Grid based 2-dimensional land surface flood model User's Manual

Yun Seok Choi, Kyung Tak Kim

Department of Hydro Science and Engineering Research
Korea Institute of Civil Engineering and Building Technology

v.G2D2021.06
March 2022

KOREA INSTITUTE of CIVIL ENGINEERING and BUILDING TECHNOLOGY

History

Grid based 2−Dimensional land surface flood model (G2D) is a squared grid−based 2D surface flood model developed by the Korea Institute of Civil Engineering and Building Technology (KICT). The development history of the G2D model is as follows.

- The G2D model was first developed in 2017.
- In 2018, simulation speed and stability were improved.
- In 2019, GUI was developed as QGIS plug−in.
- In 2020, the G2D model written by C++ was developed.
- In 2021, the simulator using NVIDIA GPU was developed.

The G2D model is constantly being developed at the KICT for improving the stability and accuracy of the models, adding functions, and developing modeling software.

April. 2021.

Table of Contents

1. Overview of the G2D

1.1 Model structure

The purpose of Grid based 2-Dimensional land surface flood model (G2D) is to simulate flooding due to flow or rainfall. The control volume, which is the minimum unit of calculation, uses a structured square grid, and constructs a domain composed of square grids for the simulation target area using DEM. It is possible to set the discharge (or depth) to the arbitrary grid within the domain as a boundary condition or to apply rainfall to the entire domain to simulate the two-dimensional flooding.

The governing equations for surface flow analysis use diffusion wave(+) (or dynamic wave(-)) equation which additionally include a local acceleration term in the diffuse wave equation. The G2D model simulates only the surface flow, and does not calculate the infiltration into the soil and the evapotranspiration. The spatial discretization of the governing equations uses the finite volume method, and the temporal difference is the implicit method. For the nonlinear terms, the convergence solution is derived using the Newton-Raphson method, and the convergence of the total control volume is calculated by the Gauss-Seidel method(Choi et al., 2019).

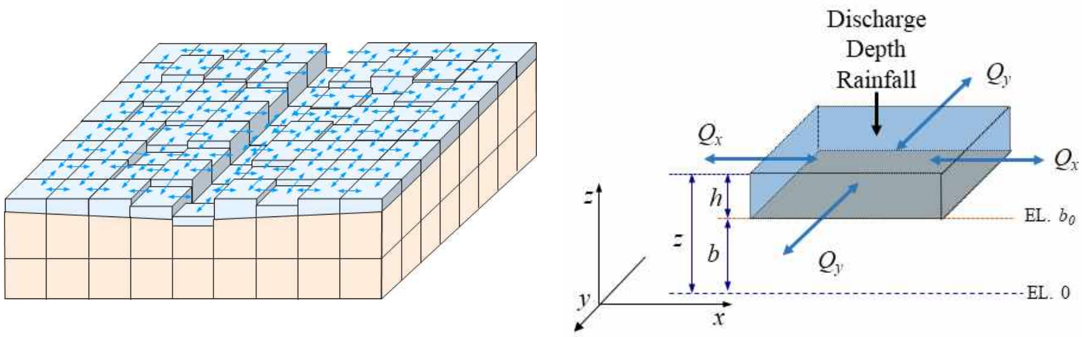


Fig. 1.1 Configuration of control volume and input/output of mass

1.2 Governing equations

For the 2 - dimensional flow analysis of the land surface, governing equations were constructed by continuity equation and momentum equation. Here, the momentum equation excludes the convective acceleration term in the dynamic wave equation.

$$\frac{dh}{dt}+\frac{dq_x}{dx}+\frac{dq_y}{dy}=s \tag{1.1}$$

$$\frac{\partial q_x}{\partial t}+gh\frac{\partial h}{\partial x}-gh(S_{bx}-S_{fx}) = 0 \tag{1.2}$$

$$\frac{\partial q_y}{\partial t}+gh\frac{\partial h}{\partial y}-gh(S_{by}-S_{fy}) = 0 \tag{1.3}$$

$$S_{fx} = \frac{u^2n^2}{h^{4/3}} \tag{1.4}$$

$$S_{fy} = \frac{v^2n^2}{h^{4/3}} \tag{1.5}$$

Where q_x and q_y are the discharge per unit width in the x and y directions, t is the time, s is the source term, g is the gravitational acceleration, h is the depth, S_{bx} and S_{by} are the ground slope in the x and y directions, S_{fx} and S_{fy} are the friction slopes in the x and y directions, u and v are the velocities in x and y direction, respectively, and n is the roughness coefficient.

