

Hydrologic Modeling using HEC-HMS

Prepared by
Venkatesh Merwade
School of Civil Engineering, Purdue University
vmerwade@purdue.edu

April 2007

Introduction

The intent of this exercise is to introduce you to the structure and some of the functions of the HEC-Hydrologic Modeling System (HEC-HMS), by simulating the runoff hydrographs resulting from a design storm on Waller Creek in Austin, Texas. This exercise involves the study of extreme discharges on Waller Creek using rainfall-runoff analysis. The physical parameters describing the watershed were developed previously using a GIS program (PrePro) available at <http://ceprofs.tamu.edu/foivera/GISTools/PrePro2002/PrePro2002.htm>. A GIS pre-processing tool called HEC-GeoHMS, available from U.S. Army Corps of Engineers can also be used for this task. HEC-GeoHMS is available at: <http://www.hec.usace.army.mil/software/hec-geohms/>

A tutorial on how to use the newer version of HEC-GeoHMS that works with the latest version of ArcGIS is available at: <http://web.ics.purdue.edu/~vmerwade/tutorial.html>

Computer Requirements

You must have a computer with the latest windows operating system, and HEC-HMS 3.1.0 installed.

HEC 3.1.0 for windows is available for free from the Hydrologic Engineering Center's home page at: <http://www.hec.usace.army.mil/software/hec-hms/download.html>. A user's manual is also available at this location. (Purdue Students: The program is loaded on computers in ITaP's JNSN B012 lab)

Data Requirements

To run HEC-HMS model, a basin file is needed to specify the physical parameters of the watershed, and a map file to give the outline of the drainage areas and creeks. These files for Waller creek (zipped as waller.zip) can be downloaded from the following link:

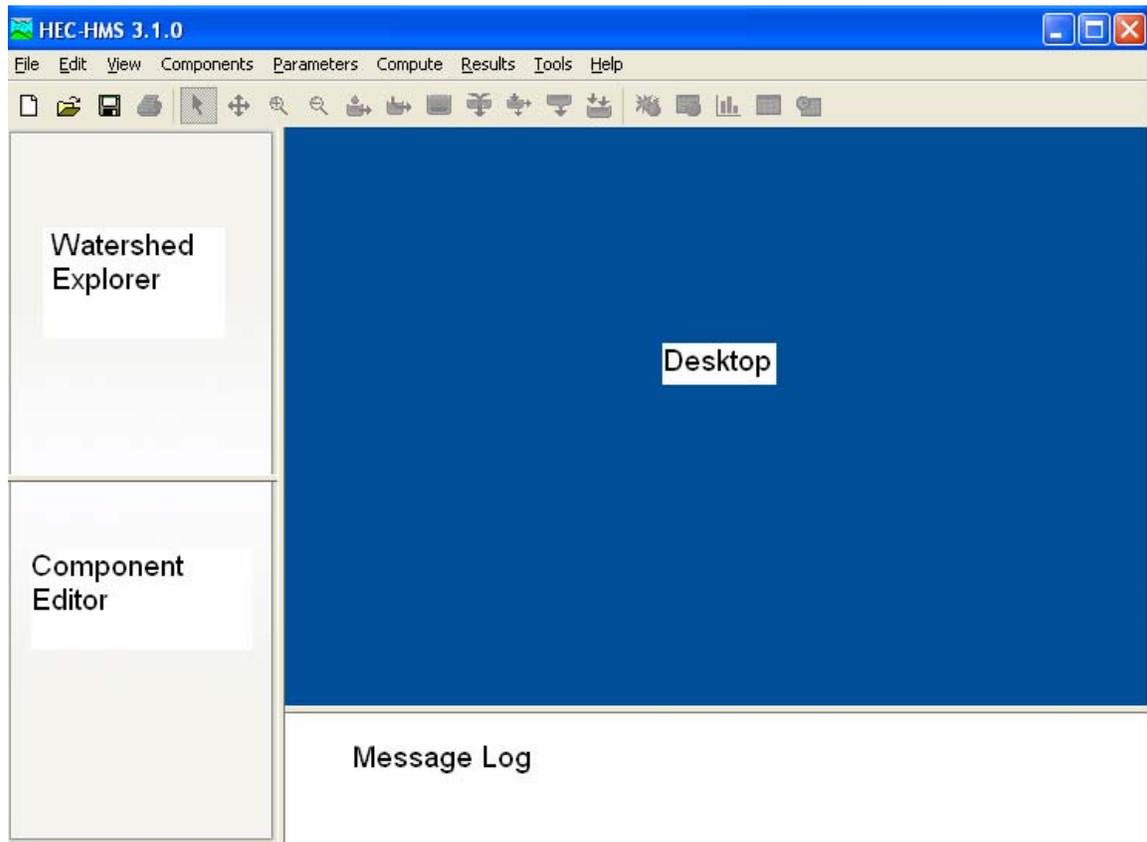
<http://web.ics.purdue.edu/~vmerwade/education/waller.zip>

Make a working directory on your computer and **download** *waller.zip*, and **unzip** its contents. .

