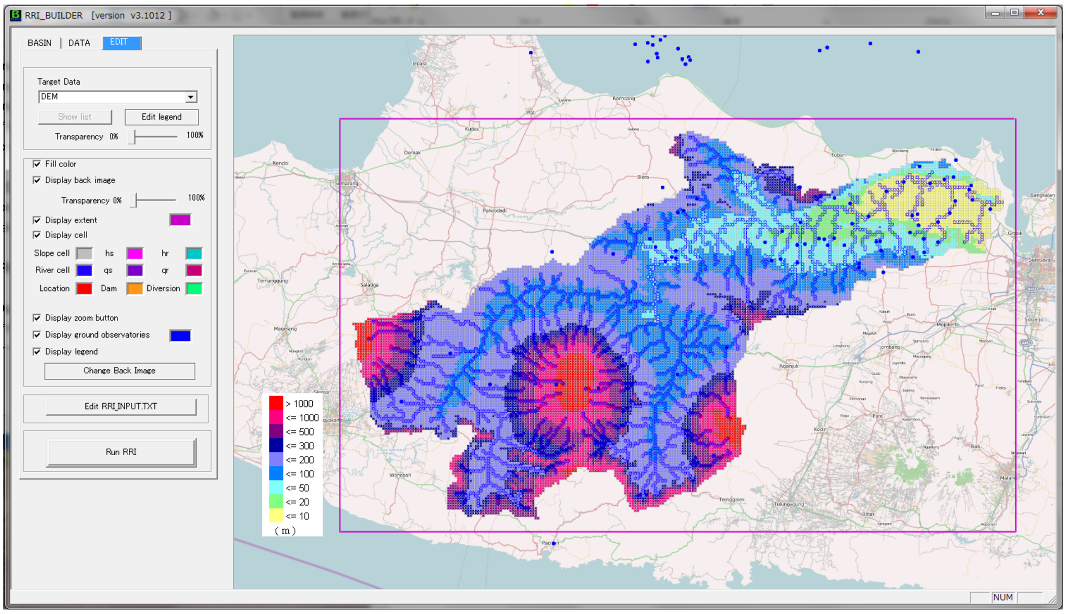
After selecting the input csv file, please choose "Yes" on the confirmation window then click "OK" with the default setting of the creating rainfall distribution file.

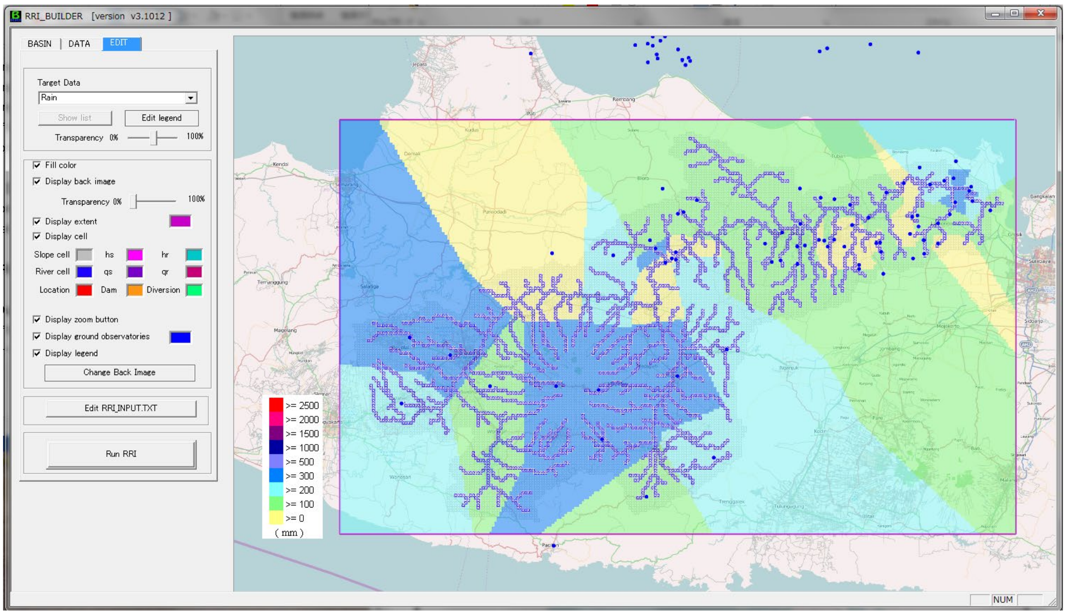


9.2.3 Running RRI Model

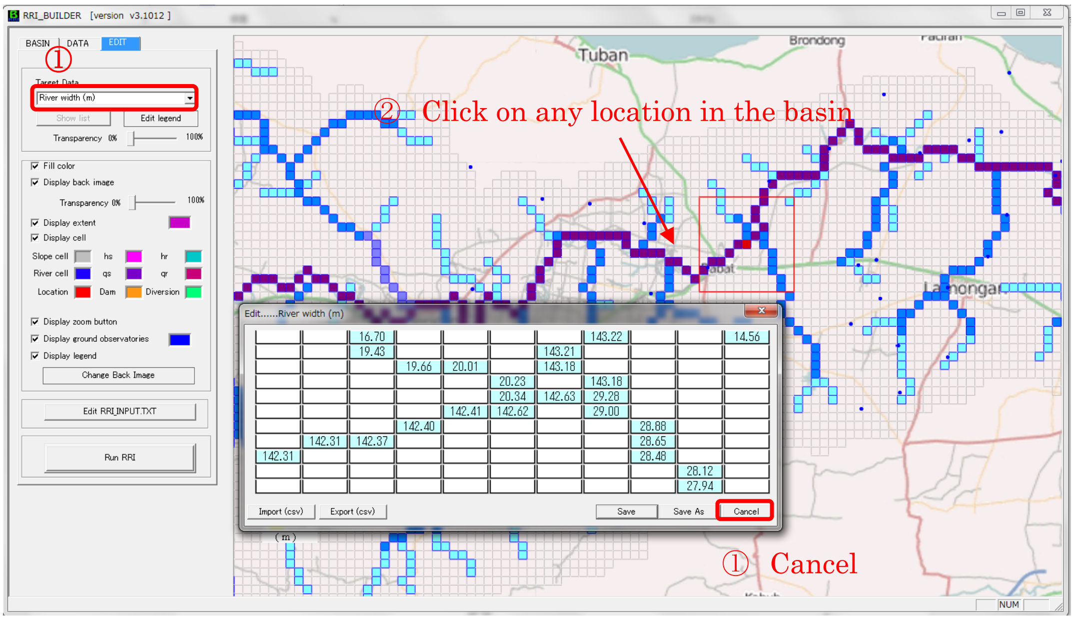
Select “Edit” tab after the topographic and rainfall data is ready.

You can confirm different distributions including DEM, ACC, DIR, River Width, River Depth, Bank height as well as cumulative Rain.