The momentum equations for the x and y directions are given by the following equations when the momentum equations using the water depth (Eq. (1.2) and (1.3)) are rewritten using the water level and the Manning's equation (Eq. (1.4) and Eq. (1.5)).

$$\frac{\partial q_x}{\partial t}+\frac{gh\,\partial z}{\partial x}+\frac{gn^2q_x^2}{h^{7/3}} = 0 \tag{1.6}$$

$$\frac{\partial q_y}{\partial t}+\frac{gh\,\partial z}{\partial y}+\frac{gn^2q_y^2}{h^{7/3}} = 0 \tag{1.7}$$

Here, z is a water level $(h+b)$ and b is a ground elevation.

1.3 Discretization of governing equations

G2D spatially discretizes governing equations using finite volume method. The position of a control volume is indicated by the subscripts (i, j) of the variable. The center of the control volume is p, the left side of the control volume is w ($-x$ direction), the right side of the control volume is e ($+x$ direction). The upper side of the control volume is indicated by n($-y$ direction), and the lower side of the control volume is indicated by s($+y$ direction). Integrating the continuity equation for the control volume $CV_{i,j}$ with respect to the terms x, y and time and then creating the discrete equation is as shown in equation (1.8). Here, the implicit method was applied to the temporal discretization.

The momentum equation is calculated as a staggered mesh using one control volume $CV_{i,j}$ and an adjacent test volume (for example, the control volume adjacent to the right is $CV_{i+1,j}$). The equation for calculating the flow in $CV_{i,j}$ to e direction is as shown in equation (1.9), and the equation can be written in the same way for w, s, and n directions.

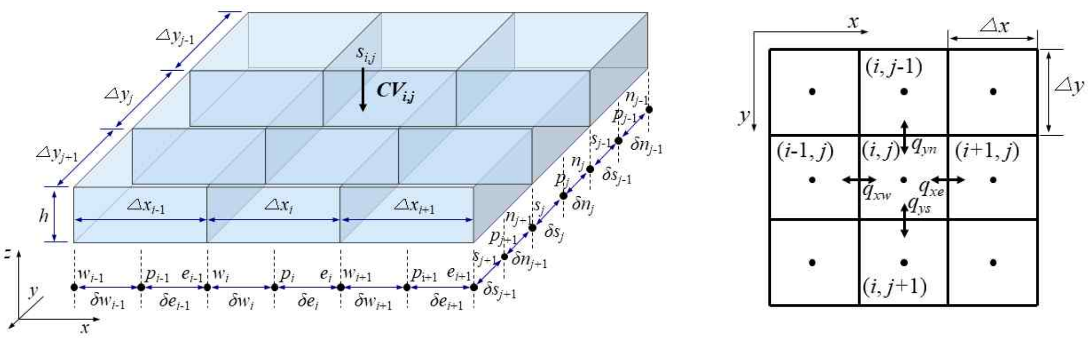


Fig. 1.2 Control volume setting for spatial discretization

$$h_{i,j}^{t+\Delta t} - h_{i,j}^{t} + \{q_{xe} - q_{xw}\}^{t+\Delta t}\frac{1}{\Delta x}\Delta t + \{q_{ys} - q_{yn}\}^{t+\Delta t}\frac{1}{\Delta y}\Delta t - s_{i,j}^{t+\Delta t}\Delta t = 0 \quad (1.8)$$

Here, $s_{i,j}$ is source term and Δt is calculation time interval.

$$q_{ei}^{t+\Delta t} - q_{ei}^{t} + \frac{g(h_f)_i \Delta t (z_{i+1} - z_i)^{t+\Delta t}}{\Delta x} + \frac{gn^2 \Delta t\, q_{xi}^{t+\Delta t}\left|q_{xi}^{t}\right|}{(h_f)_i^{7/3}} = 0 \quad (1.9)$$

Here, h_f represents the flow depth between the two control volumes. The height of the control volume bottom (ground) of a square grid based model is not continuous but has discontinuous values from the DEM elevation. Therefore, when calculating the flow between adjacent two grids *A* and *B*, the depth of the flow is set as follows using the ground elevation and the water level of each grid.

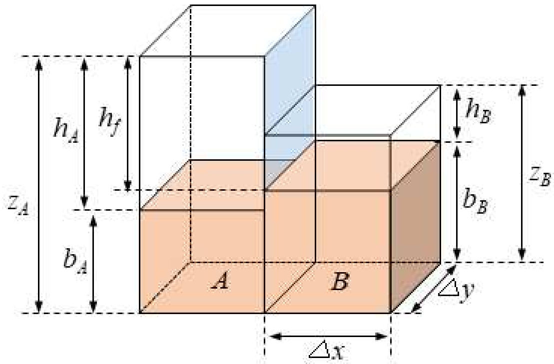


Fig. 1.3 Relationship between depth, water level, ground level, and flow depth

$$h_f = (|z_{A,B}|_{\max} - |b_{A,B}|_{\max}) \tag{1.10}$$

The calculation time step is set using the following CFL (Courant-Friedrichs-Lewy) condition during the simulation.

