

HEC-HMS Lab 1: Single Basin HEC-HMS Modeling

Created by Venkatesh Merwade (vmerwade@purdue.edu)

Learning outcomes

The objective of this lab is to explore the basic structure of HEC-HMS modeling system, and create a simple single basin model to simulate a rainfall event. By the end of this exercise, you will be able to:

- 1) Create a new HEC-HMS project with a single sub-basin model
- 2) Understand the inputs and outputs associated with HEC-HMS model
- 3) Run a simple HEC-HMS simulation and interpret the model's output
- 4) Know the meaning and use of different files created by HEC-HMS. Specifically, the .hms file, .basin file, .met file and the control file

Student Time Required: 50 minutes during lab + 50-100 minutes for HW.

Data Inputs

Numerical Inputs

Rainfall : Hourly Rainfall Hyetograph in mm
Watershed Area: 100 km²
Curve Number : 75
Lag time: 60 minutes
Simulation Time step: 10 minutes
Simulation Start Time: 01/24/2020 09:00 AM
Simulation time period: 1-3 days

Text Inputs

Loss Method: SCS
Transform Method: SCS
Baseflow Method: None

Model: HEC-HMS 4.3 (Note: You should be able to do this exercise by using any version of HEC-HMS, but the images in this document may not match exactly with what you see on your screen)

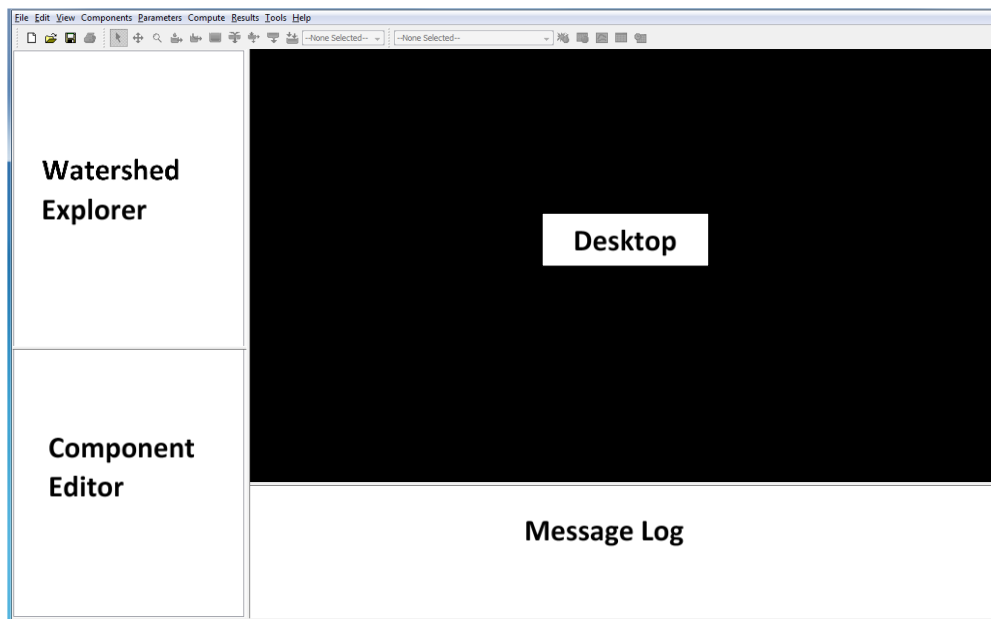
Instructions

Start HEC-HMS

In most typical installations on windows system, you go to Start→Programs→HEC→HEC-HMS 4.x. This path may be different in computer labs. Just search for HEC-HMS and open the program.