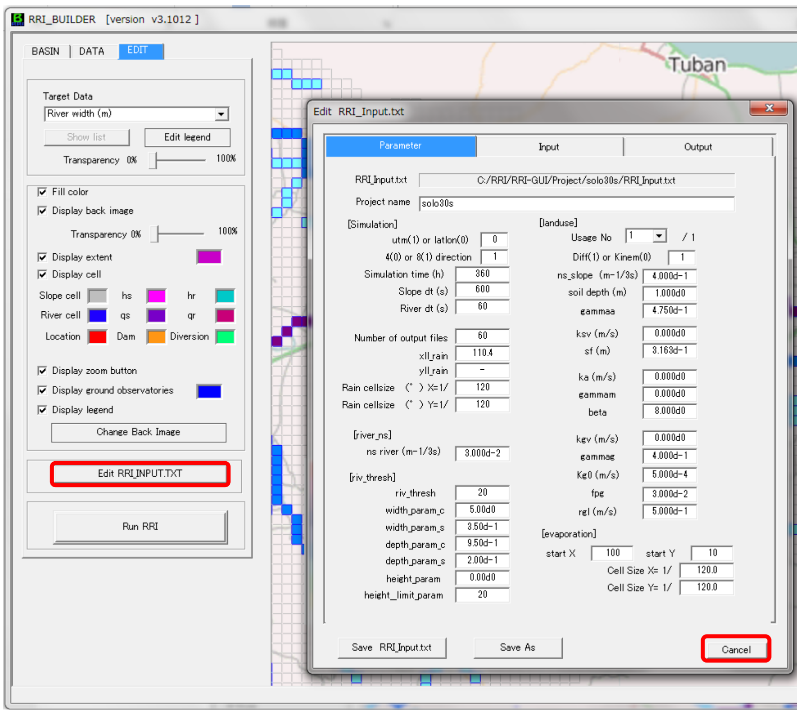




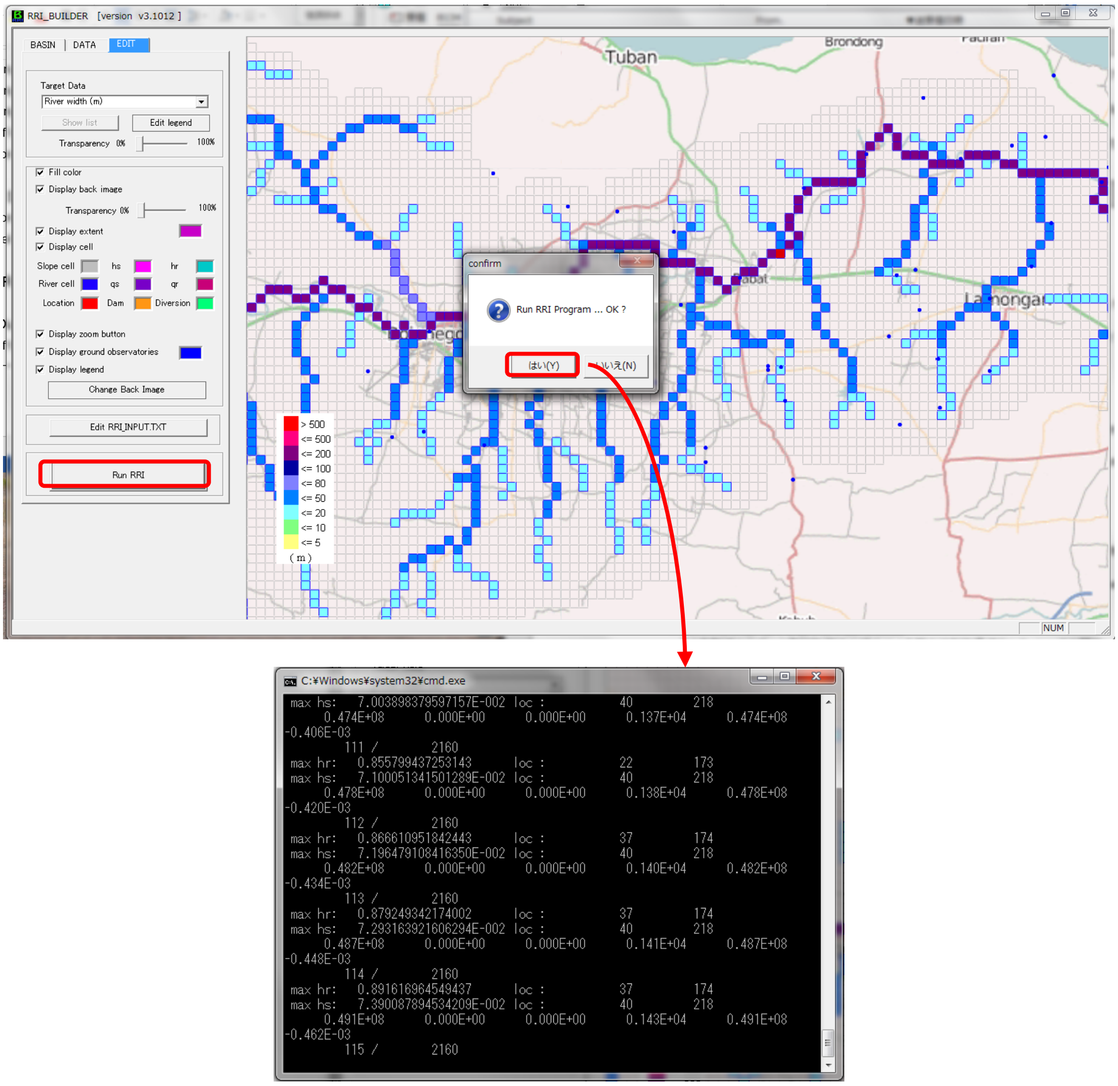
These distribution files except for the cumulative rainfall, can be edited on the window. For example you can choose river width and select any area inside the basin to display the following image. (For this exercise, no need to change the values.)



In addition, you find parameter and other input file settings if you click "Edit RRI_INPUT.TXT". The editting the values will be reflected to the RRI_Input.txt file, which is the control file of the RRI model. (For this exercise, no need to change the values.)