$$\Delta t \leq \frac{\Delta x}{V_{\max}} \tag{1.11}$$

Where C is Courant number, $V_{\max}$ is the maximum value of the velocity in the whole grid computed at time t, and Δt is the computation time interval at $t + \Delta t$ time. In addition, the von Neumann stability condition could be applied to increase the stability of numerical solution, and the calculation time step is set as follows.

$$\Delta t = \frac{\Delta x^2}{4} min\left(\frac{2n}{h_f^{5/3}} \left| \frac{\Delta z}{\Delta x} \right|^{1/2}, \frac{2n}{h_f^{5/3}} \left| \frac{\Delta z}{\Delta y} \right|^{1/2} \right) \tag{1.12}$$

2. Land cover parameter

The land cover is used to set the roughness coefficient for the calculation of the land surface flow. In the G2D, the roughness coefficient of the land surface and channel can be applied by referring to the roughness coefficient suggested by Chow (1959), Engman (1986), Vieux (2004), Choi and Kim (2018), etc. The impervious area on the surface means the area where water on the surface does not penetrate into the soil. Since the G2D model does not simulate the infiltration of water into the ground, it is applied to the impervious area, regardless of the land cover.

3. Input data

G2D runs on a project-by-project basis. Project files of the G2D model have a .g2p extension and are saved in xml format. The G2D project file stores input data, simulation environment variables, and model parameters required for the execution of the G2D model.

G2D can set the domain using DEM and set the roughness coefficient for each grid using the land cover map. The water level and depth can be set as initial conditions, and the water level, depth, and discharge can be input as boundary conditions. In addition, rainfall data can be applied for flood analysis.

DEM, land cover, and distributed rainfall are entered in files with ASCII raster format. Boundary conditions (water level, depth, discharge) and mean rainfall are entered as text files. Initial conditions (water level, depth) are entered in ASCII raster format file or text file.

Table 3.1 G2D model input data

<table>
<tr><th>Classific-ation</th><th>Input data</th><th>File format</th><th>Remarks</th></tr>
<tr><td>Topogra-phy</td><td>DEM</td><td rowspan="2">ASCII raster</td><td>Configuring the simulated domain</td></tr>
<tr><td>Land cover</td><td>Land cover map</td><td>Setting the roughnesses coefficient for all grids included in the domain</td></tr>
<tr><td rowspan="7">Hydrol-ogical data</td><td rowspan="2">Rainfall</td><td>ASCII raster</td><td rowspan="2">Distributed rainfall in ASCII raster format or domain mean rainfall time series in text file format. Optional use of two</td></tr>
<tr><td>Text file</td></tr>
<tr><td>Discharge</td><td>Text file</td><td>Boundary condition</td></tr>
<tr><td rowspan="2">Depth</td><td>Text file</td><td>Boundary condition or Initial condition</td></tr>
<tr><td>ASCII raster</td><td>Boundary condition</td></tr>
<tr><td rowspan="2">Water level</td><td>Text file</td><td>Boundary condition or Initial condition</td></tr>
<tr><td>ASCII raster</td><td>Initial condition</td></tr>
</table>

3.1 DEM

G2D sets the flood analysis domain using DEM. Therefore, a DEM should be used that reflects the ups and downs of the terrain that may affect the spread of floods. For this purpose, it is recommended to use high-resolution DEM with high accuracy. However, when using such a high-resolution DEM, the number of grids included in the domain increases, so that the calculation time becomes long. Therefore, the resolution of the DEM should be determined by considering both the computation time and the reflection of the terrain relief. If a low-resolution DEM is used, the elevation of the DEM should be modified to reflect the ups and downs of the terrain, which can have a significant impact on the flood propagation, such as the embankment and roads. DEM uses the ASCII raster format.

3.2 Land cover

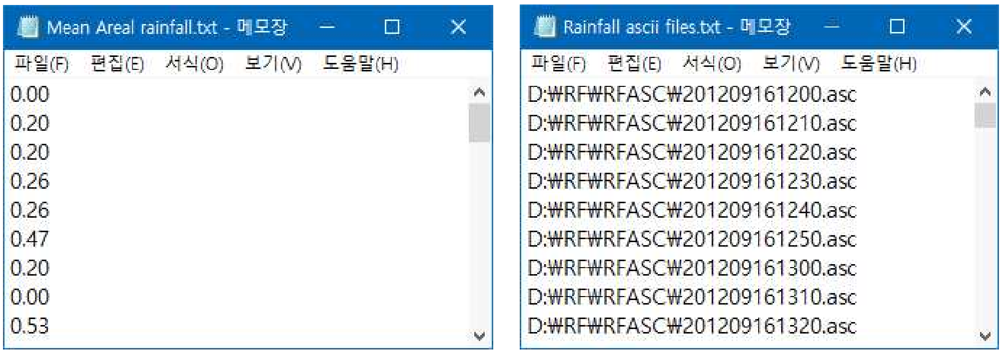
The surface roughness coefficient used in flood analysis is set in two ways. One method is to set different roughness coefficients for each grid using a land cover map, and the other is to use one roughness coefficient factor in the whole grid. The land cover map uses an ASCII raster file format with an integer land cover attribute. The roughness coefficient corresponding to each land cover attribute is applied to the G2D model through a VAT (Value Attribute Table) text file. The VAT file has three values (land cover value in raster file, land cover name, and roughness coefficient), and each value is separated by a comma (,).