Getting Started

Start HEC-HMS by clicking on the HEC-HMS icon by going to *Start → Programs → HEC-HMS → HEC-HMS 3.1.0* (Purdue students doing this tutorial in an ITap Lab may have to first run the set-up file by going to *Start → Programs → Course Software → Engineering → HEC-HMS → HEC-HMS 3.1.0*)

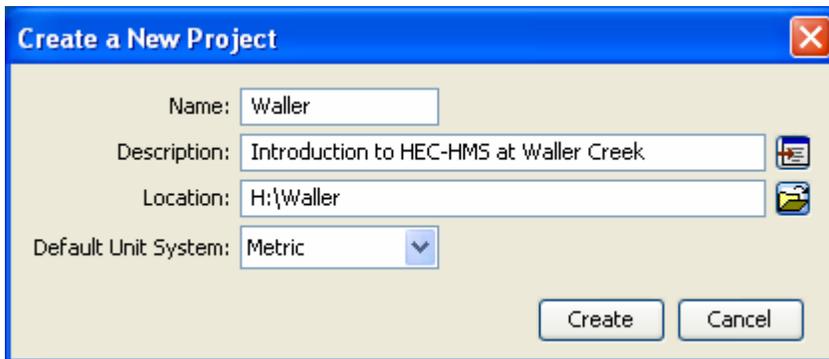
After a few seconds, the following should appear:



Henceforth, this window will be referred to as HMS Interface. 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. More description on these panes, menus and tools is provided when they are used later in the exercise.

Creating a new HEC-HMS project

The first step is to create a new HEC-HMS project by **selecting** *File*→*New* in the menu bar. Enter the project name and description as shown below, and specify your working directory (where waller.zip is downloaded).



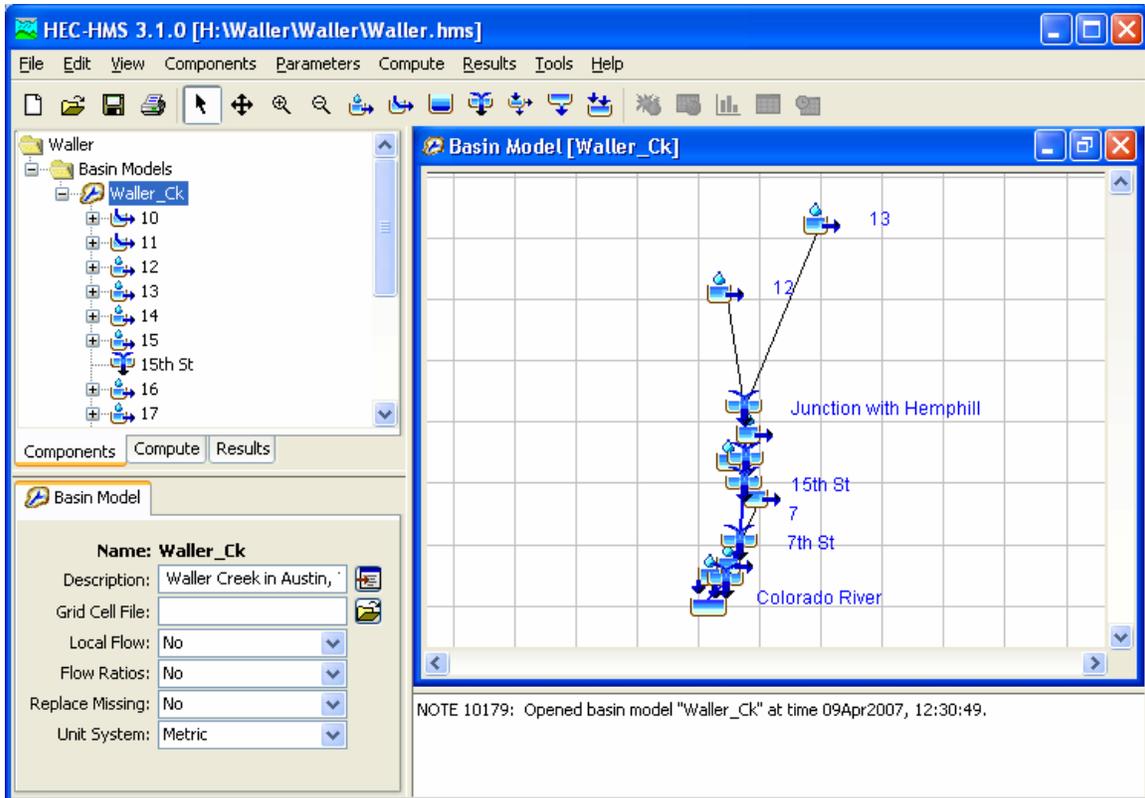
Click *Create* to create the Waller project.

HMS Model Components

HMS has four main model components (you can see these by selecting components on the menu bar): basin model, meteorologic model, control specifications and input data (time series, paired data and gridded data). The Basin Model, for instance, contains information relevant to the physical attributes of the model, such as basin areas, river reach connectivity, or reservoir data. Likewise, the Meteorologic Model holds rainfall data. The Control Specifications section contains information pertinent to the timing of the model such as when a storm occurred and what type of time interval you want to use in the model, etc. Finally, the input data component stores parameters or boundary conditions for basin and meteorologic models. Each of the sections is explored below individually

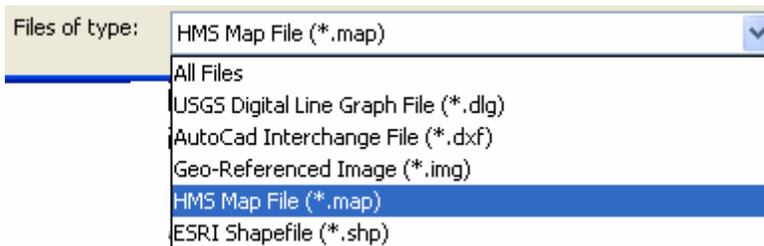
Importing a Basin Model

The first of the *Components* we will consider is the *Basin Model*. To create a hydrologic model of Waller Creek, you need to import the basin file that you just downloaded. In HMS interface, use *File*→*Import*→*Basin Model...* **Browse** to *Waller_Ck.basin* in the working directory, and **click** *Select* to add it to the HMS project. To see the Waller Creek basin model in the *Desktop* window, **expand** (by clicking on the + sign) the folders in the *Waller* folder until you get to *Waller_Ck* as shown below.