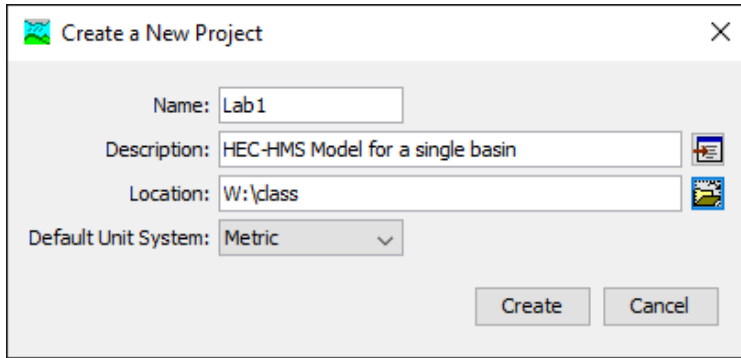
Program Screen

The HEC-HMS interface consists of a menu bar, tool bar, and four panes. These panes are referred to as the Watershed Explorer, the Component Editor, the Message Log and the Desktop as shown below. More description on these panes, menus and tools is provided when they are used later in the exercise.

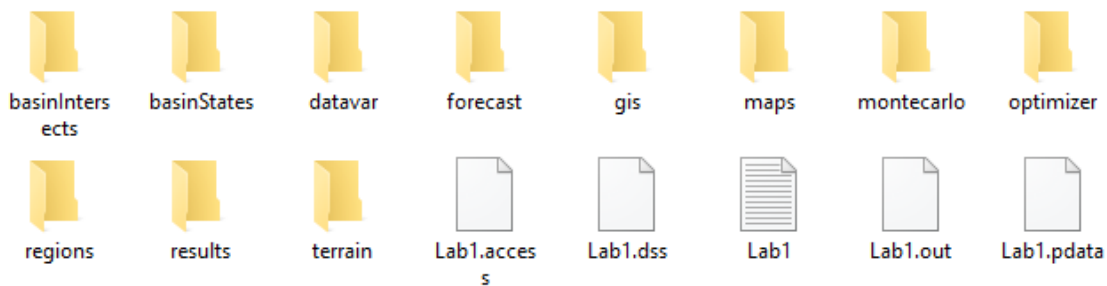


Creating and Saving a new HEC-HMS project

On the main Menu, click on File→New.. First specify the location of where you want to save your project, give a name (Lab1), accept the default unit system (Metric), and click on Create. Note that the unit system may be different for different projects.



After you create a project, go to the directory where you saved the project using windows explorer, and you will see that the following folders and files are created.



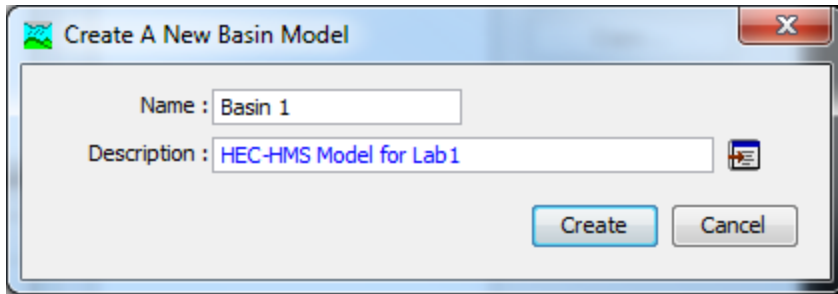
Go ahead, and open the .hms (Lab1.hms) file using Notepad. You will see that it is a simple text file with some basic information about the version of the model. Similarly, all the messages you see in the message box in the HEC-HMS program are saved in text file (lab1.txt in this case).

Now that we have created a project, lets populate this project with the required elements to create a simple hydrologic model.

Creating a Basin Model

A basin model represents the physical description of the watershed in a HEC-HMS project. First, an empty basin model is created, and then it is populated with all the necessary information needed to create the complete description of a watershed. In addition to including the physical description, a basin model also includes information on the mathematical methods (or equations) that will be used in simulating the hydrology of the basin, and the values for all the variables in those equations. The variables in all the equations are called parameters because by changing the values for these parameters we can change the output from the model.

To create a basin model, click on Components → Basin Model Manager.. Then click on New, and enter the name of the basin and some basic description as shown below, and click Create.




You will see that a folder called Basin Models is now added in the Watershed Explorer pane, expand the folder by clicking on the + sign, and you will see Basin 1. Click (or double click if single click does not work) on Basin 1, and an empty basin (white window) will be added to the desktop pane. Save your project.