Fig. 3.1 An example of land cover VAT file

3.3 Hydrological data

3.3.1 Rainfall

The G2D model can use rainfall in the domain as source term. The rainfall data can be a text file that stores the mean rainfall in the domain area, or an ASCII raster file with different rainfall for each grid. When applying the mean rainfall, the text file storing the rainfall value is used. When the ASCII raster rainfall is applied, the text file storing the list of the raster file is used. Since the G2D model only simulates surface flow, all applied rainfall is simulated as direct runoff (ie, no infiltration or evapotranspiration is calculated).



(b) Mean rainfall (c) ASCII raster file rainfall

Fig. 3.2 Examples of rainfall input files

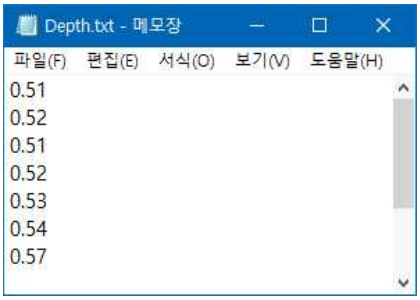
3.3.2 Initial condition

For all grids included in the domain, it is possible to set the initial water level or initial water depth condition at the starting time of flood simulation. The initial condition can apply the same value to all grids using one water level or depth value, or apply different values to each grid using ASCII raster file.

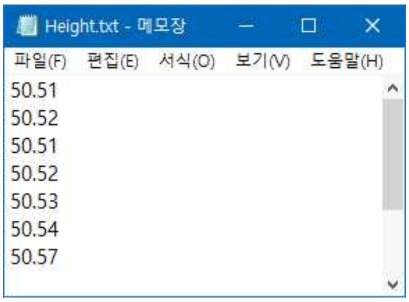
3.3.3 Boundary condition

Boundary conditions can be set for one or more grids in the domain using water level, depth, and discharge time series data. If discharge is used as a boundary condition, the discharge data is applied to the calculation as source term at the specified grid.

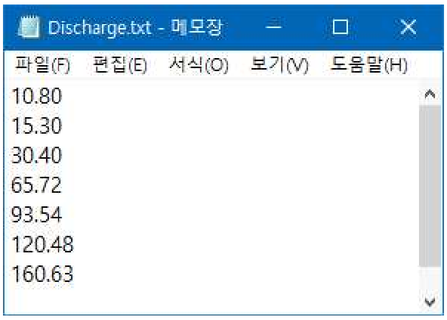
When a high-resolution DEM (DEM with a small grid size) is used, a large value of the boundary condition increases the depth value of the grid. In this case, since the depth discontinuity between the grid which the boundary condition was entered and the surrounding grid is large, many iterations are required to converge the numerical solution, or the accuracy of the calculated value may be lowered. Therefore, when applying a large value boundary condition in a high-resolution DEM, it is necessary to divide it into several grids. The G2D model automatically distributes the boundary condition values to the specified grids automatically when multiple grids are specified for one boundary condition file.



(a) Water depth



(b) Water level



(c) Discharge

Fig. 3.3 Examples of boundary condition input files

3.4 G2D project file

G2D is executed using the project file (.g2p). To run G2D in the console window, set a g2p file as an argument.

(For example, C:₩G2D₩G2D.exe "*projectFilePathAndName*.g2p")

The g2p file is saved in xml format. The tables included in the g2p file, the contents of each table, the field names and descriptions of each table, and how to set each field are shown in the following table. A g2p file can be created by a user using a text editor, but some items (such as grid positions) are difficult to obtain intuitively. Therefore, when creating a g2p file for the first time, it is convenient to automatically generate it using the GUI software of the G2D model (QGIS plug-in, etc.).

If the g2p file contains Korean, it should be saved in UTF-8 format. If Korean is not included, it should be saved in ANSI or UTF-8 format.