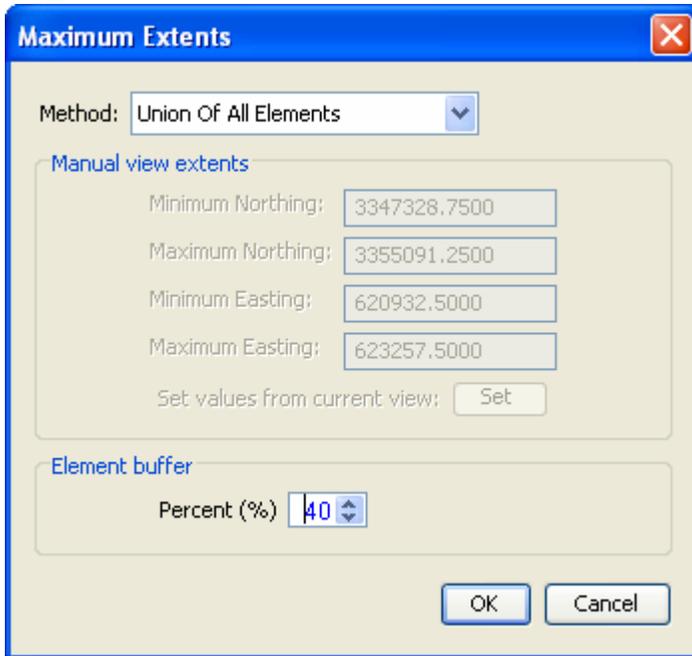
If you expand Waller_Ck, you will see different hydrologic elements in the basin in the watershed explorer. Before we get into the details of the basin, go ahead and add the basin map which is stored in the *Hms.map* file.

In the HMS interface, **select** *View*→*Background Maps*. This will prompt a Background Maps window. In the Background Maps window, **click** on *Add... Browse* to the *Hms.map* file in the working folder (make sure you change the Files of type to HMS Map File (*.map) as shown below), and **select** it.



This will add Hms.map in the Background Maps window. **Click** *OK* to add it to the HMS

You should then see a schematic of Waller Creek showing the watershed and stream map and an overlay of the hydrologic elements. You will notice that the whole map of the basin does not fit in the Desktop window. You can change the Desktop settings by **selecting** *View*→*Maximum Extents* in the Menu bar. **Select** the display method as *Union of All Maps* and Elements with an *element buffer of 40 %* or more as shown below:



Now you should see the whole basin in the desktop window. **Save** the project.

Navigating the HMS Desktop

You can use the following four tools in the tool bar to navigate through the HMS desktop:



The arrow tool  lets you select any hydrologic element in the basin. You can use the zoom-in tool  to zoom-in to a smaller area in the desktop, and zoom-out tool  to zoom out to see a larger area. The pan tool  can be used to move the display in the desktop. Go ahead and experiment with these buttons to understand better how each works.

Now lets explore the basin information.

Hydrologic Elements

The *Waller_Ck* basin contains different hydrologic elements. The following description gives brief information on each symbol that is used to represent individual hydrologic element.



Subbasin – Used for rainfall-runoff computation on a watershed.



Reach – Used to convey (route) streamflow downstream in the basin model.



Reservoir – Used to model the detention and attenuation of a hydrograph caused by a reservoir or detention pond.



Junction – Used to combine flows from upstream reaches and sub-basins.



Diversion – Used to model abstraction of flow from the main channel.



Source – Used to introduce flow into the basin model (from a stream crossing the boundary of the modeled region). Source has no inflow.

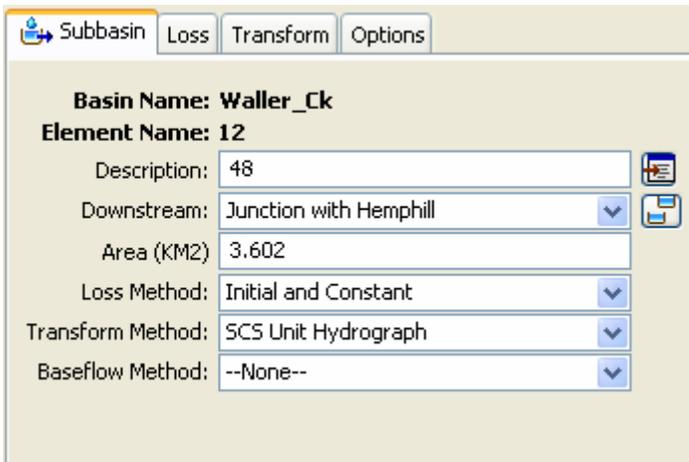


Sink – Used to represent the outlet of the physical watershed. Sink has no outflow.

The model of Waller Creek contains only 4 of these kinds of elements. There are 18 hydrologic elements in the Waller Creek model, made up of 7 subbasins, 5 river reaches, 5 junctions, and 1 sink at the point where Waller Creek flows into the Colorado River. Notice that when a stream flows through a watershed, the additional local runoff from the drainage area around the stream is not accounted for until the downstream end of the reach where its flow is combined at a junction with the flow coming from the upstream reach. The junctions have been located at points where roads cross Waller Creek.

Editing a Basin Model

Make sure the Waller_ck basin is expanded in the watershed explorer to see all the hydrologic elements in the basin. **Select** the *Arrow* tool  from the tool bar, and **click** on *sub-basin 12* icon in the watershed explorer. After this sub-basin is highlighted, information related to this sub-basin will appear in the component editor window (lower left corner).