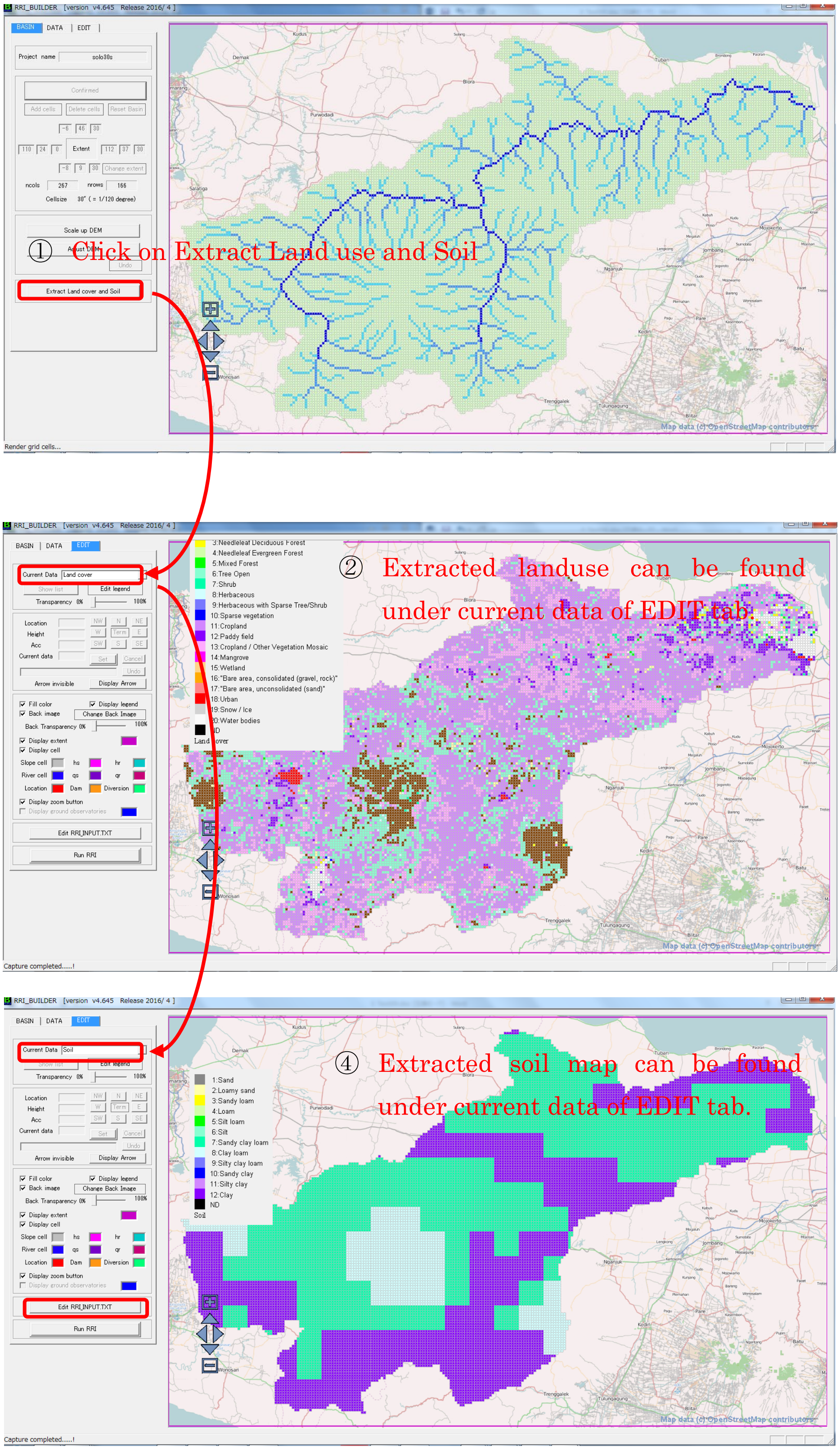


Finally, click "Run RRI" and "OK" to start the simulation.



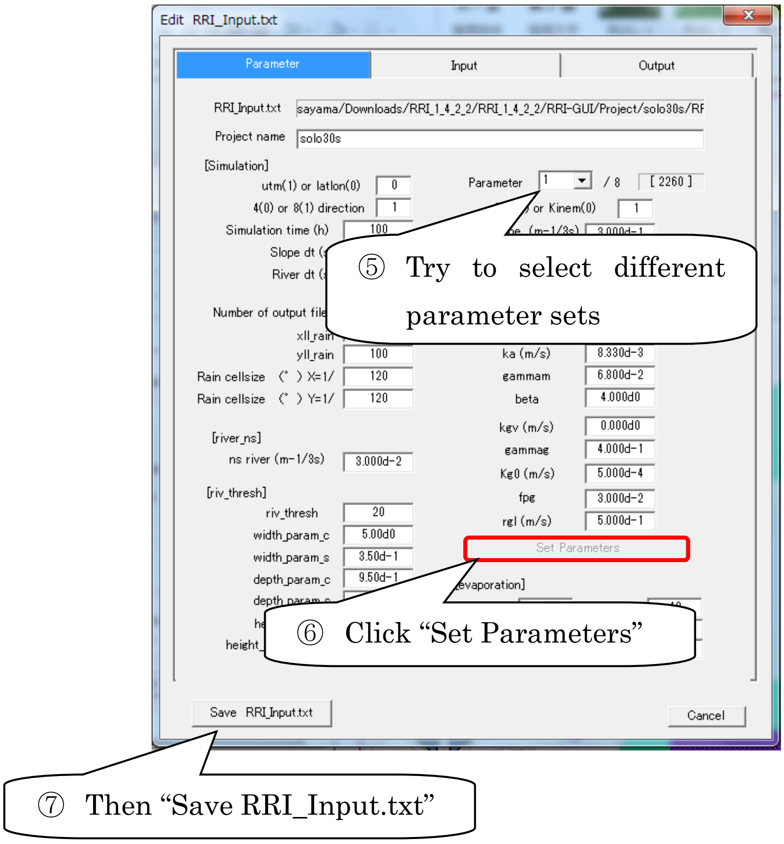
9.2.4 Additional functions of RRI_BUILDER

The latest version of "RRI_BUILDER.exe" has a function to use global dataset for landuse and soil distribution to assign different sets of model parameters. To read the maps for the target river basin, one can click "Extract Land cover and Soil" under BASIN tab.



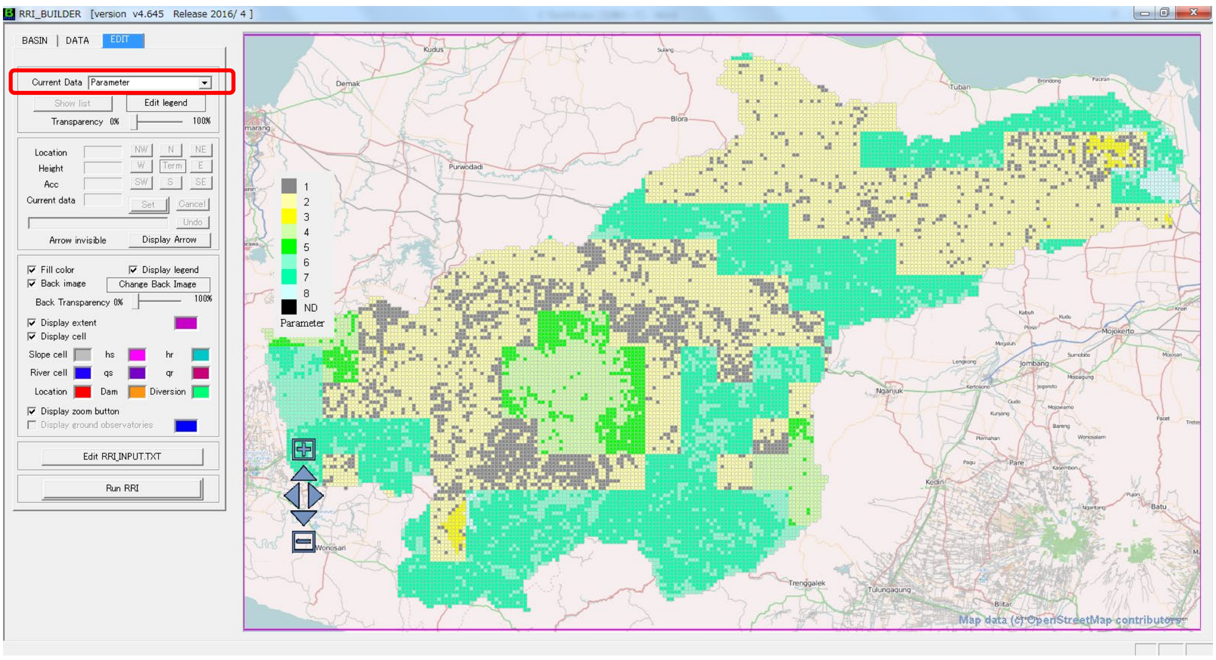
③ Click "EDIT RRI_Input.txt" to set different values for each combination of the landuse and soil distribution.

Under "EDIT RRI_Input.txt", click "Set Parameters", so that different parameters are assigned (e.g. 1, 2, 3, … 8), which can be confirmed by selecting "Parameter".



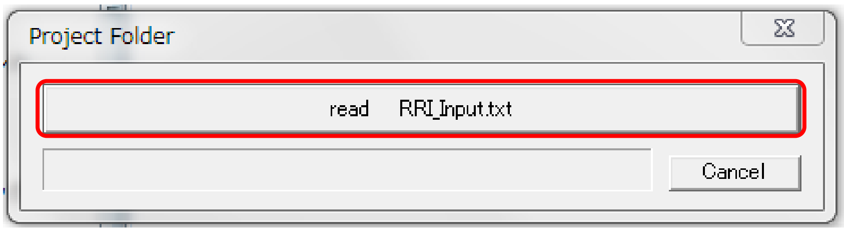
Finally Save "RRI_Input.txt" to update the RRI_Input.txt file.