Table 3.2 Table definition in the project xml file (.g2p)

Table name	Description	Required
ProjectSettings	Setting environment variables, input files, etc. for model execution	○
HydroPars	Hydraulic parameters, initial condition setting	○
BoundaryConditionData	Set boundary conditions for each cell	×
DomainToChange	Set domain DEM file information to be replaced during simulation	×

Table 3.3 ProjectSettings table specification

Field name	Description	Data format	Required
DEMFile	The path and name of the DEM file to use as the simulation domain.	String	Required
LandCoverFile	The path and name of the land cover file.	String	Optional
LandCoverVatFile	The path and name of the VAT file in which the roughness coefficient for each attribute of the land cover file is written.	String	Optional
CalculationTimeStep_sec	Calculation time interval (sec)	Double	Required
IsFixedDT	Whether fixed calculation time interval is used or not (true or false).	String	Required
MaxDegreeOfParallelism	The maximum degree of parallelism using CPU(usually the number of logical processors in the PC is applied. If -1 is applied, the model automatically uses the maximum value. If 1 is applied, parallel calculation is not applied.)	Integer	Required
UsingGPU	Whether the GPU is used (true or false). Can only be true when using NVIDIA graphics cards.	String	Required
ThreadsPerBlock	The number of threads in a thread block using CUDA. Generally, one of 16, 32, 64, 128, 256, or 512 can be used (with latest compute capability and recent devices including Volta after July 2019, 1024 is possible). Only used when UsingGPU=true.	Integer	Optional
MaxIterationAllCells	The maximum number of iterations for the convergence of the entire domain. (Typically between 5 and 20)	Integer	Required
MaxIterationACell	The maximum number of iterations for the convergence of a cell. (Typically between 5 and 20)	Integer	Required
StartDateTime	The starting time of the simulation. If the date and time format is set, enter it in DateTime format (e.g. 2012-09-16 12:00). If not, use 0	String	Required
SimulationDuration_hr	Simulation duration (hours)	Double	Required
PrintoutInterval_min	Output time interval (minutes)	Integer	Required
RainfallDataType	Rainfall data type (TextFileMAP or TextFileASCgrid)	String	Optional
RainfallDataInterval_min	Rainfall data time interval (minutes)	Integer	Optional
RainfallFile	The path and name of the file where the rainfall data is stored	String	Optional
InitialRainfallLoss_mm	Initial rainfall loss(mm)	Double	Optional

<ProjectSettings table specification (continued)>

Field name	Description	Data format	Required
BCDataInterval_min	Boundary condition data time interval (minutes)	Integer	Optional
FloodingCellDepthThresholds_cm	Depth threshold value to log the information of each flooding depth(cm). Can be set one or more number of values by using comma(,). If not set, 0.001cm is applied.	String	Optional
CellLocationsToPrint	The location of cells to print the time series of flow component. Numbering from upper left corner (0,0). The maximum value is (the number of columns - 1, the number of rows – 1). Separate multiple cells with "/". (E.g., 5, 10 / 125, 320 / 126, 320)	String	Optional
MakeASCFile	Whether to output ASCII files (true or false)	String	Required
MakeImgFile	Whether to output image files (true or false)	String	Required
OutputDepth	Whether to output water depth (true or false)	String	Required
OutputPrecision_Depth	Decimal digit of printing water depth	Integer	Required
DepthImgRendererMaxV	Maximum value of renderer in water depth image file	Double	Required
OutputWaterLevel	Whether to output water level (true or false)	String	Required
OutputPrecision_WaterLevel	Decimal digit of printing water level	Integer	Required
WaterLevelimgRendererMaxV	Maximum value of renderer in water level image file	Double	Required
OutputVelocityMax	Whether to output flow velocity (true or false)	String	Required
OutputPrecision_VelocityMax	Decimal digit of printing flow velocity	Integer	Required
VelocityMaxImgRendererMaxV	Maximum value of renderer in flow velocity image file	Double	Required
OutputDischargeMax	Whether to output discharge (true or false)	String	Required
OutputPrecision_DischargeMax	Decimal digit of printing discharge	Integer	Required
DischargeImgRendererMaxV	Maximum value of renderer in discharge image file	Double	Required
OutputFDofMaxV	Whether to output flow direction (true or false)	String	Required
WriteLog	Whether to log (true or false)	String	Required

Table 3.4 HydroPars table specification

Field name	Description	Data format	Required
RoughnessCoeff	Roughness coefficient of domain surface (If no land cover is entered, this value is used in the calculation)	Double	Optional
DomainOutBedSlope	Ground slope value of the cell in domain boundary toward domain outside	Double	Required
InitialConditionType	Initial condition type (Depth or WaterLevel) If not set, the initial condition is not applied.	Double	Optional
InitialCondition	The path and name of the ASCII raster file in which the initial cell condition is recorded (m).	String	Optional
	Initial condition value to apply to all cells with the same value (m).	Double	Optional
FroudeNumberCriteria	Upper limit of Froude number If not greater than 1, simulate all flows as a sub-critical flow	Double	Required
CourantNumber	Uppper limit of Courant number Greater than 0, less than or equal to 1 Typically, apply the ranges of 0.5~0.7.	Double	Required
ApplyVNC	Whether to apply the Von Neuman condition when calculating dt (true or false)	String	Required