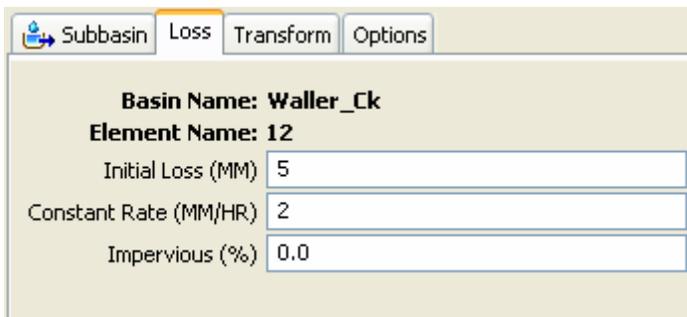


Basin Name: Waller_Ck	
Element Name: 12	
Description:	48
Downstream:	Junction with Hemphill
Area (KM2)	3.602
Loss Method:	Initial and Constant
Transform Method:	SCS Unit Hydrograph
Baseflow Method:	--None--

Remember the sub-basin element is used to convert rainfall to runoff. So the information on methods used to compute loss rates, hydrograph transformation and baseflow is required for each sub-basin element. The loss method allows you to choose the process which calculates the rainfall losses absorbed by the ground. Click (do not select any!) on the drop down menu to see your choices. Some options are *SCS Curve No.* and *Green & Ampt*. In this model, *Initial and Constant* has been selected. This loss relationship means that a quantity of rainfall will be absorbed by permeable soil initially, and a constant rate will be absorbed over the time frame of the model. The loss method will convert the rainfall hyetograph to excess rainfall (chapter 5 in the text book).

The *Transform method* allows you to specify how to convert excess rainfall to direct runoff. Again, click on the drop down menu to view your options. You may remember these options from class notes (chapter 7 in the text book). This model employs the *SCS* technique (dimensionless unit hydrograph in Chapter 7). The *modClark* model takes gridded rainfall data, subtracts the losses as specified through the *Loss Rates*, and converts the excess rainfall to a runoff hydrograph using a variation of what is known as the Clark unit hydrograph. There is no baseflow method specified for this model, but you can look at the available options. If we specify baseflow, this baseflow will be added to the resulting direct run-off hydrograph to produce total streamflow hydrograph.

Once the loss and transform methods are chosen for the sub-basin, the next step is to specify the parameters for these methods. **Select** the *Loss* tab in the component editor to look at the parameters for the loss method.

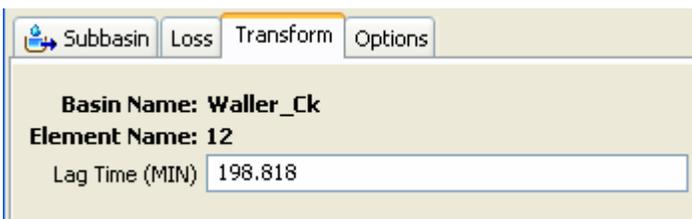


The screenshot shows a software interface with four tabs: 'Subbasin', 'Loss', 'Transform', and 'Options'. The 'Loss' tab is selected and highlighted. Below the tabs, the following information is displayed:

- Basin Name: Waller_Ck
- Element Name: 12
- Initial Loss (MM): 5
- Constant Rate (MM/HR): 2
- Impervious (%): 0.0

Each sub-basin requires an *initial loss* quantity, a **constant loss rate**, and a *percent imperviousness*. These values have been selected arbitrarily. If the % impervious value differs from 0, that % of the land area is assumed to have no losses and the loss method is applied only to the remainder of the drainage area

Similarly **select** the *Transform* tab to look at the parameters for the transform method.



The screenshot shows the same software interface as above, but with the 'Transform' tab selected and highlighted. The following information is displayed:

- Basin Name: Waller_Ck
- Element Name: 12
- Lag Time (MIN): 198.818

Note that the SCS unit hydrograph method requires only one parameter for each sub-basin: lag time between rainfall and runoff in the sub-basin. The parameter that is specified here is t_p , and the program will compute T_c (time of concentration) and q_p (peak flow) to rescale the SCS dimensionless unit hydrograph (Chapter 7). This is then used to compute the direct runoff hydrograph for this sub-basin.

The *Options* tab is used to enter observed streamflow and stage data which is left blank for this model.

After the sub-basin element, let's look at a reach element. **Click** on *reach 10*, and **look** at its parameters in the component editor.

Basin Name: Waller_Ck
Element Name: 10
Description: 90
Downstream: MLK
Routing Method: Muskingum
Loss/Gain Method: --None--

Since the reach element route flows, only one method (routing) is associated with it. **Click** on the drop-down menu to look at choices available for routing flows. The *Muskingum* method is specified here, which is the routing technique used for the reaches in this model. Routing is described in chapters 8-10 in the text book.

Select the *Route* tab to look at the parameters for the routing method (*Muskingum*).

Basin Name: Waller_Ck
Element Name: 10
Muskingum K (HR): 0.8583
Muskingum X: 0.15
Subreaches: 2

This simulation routes the water through the reaches by the Muskingum method in which K is the travel time of a flood wave passing through the reach, X is a measure of the degree of storage ($X = 0$ means a level-pool reservoir or maximum storage, $X = 0.5$ means a pure transmission reach in which there are no storage effects, and X ranges between 0 and 0.5). The reach is divided into a number of subreaches if necessary to keep the computations numerically stable.

You can explore the junctions, source and sink elements to see how they are specified.