Note that now new map called "Parameter" is created under Current Data of Edit tab. The map shows the distributions of ID numbers, corresponding to the above parameter set created by the combination of landuse and soil distribution. For example, the area with the gray color in the following map (i.e. ID = 1) will be assigned with parameter set 1.

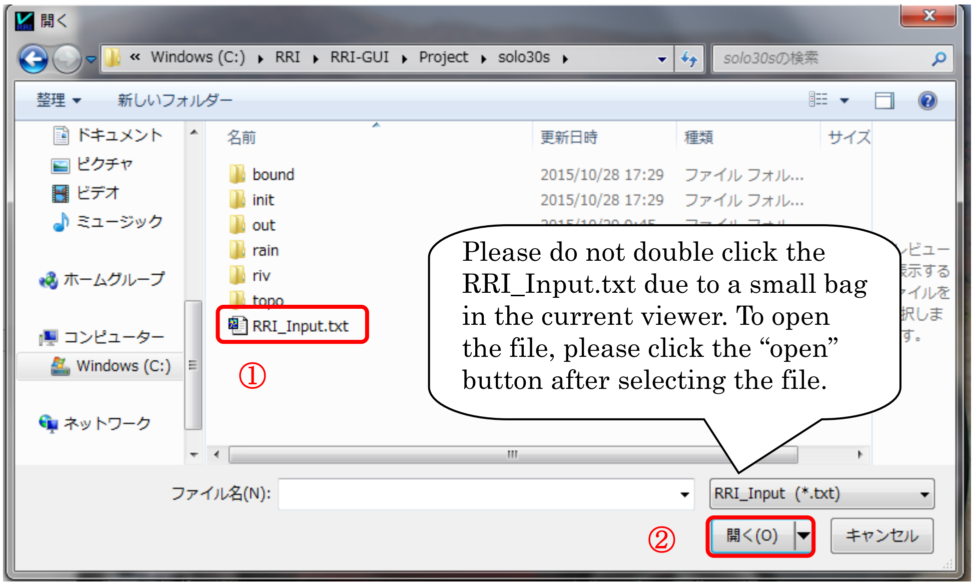


9.3 Visualizing results with RRI_VIEWER

Execute “RRI_VIEWER_64.exe” (or _32.exe for a 32 bit machine).

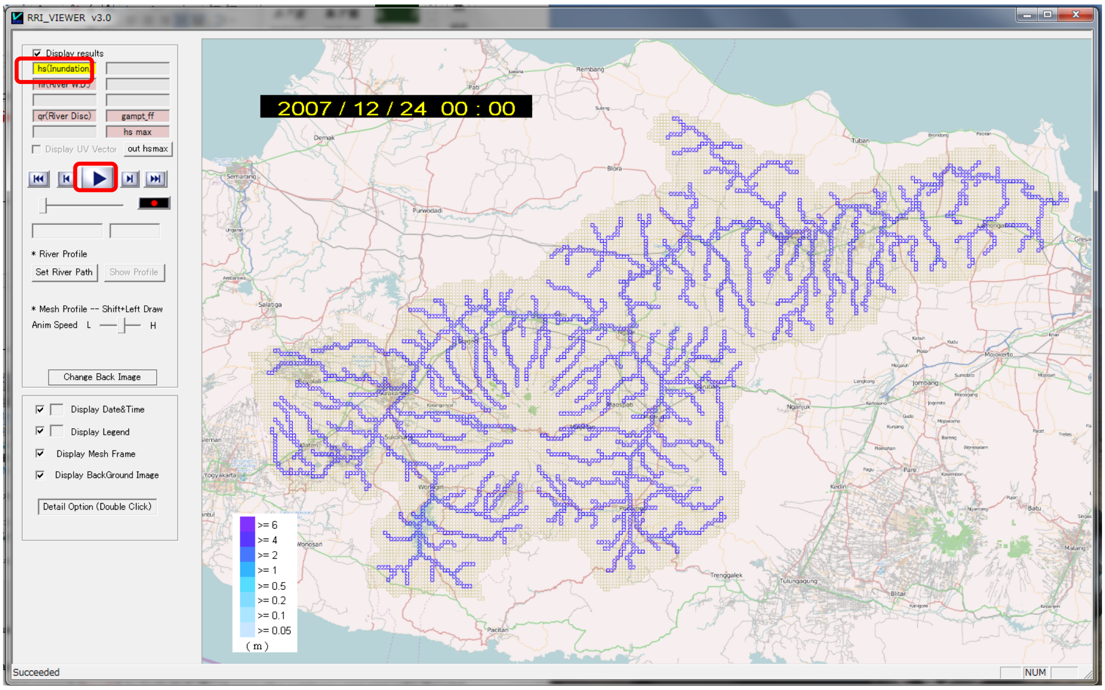


Read RRI_Input.txt file prepared in the previous subsection. In this exercise, find **RRI-GUI/Project/solo30s**