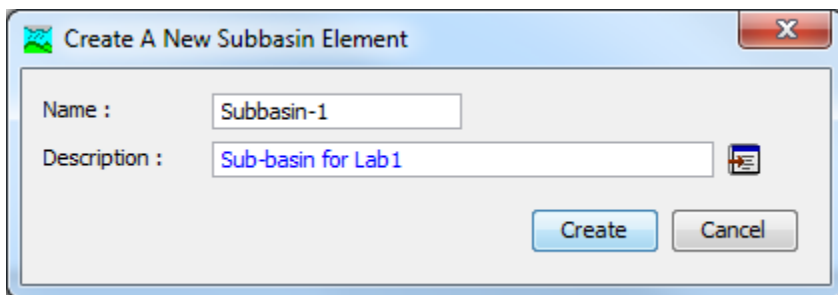
After a basin is created, all the tools (see below) necessary to populate the basin will also become available.



In the above figure, these elements (from left to right) can be used to add sub-basin, reach, reservoir, junction, diversion, source and sink, respectively. In this lab, we will create a single sub-basin that has an area of 100 km².

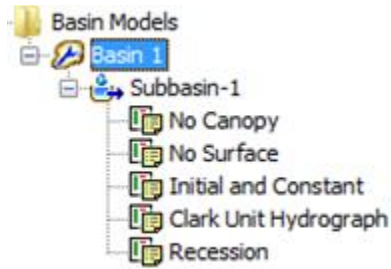
Create Sub-basin

This is done by using the create sub-basin tool (). Select the sub-basin tool, and click on the basin model (white window in the desktop pane). You will be prompted to enter the name and description of the new sub-basin as shown below.



Click Create. You will see that your basin model now has a sub-basin and all the information related to this sub-basin can be entered in the component editor. A sub-basin is just one of the components of your basin model.

Expand Basin 1 and Sub-basin 1 in the watershed explorer, and you will see the following (or a variation of it):



This information can be edited by using the component editor, and each label above is related to a “method” in the basin model. You can think of a method as a hydrologic process, and then you have to choose which equation you want to use to simulate this hydrologic process. The Canopy method allows users to model the interception and evapotranspiration in the watershed. For this lab, we will ignore this method. The surface method allows users to represent depression storage in the watershed. Again, we will ignore surface method for this lab. The next method allows users to represent losses (water lost due to infiltration) from rainfall. We know that not all rainfall becomes runoff because some of it is lost to infiltration. The loss method lets users to model this infiltration by using various equations. We have not covered this topic in the class yet, but for completing this lab, just change the loss method from the default “Initial and Constant” to “SCS Curve Number” for now by using the component editor as shown below.

The transform method allows users to convert the rainfall (after subtracting losses) to streamflow hydrograph. Change the transform method to SCS Unit Hydrograph. Baseflow method allows users to account for existing flow conditions in the outlet stream. Again, we will ignore baseflow for this lab.

Finally, enter the area of the watershed (100 km²) in the area box of the component editor. If you see input for latitude, longitude, etc., simply ignore those for now. The exact placement of this sub-basin is not important for this lab.

Basin Name:	Basin 1
Element Name:	Subbasin-1
Description:	Sub-basin for Lab 1
Downstream:	--None--
*Area (KM2)	100
Latitude Degrees:	
Latitude Minutes:	
Latitude Seconds:	
Longitude Degrees:	
Longitude Minutes:	
Longitude Seconds:	
Canopy Method:	--None--
Surface Method:	--None--
Loss Method:	SCS Curve Number
Transform Method:	SCS Unit Hydrograph
Baseflow Method:	--None--

By specifying names for each method, we are basically telling the program which equation to use. To execute these equations, we need to also specify values for all the variables (parameters) needed. The values for these parameters are provided in the additional tabs that you see in the component editor for each method. Because we only chose two methods (Loss and Transform), you will see that there is a tab for each method in the component editor. Click on the Loss tab, and enter a value of zero for initial abstraction, 75 for curve number and zero for impervious cover. We will learn about these variables later in the semester.