Table 3.5 BoundaryConditionData table specification

Field name	Description	Data format	Required
CellXY	The location of the cells to enter the boundary conditions. Numbering from upper left corner (0,0). The maximum value is (the number of columns - 1, the number of rows – 1). Separate multiple cells with "/". (E.g., 5, 10 / 125, 320 / 126, 320)	String	Required
DataType	Boundary condition type. Select one of Discharge, Depth, and WaterLevel	String	Required
DataFile	Path and name of boundary condition file	String	Required

Table 3.6 DEMFileToChange table specification

필드 명	Description	Data format	Required
TimeMinute	Time to replace DEM (min) Elapsed time since modeling started	Integer	Required
DEMFile	Path and name of DEM file	Integer	Required

4. Output data

The G2D model computes the depth and velocity of a grid which has water depth (effective grid) among the grids contained in the domain. And the results are stored in ASCII raster files or image files. The log file also records the computation time interval (dt), the time used for the calculation, the number of effective grids, and the number of grids per specified depth. Simulation results can be selectively output.

Table 4.1 G2D output file

Output files	Remarks
*[Project name]*_Depth_*[data time]*.out	Water depth (m)
*[Project name]*_WaterLevel_*[data time]*.out	Water level (m)
*[Project name]*_Discharge_*[data time]*.out	Discharge (㎥/s)
*[Project name]*_Velocity_*[data time]*.out	Maximum flow velocity in the four-direction flow of a grid (m/s)
*[Project name]*_FDirection_*[data time]*.out	The flow direction of maximum velocity (1 : east, 3 : south, 5: west, 7 : north)
*[Project name]*_*[Flux]*_CellValue.csv	Time series of flow components for specified cells. (Flux : water depth, water level, max discharge, max velocity, flow direction of max velocity)
[Project name].log	Simulation log file. Calculation time interval (dt), time used for calculation, number of effective grids, number of grids per specified depth, etc.

References

Choi, Y.S., Kim, K.T. 2019. Grid based Rainfall-runoff Model User's Manual. Korea Institute of Civil Engineering and Building Technology, pp. 14-15.

Chow, V.T. 1959. Open-channel hydraulics. McGraw-Hill, pp. 101-123.

Engman, E.T. 1986. Roughness coefficients for routing surface runoff. Journal of Irrigation and Drainage Engineering, 112(1), pp. 39-53.

Vieux, B.E. 2004. Distributed Hydrologic Modeling Using GIS. Kluwer Academic Publishers.