There are a couple of ways to look at Basin model. If you expand the sub-basin model (by clicking the + sign next to it) in the watershed explorer, you will see the methods specified for the sub-basin, and when you click on the method, you see the parameters in the component editor. Alternatively, you can look at the parameters for all hydrologic elements by selecting Parameters in the menu bar and selecting a method. For example, by **selecting** *Parameters* → *Transform* → *SCS Unit Hydrograph* gives a list of lag times for all the sub-basins in the model as shown below:

The screenshot shows a software window titled "SCS Transform[Waller_Ck]". At the top left, there is a dropdown menu labeled "Show Elements:" with "All Elements" selected. Below this is a table with two columns: "Subbasin" and "Lag Time (MIN)". The table contains seven rows of data for subbasins 12 through 18. At the bottom right of the window, there are two buttons: "Apply" and "Close".

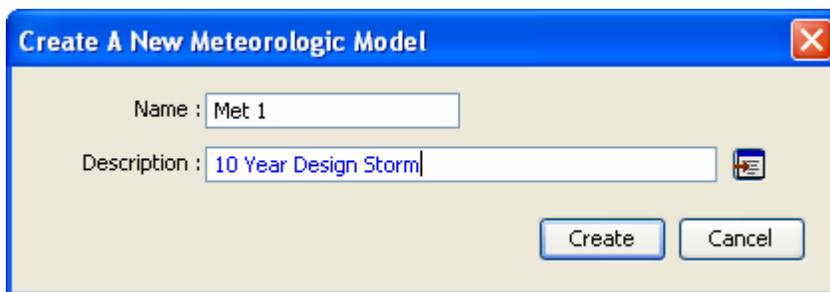
Subbasin	Lag Time (MIN)
12	198.818
13	320.53
14	66.547
15	58.3529
16	104.706
17	49.4559
18	43.25

Creating a Meteorologic Model

Having established the Basin Model, we will now create the Meteorologic Model. The precipitation and evapotranspiration data necessary to simulate a watershed are stored in the meteorologic model. From a statistical study of extreme storm rainfall data recorded at gages, tables and maps have been prepared for the whole US which specify the storm precipitation depth to be expected as a function of the return period of the event and the duration of the rainfall. A table of such values is shown below for Travis County, in which the City of Austin is located. For example, for a 10 year return period event, we expect that in 5 min 0.85" of rainfall will fall, in 15 min 1.66", and so on, up to 720 min (12 hours) and 1440 min (24 hours). These precipitation depths are the values which would be equalled or exceeded on average once in 10 years when considering a very long period of data. As rainfall duration increases, the cumulative depth of rainfall increases but the average intensity over the duration decreases because severe rainfall cannot be sustained for very long.

<i>Design Precipitation Depths for Travis County, Texas (in)</i>						
	Return Period (yr)					
Duration (min)	2	5	10	25	50	100
5	0.60	0.75	0.85	0.98	1.07	1.24
15	1.15	1.47	1.66	1.93	2.12	2.43
60	1.94	2.55	2.91	3.41	3.80	4.31
120	2.35	3.13	3.57	4.21	4.74	5.35
180	2.60	3.47	3.98	4.71	5.34	6.02
360	3.05	4.11	4.75	5.65	6.45	7.28
720	3.54	4.87	5.57	6.71	7.75	8.69
1440	4.06	5.67	6.52	7.94	9.26	10.39

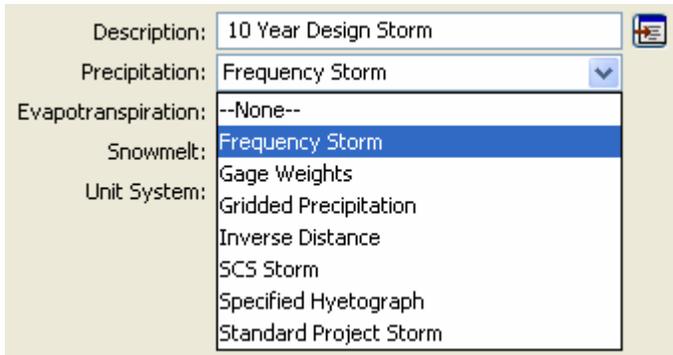
We will input the rainfall to HMS in English units (inches). To import an existing basin or meteorological model, one can use the menubar by selecting File→Import (like you did for the basin model). To create a new model, we use Components menu. To create a new Meteorologic Model, **select** *Components*→*Meteorologic Model Manager*. In the Meteorologic model manager, **click** *New* and enter the following information:



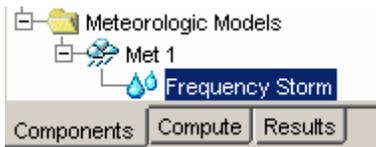
Click *Create* and **close** the Meteorologic model manager. This will add Meteorologic Models folder to the watershed explorer. **Expand** the *Meteorologic Models* folder, and **select** the *Met 1* model as shown below:



Once the Met 1 model is selected in the watershed explorer, its details will appear in the component editor. The description box includes the description you entered earlier when creating the model. By default, the precipitation data type is Specified Hyetograph (which is the most common data type). In this exercise, however, we will use design precipitation data for Travis County. From the drop-down menu for precipitation, **choose** *Frequency Storm*, and specify US Customary units as shown below:



Similarly **select** the *Basins* tab and **specify** *Yes* to include sub-basins. Once we specify the precipitation type, the next step is to enter the data. Unlike Basin model, there is no extra tab in the component editor to enter the data after you choose the precipitation type. To enter the data, you need to expand the *Met 1* model in the watershed explorer and then choose the precipitation type (Frequency Storm) as shown below:



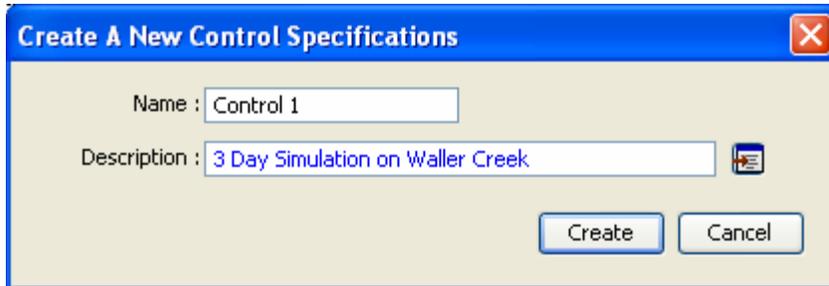
This will prompt a precipitation tab in the component editor. **Fill** in the values shown in the Travis County table above for a 10 year storm (10% chance of being equalled or exceeded in any year). The storm is configured by selecting the exceedance probability (10 % or 10 year return period), output type (Annual duration), duration of maximum intensity (5 mins), storm duration (one day), and percentage of storm that occurs before the peak intensity (50 %) as shown below:

Precipitation	
Name: Met 1	
Probability:	10 Percent
Output Type:	Annual Duration
Intensity Duration:	5 Minutes
Storm Duration:	1 Day
Intensity Position:	50 Percent
Storm Area (MI2)	
5 Minutes (IN)	0.85000
15 Minutes (IN)	1.6600
1 Hour (IN)	2.9100
2 Hours (IN)	3.5700
3 Hours (IN)	3.9800
6 Hours (IN)	4.7500
12 Hours (IN)	5.5700
1 day (IN)	6.5200

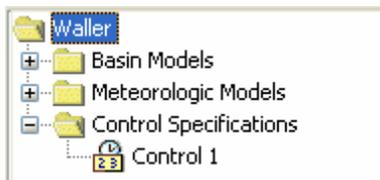
Save the project. For each project, the HMS creates an output Data Storage System DSS file which stores calculated data from all runs for a given project so that results from a previous run can be directly compared to results from a more current run.

Defining the Control Specifications

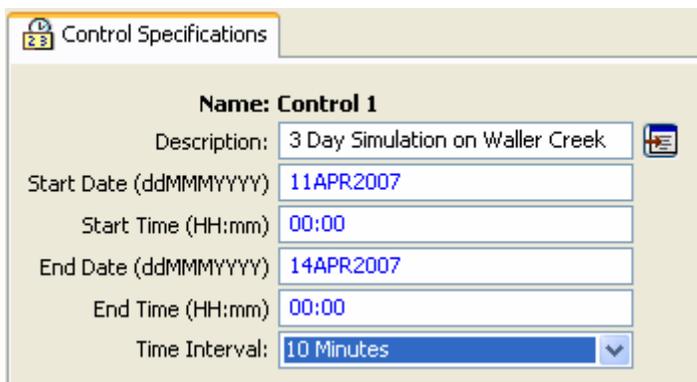
The final task in the model setup involves establishing the model's time limits. **Select Components** → **Control Specifications Manager**. **Select New** in the control specifications manager and **type** the following information:



Click Create and **close** the control specifications manager. This will add a Control Specifications folder in the watershed explorer. To see the control specifications file, **expand** the folder, and select Control 1



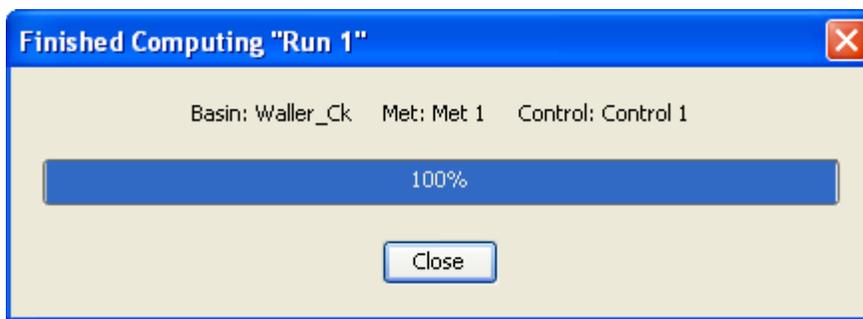
This will prompt the control specifications tab in the component editor. **Specify** the duration of the simulation in date and time, and also the time interval of the calculations (10 minutes) as shown below:



In this case, the duration is arbitrary; long enough to depict the runoff from a 1-day storm, but the 10 minute time interval is part of the Basin file model setup and should remain fixed for this Waller Creek model.

Executing the HMS Model

Finally, you have finished perusing the data involved in creating the Waller Creek model. The last step is to run the model. **Select** *Compute* → *Create Simulation Run*. **Accept** the default name for the run (Run 1), **click** *Next* to complete all the steps and finally **Click** *Finish* to complete the run. Now to run the model, **select** *Compute* → *Select Run* → *Run 1*, and then go to *Compute* → *Compute Run [Run 1]* to see the following window (alternatively you can click the compute run tool  in the tool bar):



Click *Close*. You will see a log in the message log as program executes the model. If there are errors in the model, you will see them in red color. For this model, there are no errors.

If you want to make runs with alternative model files, you can do so by first creating/importing new model files (basin, Meteorologic and control specifications), creating a new run (say Run 2) by going to *Compute* → *Create simulation run* and selecting the new files while creating Run 2. Though the model used in this exercise has one dataset each for basin, meteorology and control specifications, HMS is slick in that it allows the user to have multiple data sets available to include conveniently in different runs.

Viewing HMS Results

The HMS allows you to view results in tabular or graphical form. To view a global results table, **select** *Results* → *Global Summary Table* (alternatively you can click the Global Summary tool  in the tool bar). You will get a window like the one shown below which summarizes the peak discharge and time, the total volume of storm runoff and the drainage area from which it came.