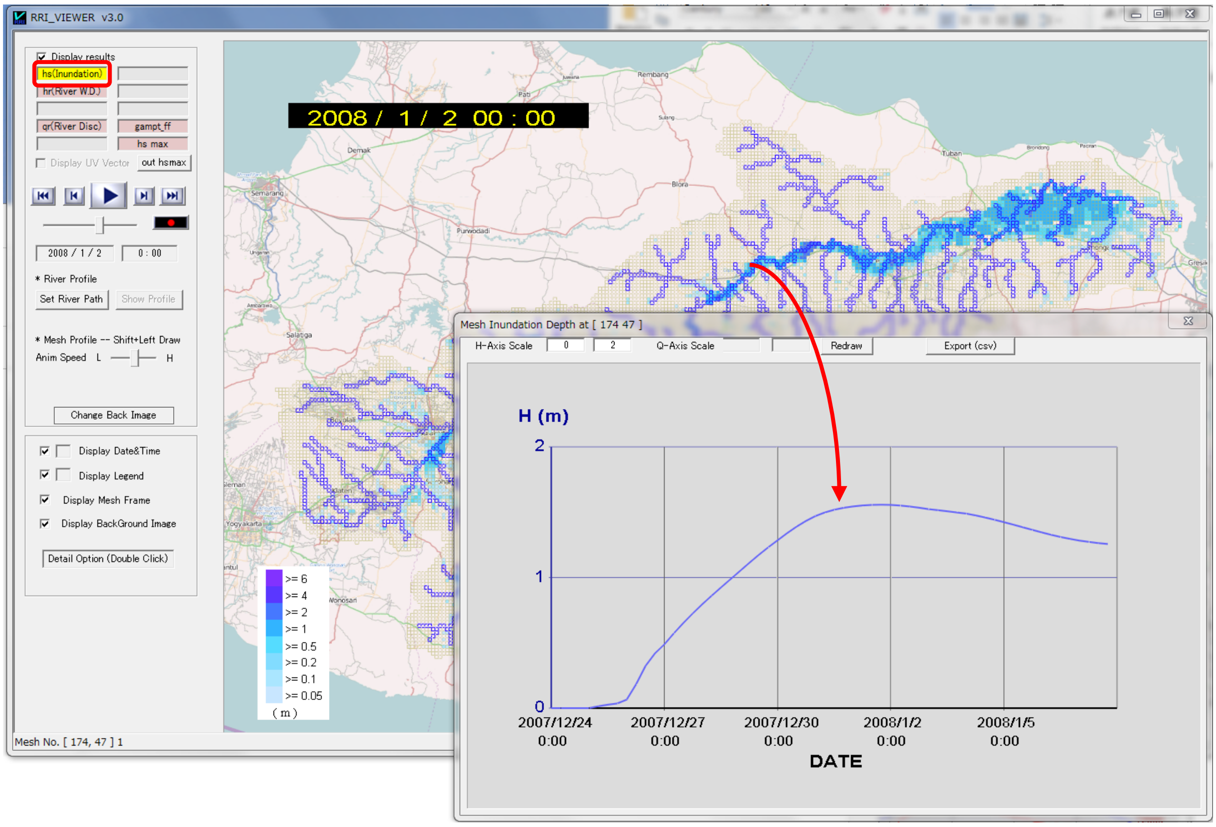


9.3.1 Visualize flood inundation

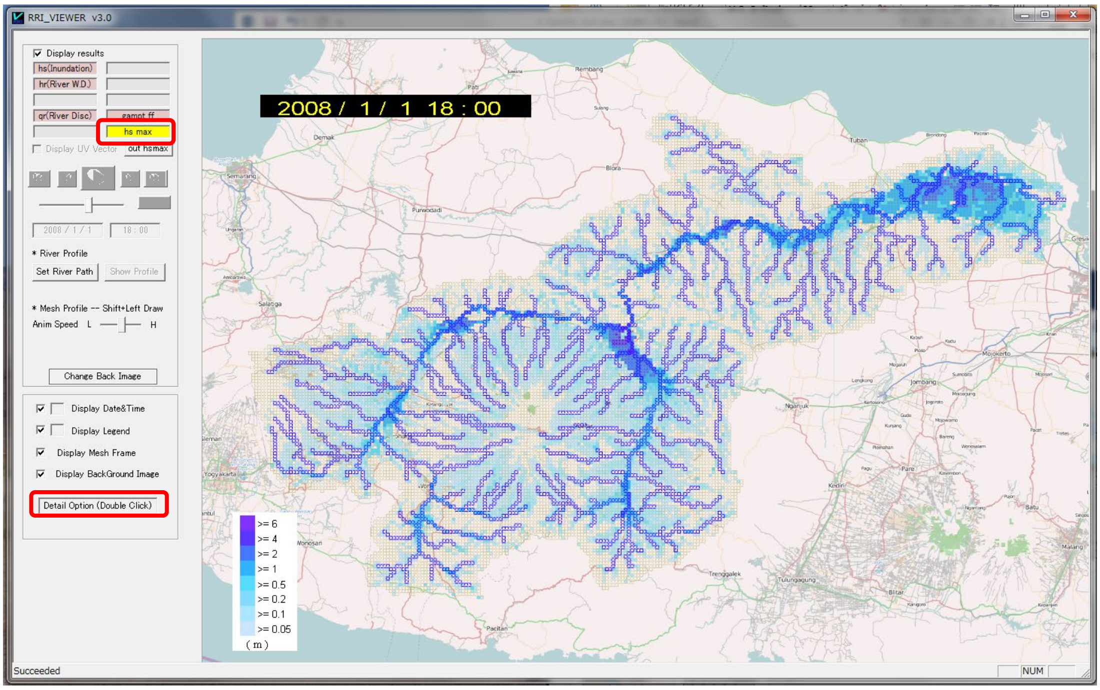
On the displayed map image, one can use CTRL and left drag to move the map and also CTRL and mouse scroll to zoom in and out. This operation is the same as RRI_BUILDER_64.exe



To display the animation of flood inundation depth distribution, please select inundation on the top left panel and click the start button.

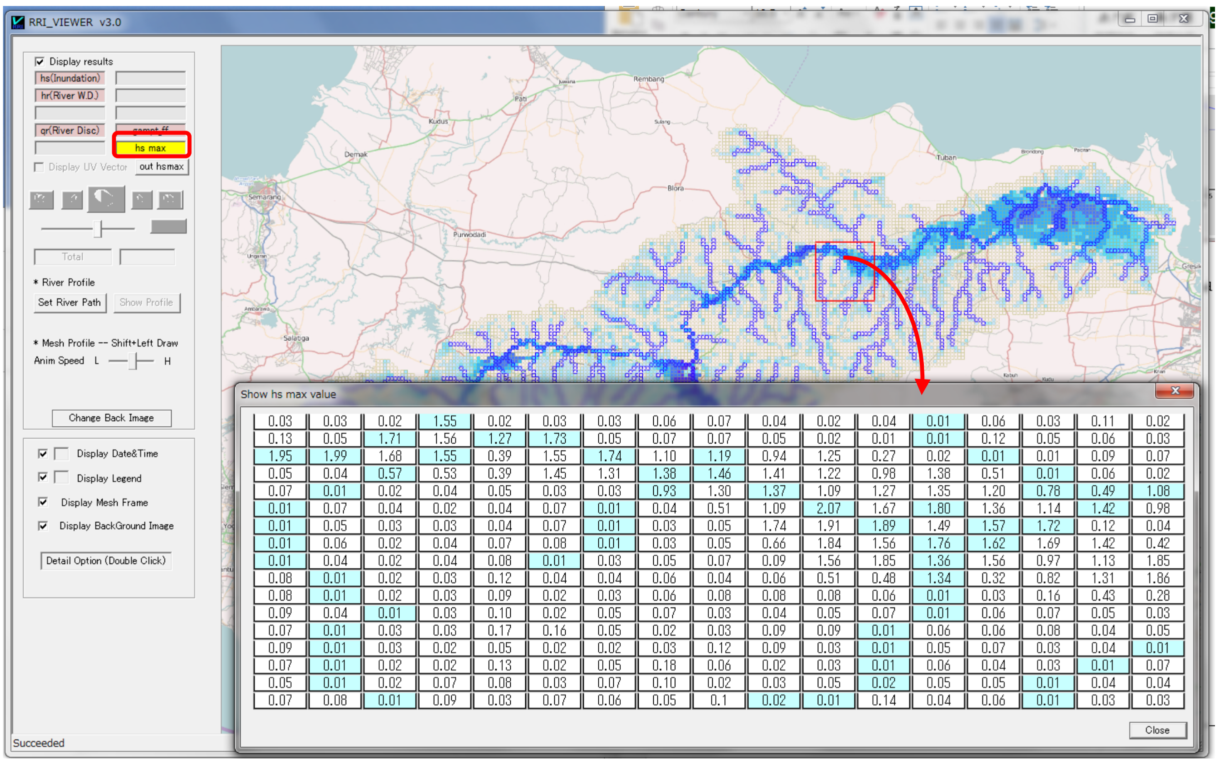


After stopping the animation, try to click any grid cell inside a basin to visualize the time seris of flood inundation depths.