Subbasin	Loss	Transform	Options
Basin Name: Basin 1			
Element Name: Subbasin-1			
Initial Abstraction (MM)	0		
*Curve Number:	75		
*Impervious (%)	0.0		

In the transform tab, enter a value of 60 min for lag time.

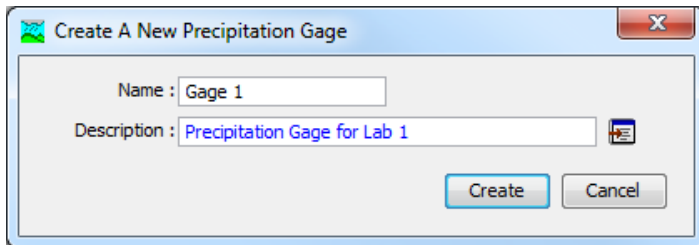
Subbasin	Loss	Transform	Options
Basin Name: Basin 1			
Element Name: Subbasin-1			
Graph Type:	Standard (PRF 484) ▼		
*Lag Time (MIN)	60		

Do not input any information for options. Select "None" for Canopy, Surface and Baseflow methods. Save your project. You have now entered all the necessary information needed to represent a watershed in the basin model.

Go back to windows explorer, and open the Basin1.basin file in notepad to see how all the information in the basin model is stored in a simple text file. Now that we have a basin model, we need some input to drive the simulation. The primary driver of a hydrologic event is precipitation. Note that precipitation/rainfall are typically recorded using weather stations or rainfall gauges. To provide rainfall input, we first have to create a precipitation gage.

Creating a Precipitation Gage

To create a precipitation gage, go to Components Editor→Time Series Data Manager. Select Data Types as precipitation gages, and click on New.

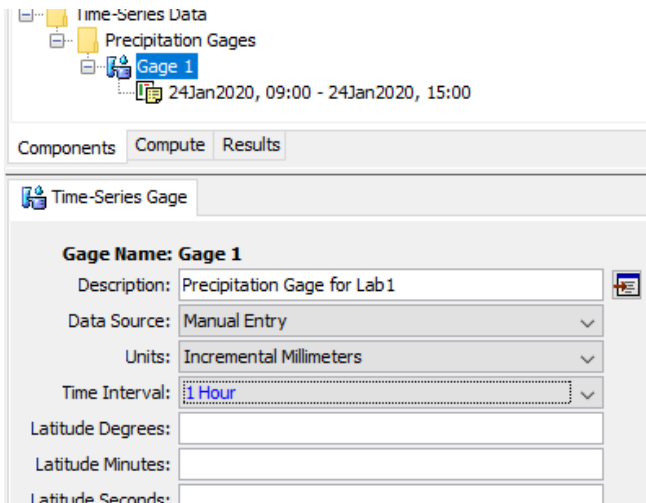


Use the default name and provide some basic description for the gage. Click Create.

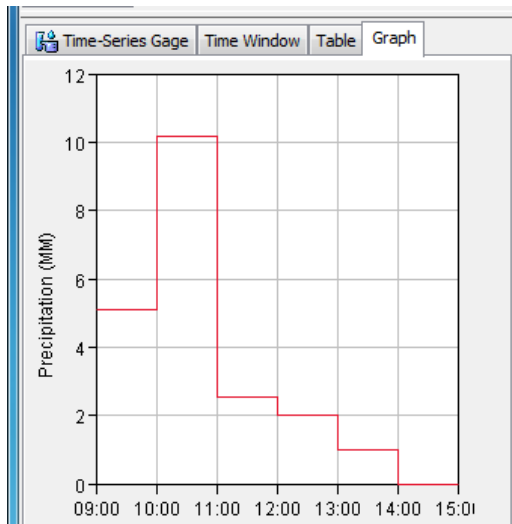
In the watershed explorer, you will see that a new folder called Time Series Data is created. Expand this folder to see another folder called Precipitation Gages, and then Gage 1 is created inside the Precipitation Gages folder. We will populate this gage with the information presented in the table below. What we have here is hourly rainfall data in mm from 9 AM to 3 PM on Jan 24, 2020. Note that this information is hypothetical!