Appendix

A. Example of G2D project file


```xml
<?xml version="1.0" standalone="yes"?>
<projectds xmlns="http://tempuri.org/projectds.xsd">
  <ProjectSettings>
    <DEMFile>C:₩G2D₩SampleData₩domain₩DEM_JHW_10m.asc</DEMFile>
    <LandCoverFile>C:₩G2D₩SampleData₩domain₩LC_JHW.asc</LandCoverFile>
    <LandCoverVatFile>C:₩G2D₩SampleData₩domain₩LC_JHW.VAT</LandCoverVatFile>
    <CalculationTimeInterval_sec>1.0</CalculationTimeInterval_sec>
    <IsFixedDT>false</IsFixedDT>
    <MaxDegreeOfParallelismCPU>18</MaxDegreeOfParallelismCPU>
    <UsingGPU>false</UsingGPU>
    <ThreadsPerBlock>512</ThreadsPerBlock>
    <MaxIterationAllCells>7</MaxIterationAllCells>
    <MaxIterationACell>5</MaxIterationACell>
    <SimulationDuration_hr>25</SimulationDuration_hr>
    <PrintoutInterval_min>60</PrintoutInterval_min>
    <StartDateTime>0</StartDateTime>
    <RainfallDataType>TextFileMAP</RainfallDataType>
    <RainfallDataInterval_min>10</RainfallDataInterval_min>
    <RainfallFile></RainfallFile>
    <InitialRainfallLoss_mm>0</InitialRainfallLoss_mm>
    <BCDataInterval_min>60</BCDataInterval_min>
    <FloodingCellDepthThresholds_cm>1, 10, 30, 50</FloodingCellDepthThresholds_cm>
    <CellLocationsToPrint>179,762 / 283,641 / 22, 289</CellLocationsToPrint>
    <OutputDepth>true</OutputDepth>
    <OutputPrecision_Depth>5</OutputPrecision_Depth>
    <OutputWaterLevel>false</OutputWaterLevel>
    <OutputPrecision_WaterLevel>5</OutputPrecision_WaterLevel>
    <OutputVelocityMax>false</OutputVelocityMax>
    <OutputPrecision_VelocityMax>5</OutputPrecision_VelocityMax>
    <OutputDischargeMax>false</OutputDischargeMax>
    <OutputPrecision_DischargeMax>2</OutputPrecision_DischargeMax>
    <OutputFDofMaxV>false</OutputFDofMaxV>
    <DepthImgRendererMaxV>3</DepthImgRendererMaxV>
    <WaterLevelimgRendererMaxV>200</WaterLevelimgRendererMaxV>
    <VelocityMaxImgRendererMaxV>10</VelocityMaxImgRendererMaxV>
    <DischargeImgRendererMaxV>10000</DischargeImgRendererMaxV>
    <MakeASCFile>true</MakeASCFile>
```

```xml
        <MakeImgFile>false</MakeImgFile>
        <WriteLog>false</WriteLog>
    </ProjectSettings>
    <HydroPars>
        <RoughnessCoeff>0.03</RoughnessCoeff>
        <DomainOutBedSlope>0.001</DomainOutBedSlope>
        <InitialConditionType>Depth</InitialConditionType>
        <InitialCondition>0</InitialCondition>
        <FroudeNumberCriteria>1</FroudeNumberCriteria>
        <CourantNumber>0.6</CourantNumber>
        <ApplyVNC>false</ApplyVNC>
    </HydroPars>
    <BoundaryConditionData>
        <CellXY>162, 778 / 163, 778 / 164, 778 / 165, 778 / 166, 778</CellXY>
        <DataFile>C:₩G2D₩SampleData₩BCdata₩jhw_main_discharge_cms_dt60min.txt</DataFile>
        <DataType>Discharge</DataType>
    </BoundaryConditionData>
    <BoundaryConditionData>
        <CellXY>341,485 / 341,486 / 341,487</CellXY>
        <DataFile>C:₩G2D₩SampleData₩BCdata₩jhw_Oh_discharge_cms_dt60min.txt</DataFile>
        <DataType>Discharge</DataType>
    </BoundaryConditionData>
    <BoundaryConditionData>
        <CellXY>1,294 / 1,295 / 1,296</CellXY>
        <DataFile>C:₩G2D₩SampleData₩BCdata₩jhw_Sul_discharge_cms_dt60min.txt</DataFile>
        <DataType>Discharge</DataType>
    </BoundaryConditionData>
    <DEMFileToChange>
        <TimeMinute>600</TimeMinute>
        <DEMFile>C:₩G2D₩SampleData₩domain₩DEM_JHW_10m_BP1.asc</DEMFile>
    </DEMFileToChange>
</projectds>
```

B. Executable file and sample data

The G2D model does not require a software installation process because it simply copies the exe file to be used.

1. Download the G2D.exe and sample data (SampleData.zip) from

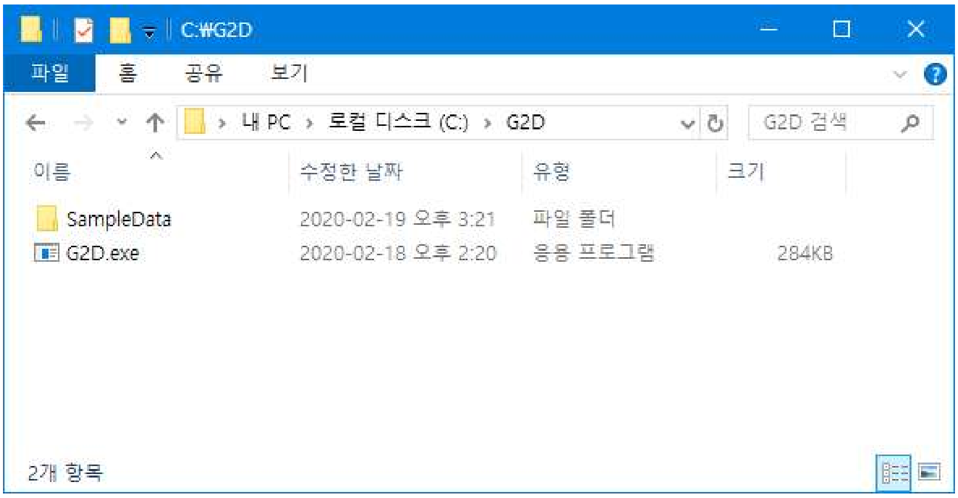
   https://github.com/floodmodel/G2D/tree/master/DownloadStableVersion