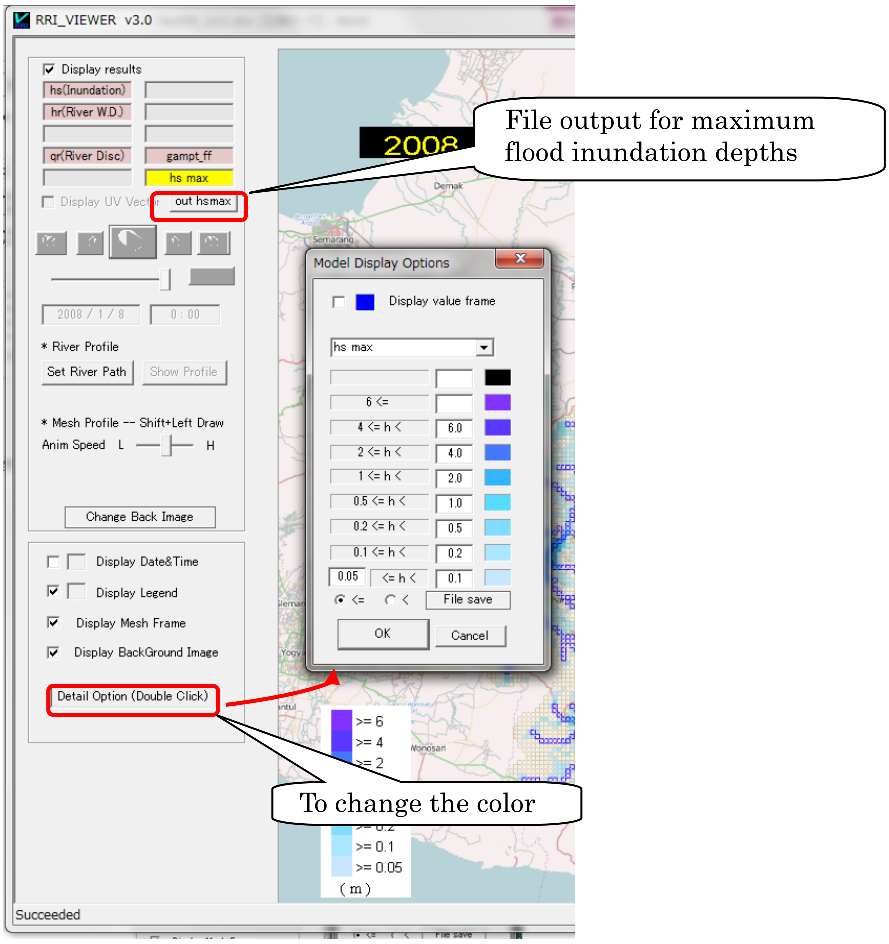


Then display the maximum inundation depth distribution by choosing hs max.

For the maximum inundation depths, one can check values by selecting a area on the map.



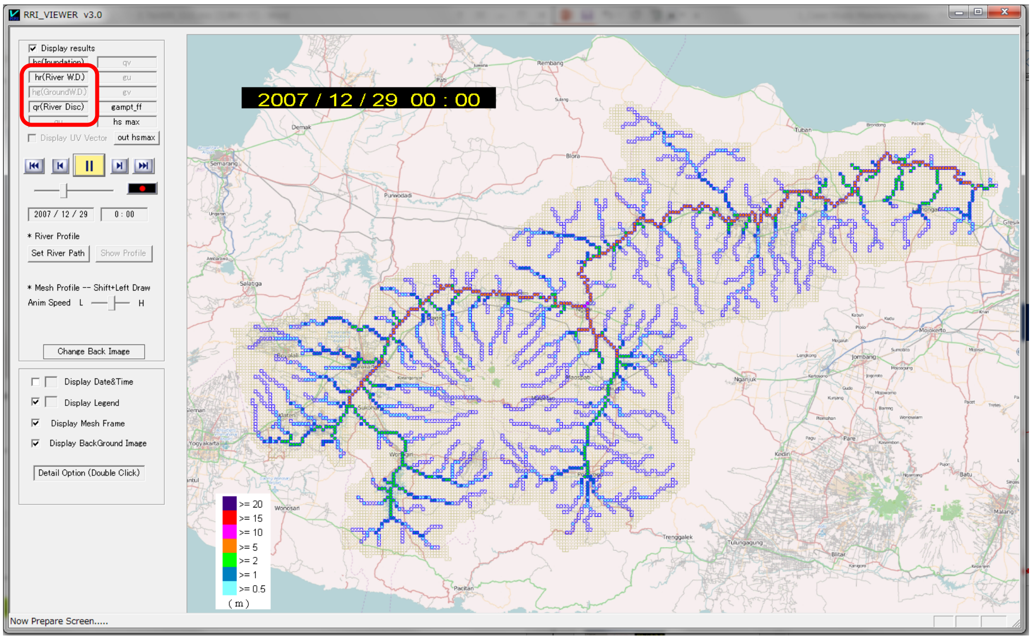
(Optional)



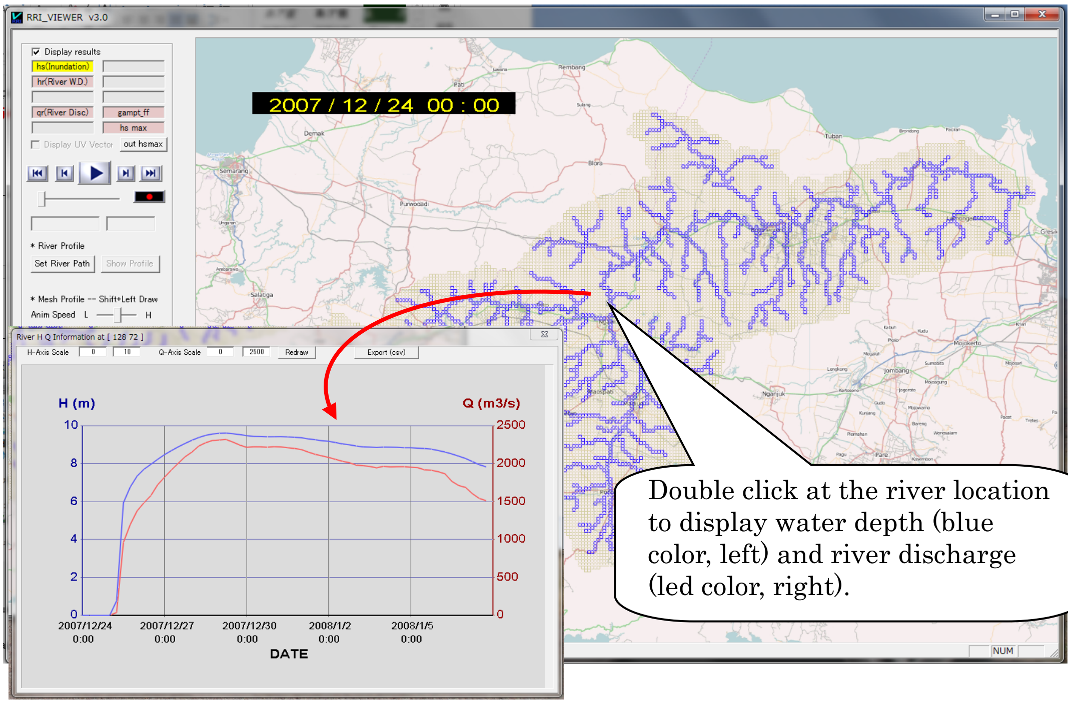
9.3.2 Visualize river discharge and water depth

To display the animation of river discharge or river water depth distribution, please select qr (River Disc.) or hr (River WD) on the top left panel and click the start button.