Date	Time	Precipitation (mm)
1/24/2020	9:00	0.000
1/24/2020	10:00	5.080
1/24/2020	11:00	10.160
1/24/2020	12:00	2.540
1/24/2020	13:00	2.000
1/24/2020	14:00	1.000
1/24/2020	15:00	0.000

Select Gage 1 in watershed explorer, and change the time interval to match the data (one hour incremental precipitation). Next, Click on the time-stamp label under Gage 1 in watershed explorer and enter the Start date/time and end date/time for this gage as shown below in the component editor.



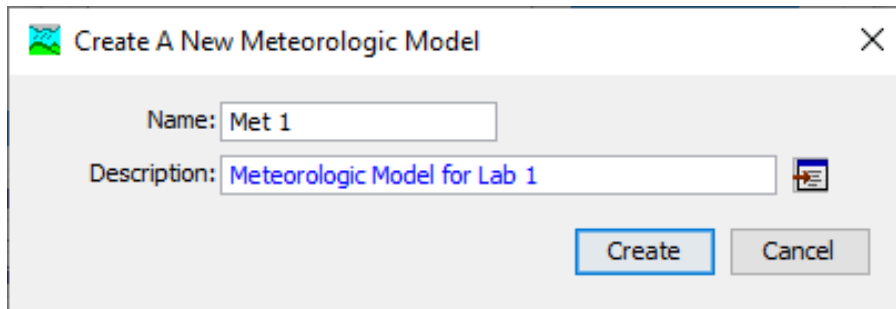
Next, click on the table tab, enter the information from the table above to populate the table, and also see the graph (hyetograph). What you see below is called Rainfall Hyetograph.



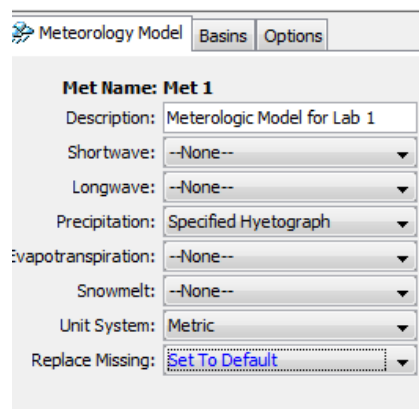
The gage that we just created has the input that we need, but the program does not know that this gage is located in the sub-basin we have created. To link this precipitation to the sub-basin, we need to create a meteorologic model or Meteorologic file. Before you proceed, check that the units for rainfall are correct.

Creating a Meteorologic Model

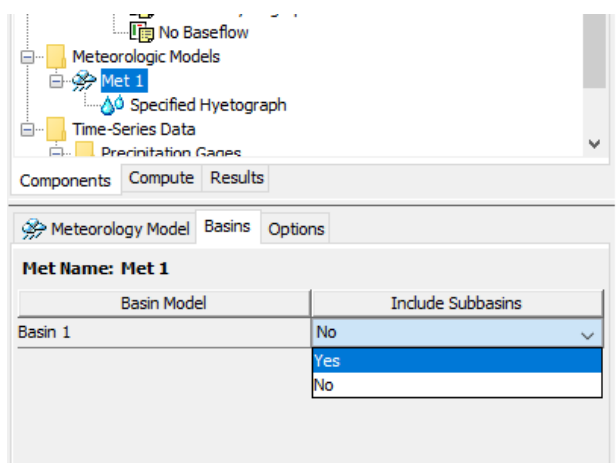
A meteorologic model or file contains meteorologic information to drive the hydrologic simulation. You cannot run a HEC-HMS simulation without a meteorologic model/file. To create a meteorologic model, go to Components → Meteorologic Model Manager, and create New. Use the default name and provide some basis description and click Create.