   *** To use G2D v.2021 and later, CUDA tool kit 11.2 or later has to be installed.*

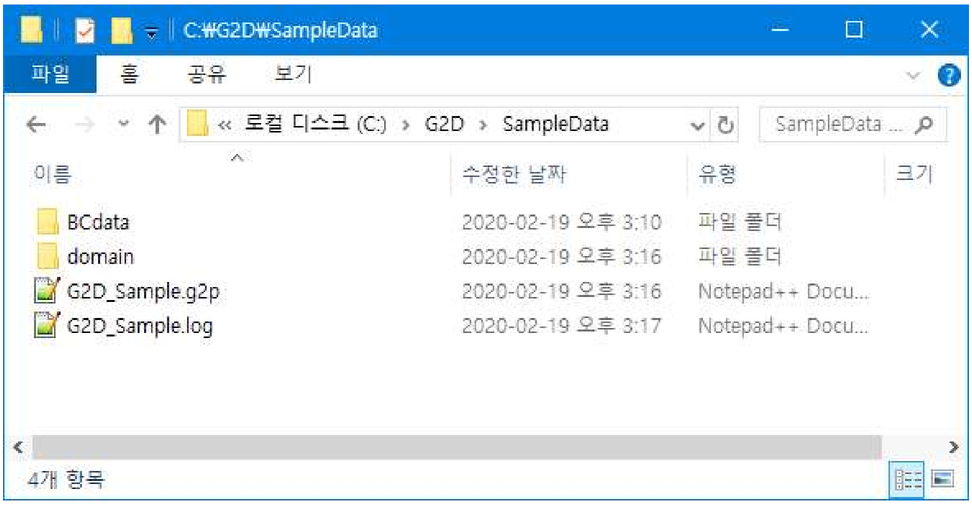
2. In Windows 10 or higher OS, users have to check on 'Unblock' in file property window the downloaded executable files (exe, zip, etc) from web.

3. In the case of allocating the G2D files in "C:₩G2D," the folder and files are as the shown below.

4. The 'SampleData' folder contains sample project and data to run the G2D. When the G2D files are in an other folder, the file paths in the g2p file (~~₩SampleData₩G2D_Sample.g2p) have to be changed.



<Executable and sample files placed in 'C:₩G2D'>



<'C:₩G2D₩SampleData' folder contents>

5. When the simulation using CUDA (UsingGPU option is set as 'True') is applied and if an error related with CUDA is occurred, locate "cudart64_110.dll" file in the same folder with G2D.exe file.

C. How to run the G2D (Console window)

To run the G2D, either the menu in a modeling softwares (QGIS-G2D) can be used or the user can manually run it in a console window. In the console window, run G2D.exe with the G2D model project file (.g2p) as an argument as shown below.

C:₩G2D₩G2D.exe "*projectFilePathAndName*.g2p"

For example,

The execute statement when the G2D.exe file is in the 'C:₩G2D' folder and the G2D_Sample.g2p file is in the 'C:₩G2D₩SampleData' folder is as follows:

C:₩G2D>G2D.exe C:₩G2D₩SampleData₩G2D_Sample.g2p

If there are spaces in the project file name or path, quotation marks "" are used to enclose it for input.

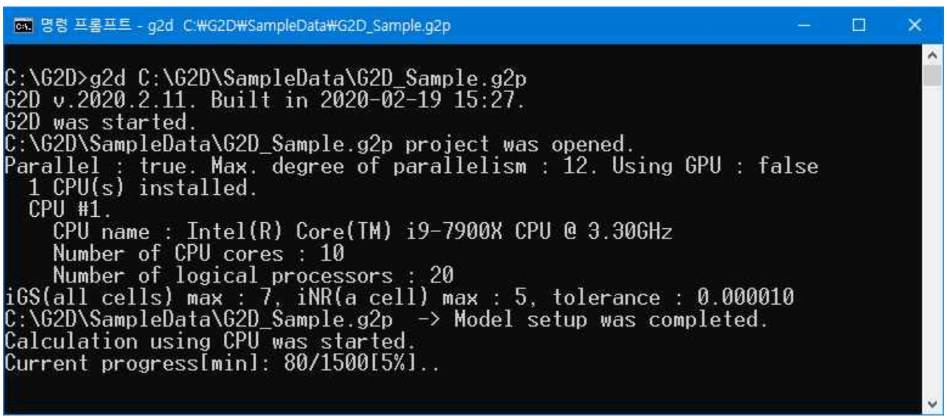
C:₩G2D>G2D.exe "C:₩G2D₩SampleData₩G2D Sample.g2p"

When the G2D.exe and g2p files are in the same folder, the project file path does not have to be entered. Thus, the following example shows how to run it when the G2D.exe and g2p files are in the 'C:₩G2D' folder.

C:₩G2D>G2D.exe G2D_Sample.g2p

"/?" can be entered to seek help.

C:₩G2D>G2D.exe /?



<Running window of the G2D model>