River discharge (qr) or river water depth (hr)

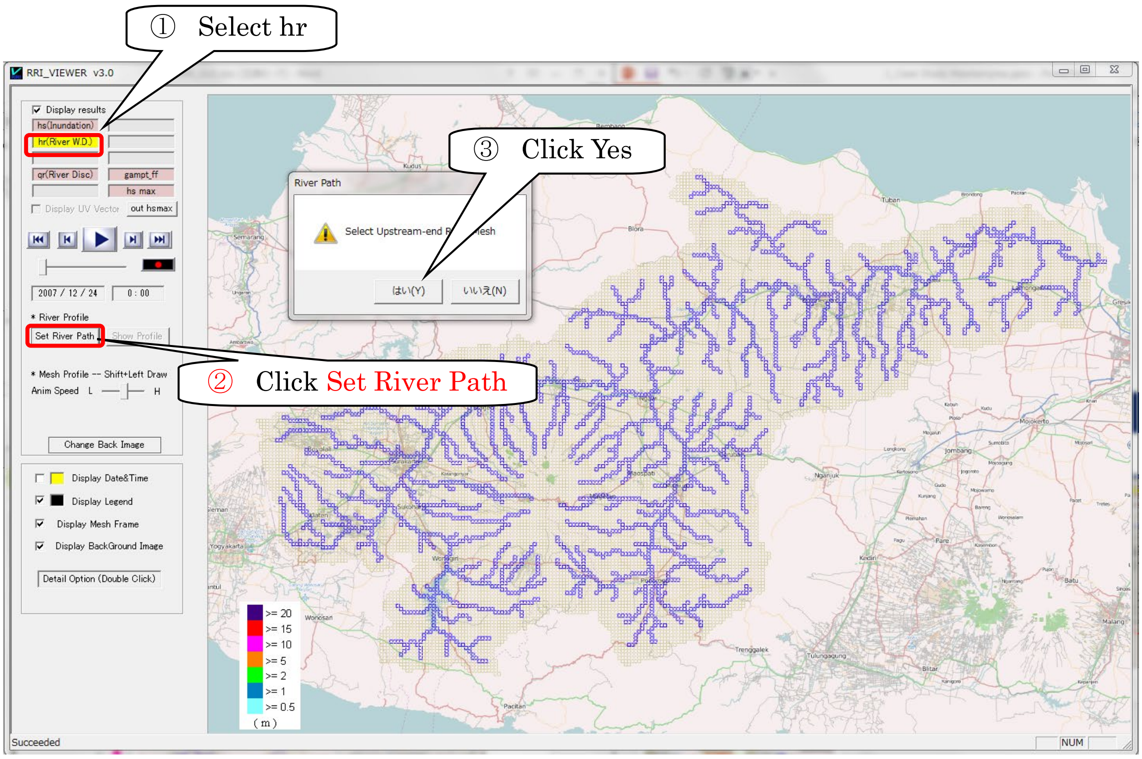


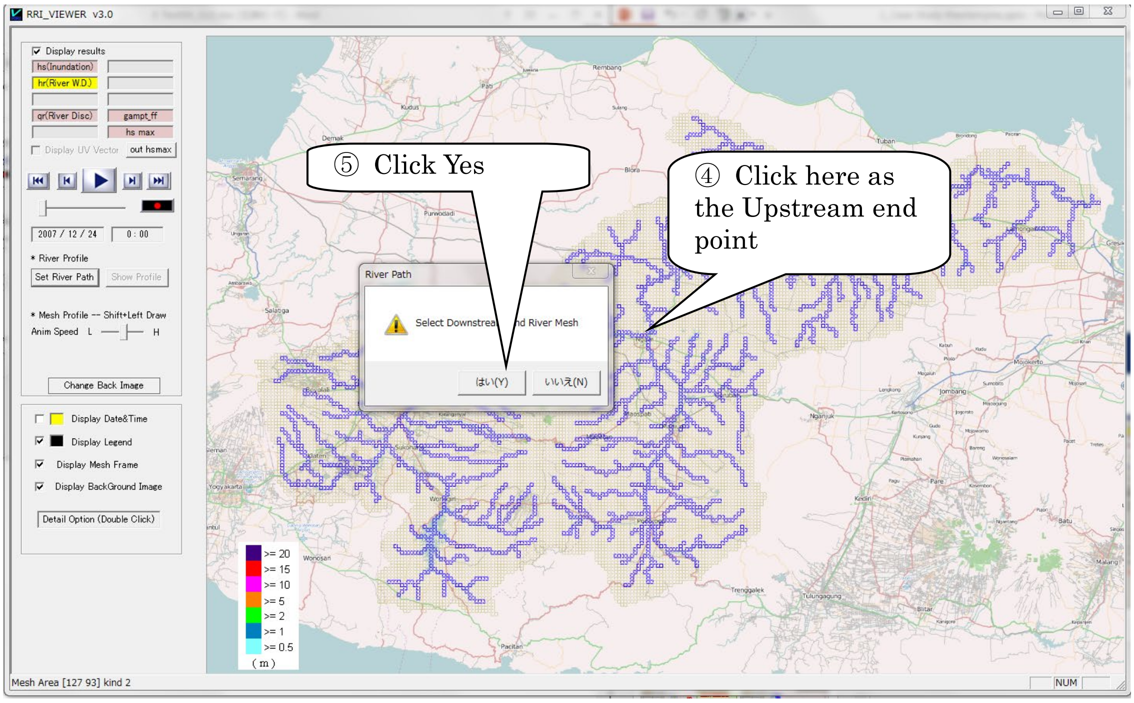
After stopping the animation, try to click any river grid-cell to visualize the time serises of river discharge (i.e. hydrograph) and river water depth.