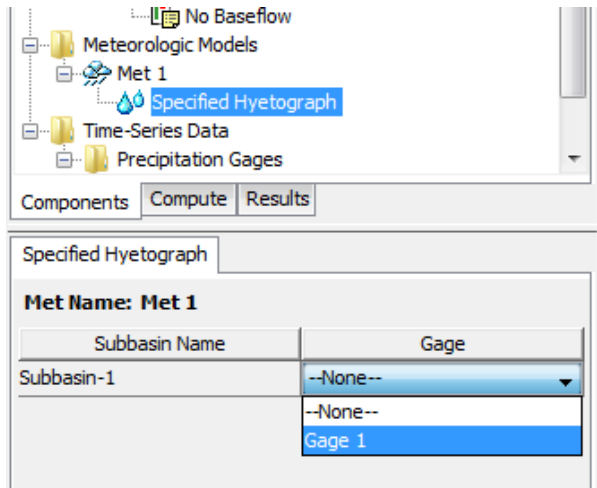
You will see a new folder called Meteorologic Models in the watershed explorer, and if you expand this folder, you will see the Met 1 file that you just created. Under Met 1 label, you will see Specified Hyetograph label. In the Meteorology Model Tab, change the Replace Missing option from “Abort” to “Set to Default” as shown below. The earlier option (abort) will not let the model run if the simulation time is longer than the rainfall duration.



Next, select the Basins tab in the Component editor, and then click Yes on Include sub-basins. This will link Gage 1 to sub-basin 1.



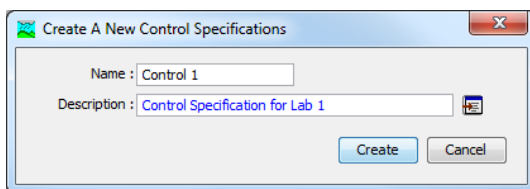
Next, select the specified hyetograph label in watershed explorer, and then select Gage 1 for subbasin-1 in the component editor as shown below. This step will tell the model that Gage 1 data is meant for Sub-basin 1.



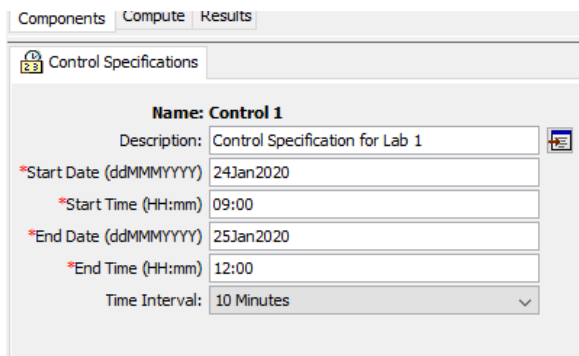
Save your project. Now we have the watershed description and the input information. The next thing is to provide some basic information on the time-step to run the model and the duration because we know that the rainfall occurred from 9 AM to 3 PM, but the water from this event will keep flowing into the river even after 3 PM so we need to specify when to start and end the simulation. This is done by creating a control specification model.

Creating a Control Specification Model

The control specification file basically tells HEC-HMS how long the simulation will last and the time step to continue the simulation. You cannot run a HEC-HMS simulation without creating a control specification file. To create a control specification model, go to Components → Control Specification Manager.. and create New. Accept the default name, enter some basic description, and then click Create.



You will now see that another folder called Control Specifications is now added in the watershed explorer. Expand the folder, select Control 1 and enter the start date/time, end date/time and time interval in the component editor as shown below.



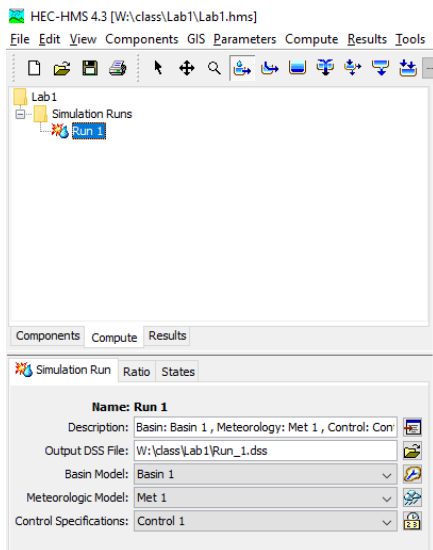
Basically, we will run the model longer than the rainfall event to make sure we account for all the flow that is coming to the outlet from this event. Save your project. Once we have created the basin file, meteorologic file and control specification file for the project, the model is ready for simulation.

Creating a Simulation Run

To create a simulation Run go to Compute→Create Compute→Simulation Run. Accept the default name Run 1 and then accept all the default models and finish creating the run. If you had more than one model for Basin, Meteorology and Control specification, you get to select them during this step. Because we have only one model for each, the process is straight forward. Save the project.

Running a Simulation

Once a simulation run is created, select the simulation by going to the Compute tab in the watershed explorer and select the run as shown below.



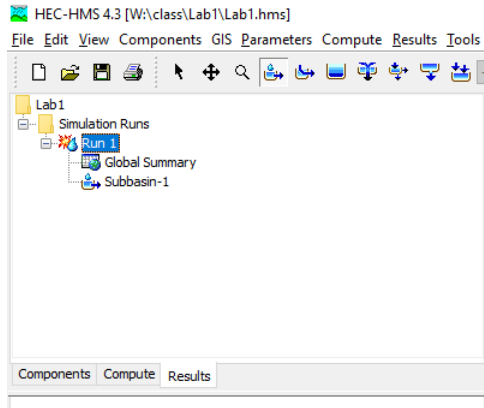
Next, go to the Compute tab on the menu bar, and select Compute Run [Compute→Compute Run].

(Note: The simulation should not take more than a minute to run. If it is taking longer, cancel the simulation and check for the following:

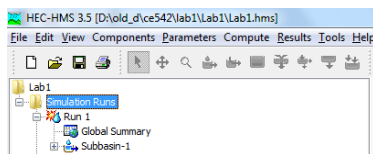
- Your rainfall gage is included in the sub-basin
- Your control specification time period covers the rainfall event
- Your units are correct including the area of the watershed)
- Your replaced missing is set to Default for the Meteorologic model

Looking at Results

To look at the results from simulation, you will use the Results Tab in the Watershed Explorer Window and then select the Run as shown below (the name of the Run for you may be different)



Expand Run1 (or whatever name given by you), and you will see Global Summary and the sub-basin name. If you expand subbasin-1, you will see more options to explore results related to individual component of this sub-basin such as losses and runoff.



Look the global summary and the Summary Table for subbasin-1. Because we have only one basin, the information such as peak discharge and volume for the entire model is same as for Subbasin-1. If we have more than one sub-basins, we will see the final results for the entire watershed in the global summary table and individual results for sub-basins in the Summary Table for each basin.

Turn-in (hard copy) by 1:30 PM on Jan 31, 2020:

Part A

- 1) A plot of rainfall hyetograph. Define what is a rainfall hyetograph.
- 2) Hydrograph of the sub-basin (either take a screen shot of just the hydrograph or create a graph in Excel). Hydrograph is the plot you see by using the Graph option for subbasin-1 in the Results Tab. Looking at this graph, provide your definition or explanation of hydrograph.
- 3) Write some of the differences that you see between the rainfall hyetograph and the discharge hydrograph.
- 4) Peak or maximum discharge in m^3/s from sub-basin 1
- 5) Total duration of the hydrograph (in hours). This is also called the base time. Time when the hydrograph starts and ends with a zero value.
- 6) Water budget for subbasin-1 in mm (how much water did it get from rainfall, how much was lost to infiltration, and how much came out as runoff)
- 7) Explain what you understand by basin model in HEC-HMS
- 8) Explain what you understand by meteorologic model in HEC-HMS
- 9) Explain what you understand by the control specification model in HEC-HMS

10) Look at other results, specifically cumulative precipitation, soil infiltration, excess precipitation, precipitation loss and direct runoff, and briefly describe what you understand by each of these.