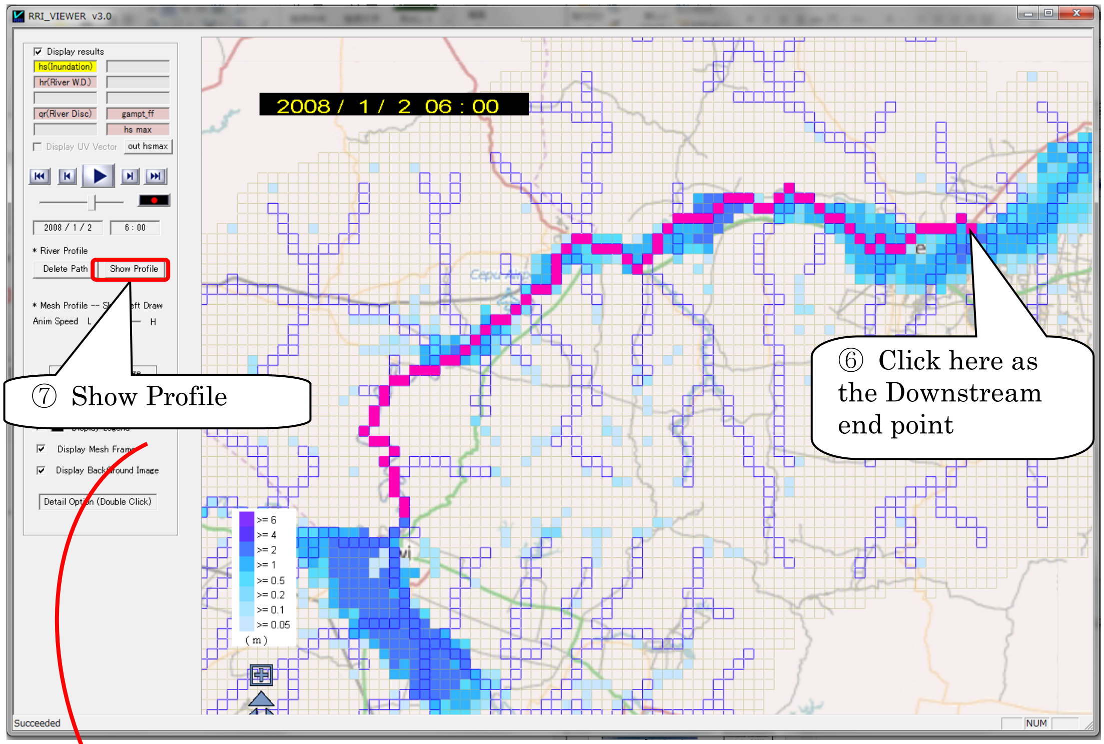


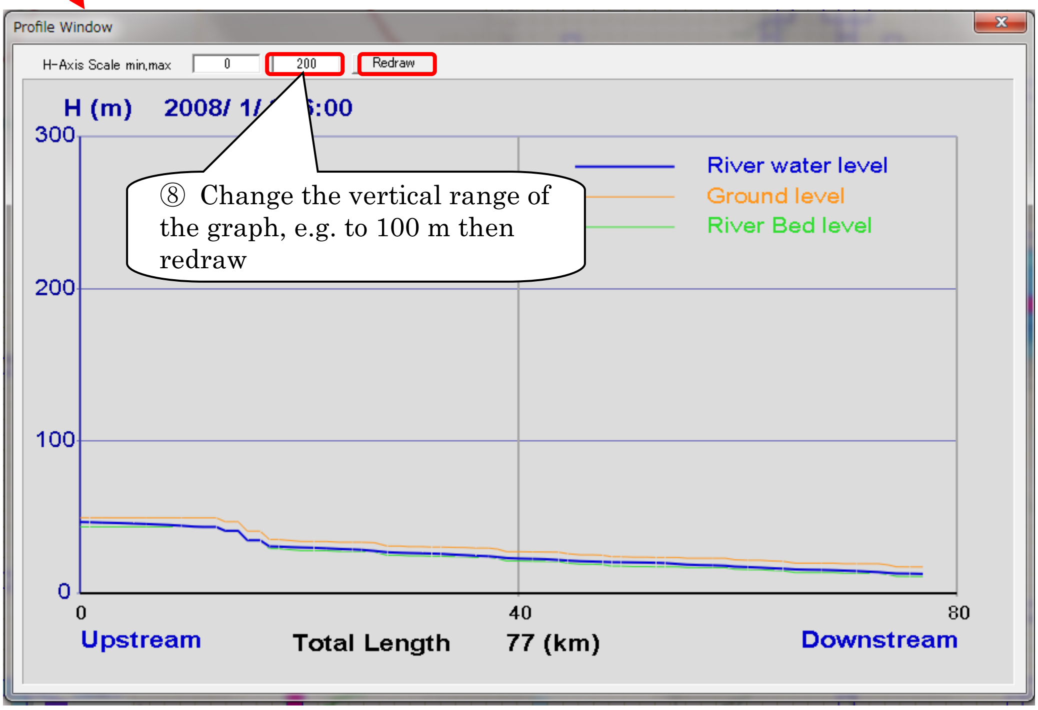
9.3.3 Visualize the longitudinal profile of river water level

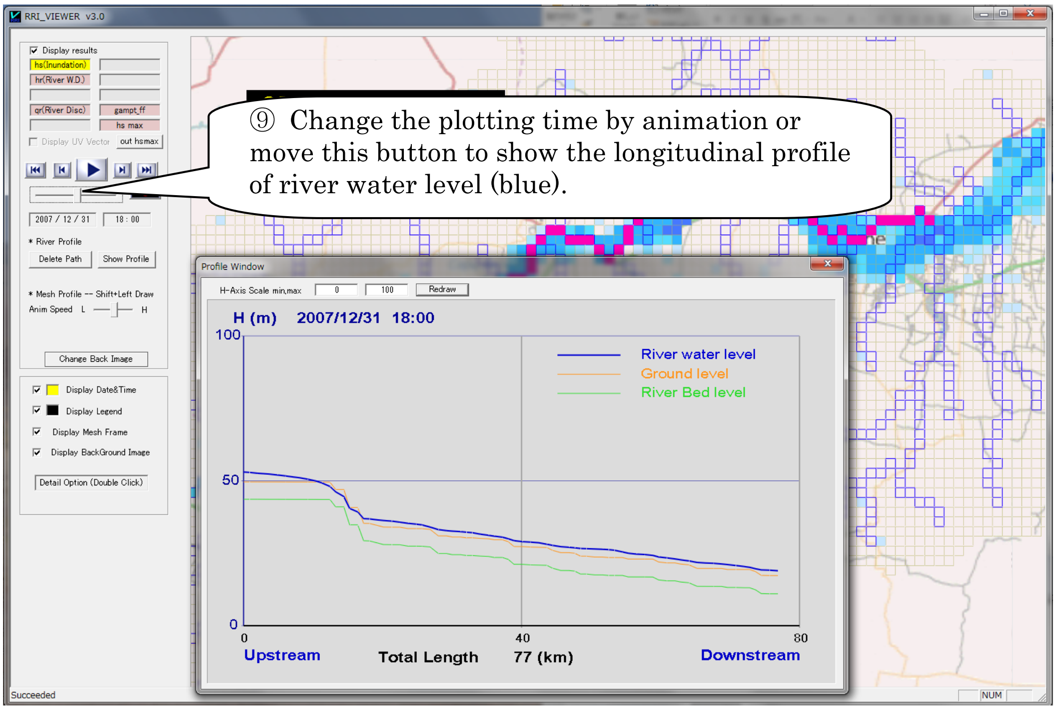
To visualize the longitudinal profile of river water level, first select hr (River WD) and click "Set River Path" on the left pannel.


① Select hr
③ Click Yes
② Click Set River Path
RRI_VIEWER v3.0
Display results
hs(Inundation)
hr(River W.D.)
qr(River Disc)
gampt_ff
hs max
Display UV Vector
out hsmax
2007 / 12 / 24
0 : 00
River Profile
Set River Path
Show Profile
Mesh Profile -- Shift+Left Draw
Anim Speed
Change Back Image
Display Date&Time
Display Legend
Display Mesh Frame
Display BackGround Image
Detail Option (Double Click)
River Path
はい(Y)
いいえ(N)
>= 20
>= 15
>= 10
>= 5
>= 2
>= 1
>= 0.5
(m)
Succeeded
NUM


⑤ Click Yes
④ Click here as the Upstream end point
RRI_VIEWER v3.0
Display results
hs(Inundation)
hr(River W.D.)
qr(River Disc)
gampt_ff
hs max
Display UV Vector
out hsmax
2007 / 12 / 24
0 : 00
River Profile
Set River Path
Show Profile
Mesh Profile -- Shift+Left Draw
Anim Speed
Change Back Image
Display Date&Time
Display Legend
Display Mesh Frame
Display BackGround Image
Detail Option (Double Click)
River Path
はい(Y)
いいえ(N)
>= 20
>= 15
>= 10
>= 5
>= 2
>= 1
>= 0.5
(m)
Mesh Area [127 93] kind 2
NUM


RRI_VIEWER v3.0
Display results
hs(Inundation)
hr(River W.D.)
qr(River Disc)
gampt_ff
hs max
Display UV Vector
out hsmax
2008 / 1 / 2
6 : 00
River Profile
Delete Path
Show Profile
Display Mesh Frame
Display Background Image
Succeeded
NUM
2008 / 1 / 2 06 : 00
>= 6
>= 4
>= 2
>= 1
>= 0.5
>= 0.2
>= 0.1
>= 0.05
(m)
⑦ Show Profile
⑥ Click here as the Downstream end point


Profile Window
H-Axis Scale min,max
0
200
Redraw
H (m)
300
200
100
0
0
40
80
River water level
Ground level
River Bed level
Upstream
Total Length
77 (km)
Downstream
⑧ Change the vertical range of the graph, e.g. to 100 m then redraw



Click "Delete Path" to delete the selected longitudinal path.

9.3.1 Visualize the profile of flood inundation depth

To visualize the profile of flood inundation depth, one can draw a profile line (e.g. red line below) as "Shift + Left Draw".