Part B

In the Lab 1 HEC-HMS model, create a new precipitation gage and a corresponding new Meteorologic model for each of the rainfall event given below (use the same date as you used before to keep the control specification model same for all simulations):

Event 2

Time	Precipitation (mm)
09:00	0.00
10:00	2.00
11:00	3.08
12:00	5.16
13:00	5.00
14:00	2.54
15:00	2.00
16:00	1.00

Event 3

Time	Precipitation (mm)
09:00	0.00
10:00	1.50
11:00	2.50
12:00	3.54
13:00	4.00
14:00	4.16
15:00	3.08
16:00	2.00

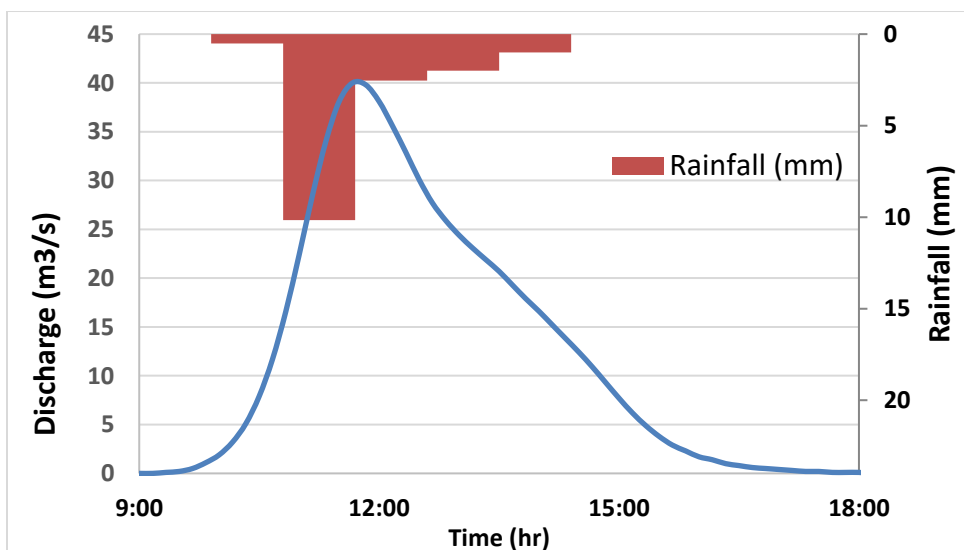
Event 4

Time	Precipitation (mm)
09:00	0.00
10:00	3.00
11:00	3.28
12:00	2.90
13:00	2.90
14:00	2.90
15:00	2.90
16:00	2.90

Copy the rainfall data for Events 2, 3 and 4 from this handout into an Excel file. Save this excel file as lab1.xlsx. Create rainfall hyetographs for each event and past them on a single page in MS Word. Make sure your hyetograph has a title, axes names and units.

Using the lab1 HEC-HMS model, create a new simulation run for each event (check the control specification file you created is valid for these events). For each run your basin and control specification file is same, but your Meteorologic model/file will be different for each event. These are hypothetical rainfall scenarios that you are creating to see how the hydrograph will change.

After each run, copy the time column and the output hydrograph (Total Flow column in the Time Series Table from the results tab) in a MS Excel file and save this file. You can use one excel file to store results from all simulations. By using the rainfall data in the Excel file and the hydrograph, create a presentation quality graph to show both the rainfall hyetograph and discharge hydrograph (rainfall-runoff plot) in one single plot for each event as shown below.



Make sure you include legend, axes titles and chart title. If you have never created a plot like this before, here are some keywords: secondary axis, reverse order, chart type. Present all three four plots on a single page.

Turn-in Part B (January 31, 1:30 PM) – hard copy

Page 1: Rainfall hyetographs (Excel) for three events

Page 2: Rainfall-runoff plots for all three events

Page 3: For each event, report the total amount of rainfall in mm, peak discharge value in m³/s, time to peak in hours, basin lag (time from rainfall peak to hydrograph peak) in hours and base time in hours of the output hydrograph. Present this information in a Table. After this table, write few sentences (at least 50 words) on how the rainfall distribution is affecting the streamflow hydrograph. Specifically, I want you to look at the total amount of rainfall for each event, the shape of the rainfall hyetograph and the discharge hydrograph, and how the time to peak, basin time and base time varies for these events.