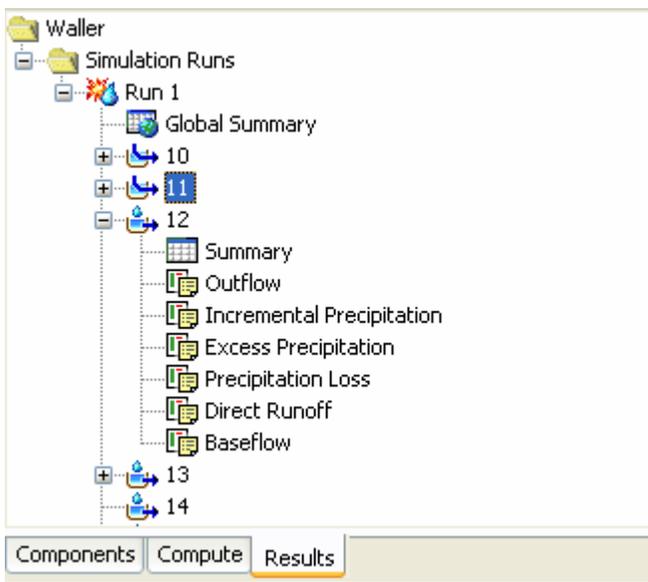
Project: Waller Simulation Run: Run 1

Start of Run: 11Apr2007, 00:00 Basin Model: Waller_Ck
End of Run: 14Apr2007, 00:00 Meteorologic Model: Met 1
Compute Time: 09Apr2007, 13:02:02 Control Specifications: Control 1

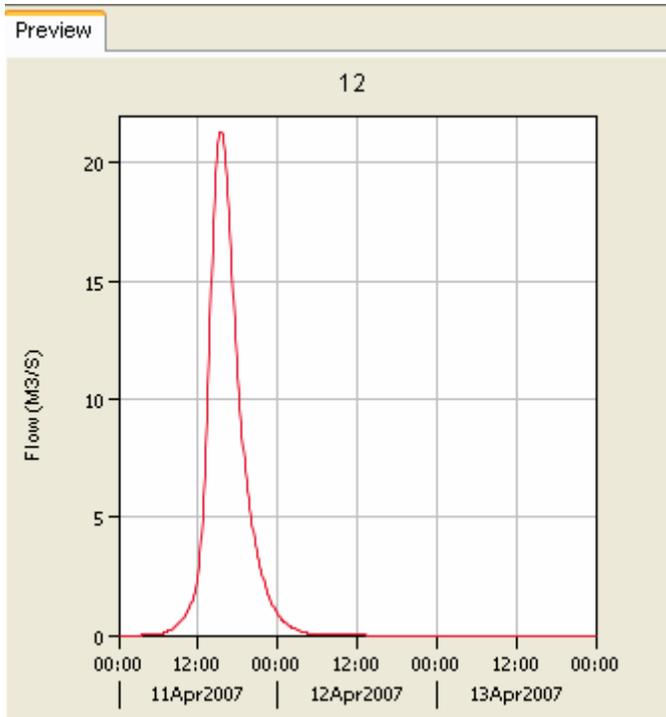
Volume Units: MM 1000 M3

Hydrologic Element	Drainage Area (KM2)	Peak Discharge (M3/S)	Time of Peak	Volume (MM)
10	10.514	44.0	11Apr2007, 17:10	117.07
11	11.480	44.2	11Apr2007, 17:40	117.11
12	3.602	21.3	11Apr2007, 15:30	117.27
13	6.912	27.6	11Apr2007, 17:30	116.97
14	0.966	12.8	11Apr2007, 13:10	117.51
15	0.563	8.1	11Apr2007, 13:00	117.55
15th St	12.043	44.5	11Apr2007, 17:40	117.13
16	1.633	15.8	11Apr2007, 13:50	117.45
17	0.437	7.0	11Apr2007, 12:50	117.56
18	0.252	4.4	11Apr2007, 12:50	117.58
1st St	14.113	44.3	11Apr2007, 19:20	117.18
7	12.043	43.6	11Apr2007, 18:40	117.13
7th St	13.676	44.6	11Apr2007, 18:40	117.17
8	13.676	44.2	11Apr2007, 19:20	117.17
9	14.113	44.0	11Apr2007, 19:40	117.18
Colorado Ri...	14.365	44.0	11Apr2007, 19:40	117.19
Junction wit...	10.514	44.9	11Apr2007, 16:20	117.07
MLK	11.480	44.6	11Apr2007, 17:10	117.11

In addition to viewing global results, you may also view results for each element within the model. Again there are a couple of options to do this, and each option provides output in different ways. One option is to use the watershed explorer and component editor tab. To view results, you select the Results tab in the watershed explorer, expand the Simulation Runs folder, and expand Run 1. To see results for any element, expand that element as seen below:

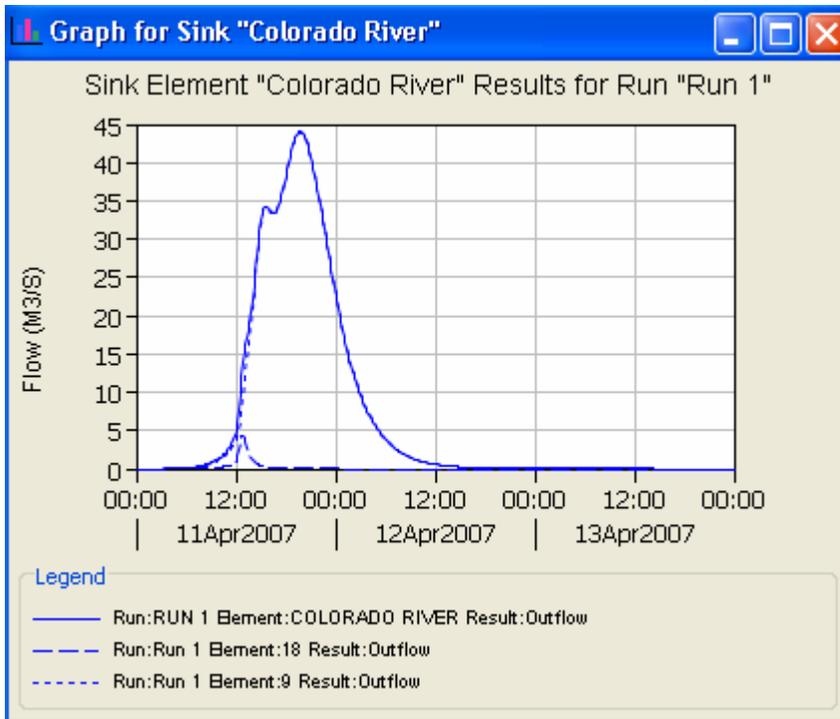


To see the outflow from sub-basin 12, you can select outflow and see the outflow hydrograph in the component editor as shown below:

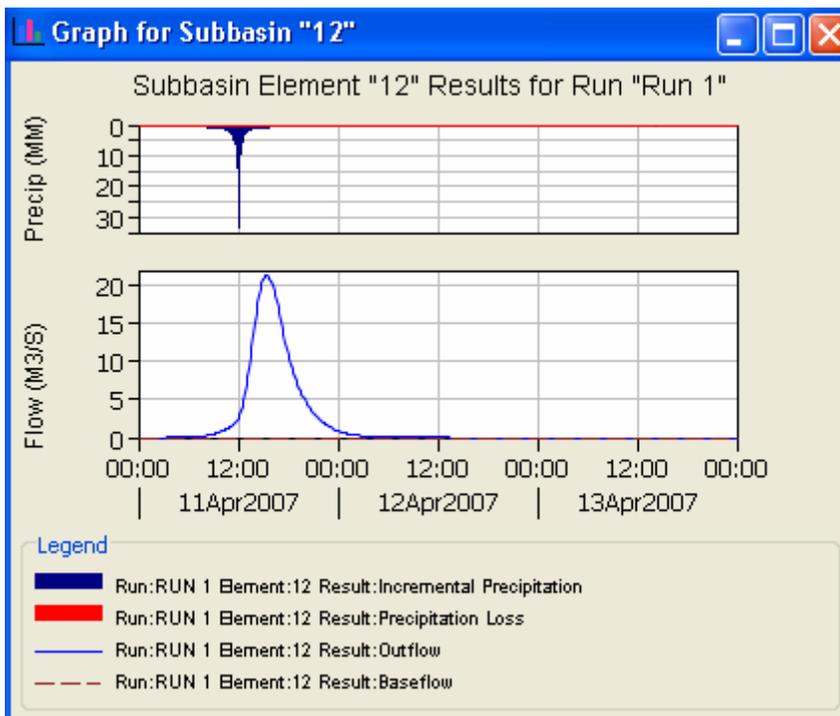


Similarly you can look at other graphs in the component editor by selecting the variable in the watershed explorer. You can select a reach element and see the attenuation in the inflow and outflow hydrograph by selecting the combined inflow and outflow option in the watershed explorer. Each element also has a summary option that gives the results from the global summary table (a single row of the table) for that particular element.

Another way of looking at results is by using the tools on the tool bar, which show results in a different way than the component editor. However, to use these tools, you need to select the element by using the component tab in the watershed explorer. For example, **select** *Colorado River* (Sink) in the watershed explorer, and **click** on the view graph tool  to get the following graph:



The dashed line hydrographs are the inflow data from the sub-basins immediately upstream of the Colorado River which is added to the routed flow in the channel to produce the total outflow curve. If you **click** on a sub-basin (12), you see the rainfall at the top and the runoff at the bottom as shown below:



Unlike a single graph in the component editor, you get to see all graphs (input precipitation, outflow hydrograph, baseflow, precipitation losses) in a single window using this option. You can also see the results in tabular form by using the view time series table tool  in the toolbar. These functions are also accessed through the Results menu on the menu bar.

OK, you are done with learning the basics of HEC-HMS for event based modeling. The following sections are designed to use your skills in doing some hydrologic design analysis and create new model from scratch.

1. Waller Creek Diversion Design Study

The citizens of Austin approved a bond issue during 1998 which authorizes the City to borrow approximately \$25 million for the construction of a flood diversion tunnel for Waller Creek. At present the 100 year flood plain for the creek covers many city blocks in downtown Austin. The goal is to make this area safer from floods and also to encourage the development of a river walk area, perhaps similar to that in San Antonio. Assume that you are the hydrologist whose job it is to determine the design discharge at the inlet point of this tunnel which is just on the downstream side of 15th Street. At this point an overflow weir will divert a significant part of the flood discharge and convey it to Town Lake in a 20 ft diameter tunnel.

Recompute the flows from the HMS model using the 100 year design storm instead of the 10 year design storm. How much water would have to be diverted at 15th street if the diversion tunnel is to reduce the peak flow at 15th during a 100 year storm to the level that it has now during a 10 year storm?

Recompute the flows from the HMS model using a 2-year, 5-year, 25-year, and 50-year storms. Make a plot showing the peak discharge to be expected from these events at 15th St as a function of the storm severity from 2-years to 100-year return periods.

To be turned in: A graph and a table showing the relationship between flood peak discharge and return period for Waller Creek at 15th St. Determine the flow diversion needed at 15th St to reduce the natural 100 year flow to the 10 year discharge level.

2.0 Create a simple model to create a runoff hydrograph.

An 8-mi² watershed has a time of concentration of 1.0 hour. Use HEC-HMS to determine the direct runoff for a storm (rainfall hyetograph given below) using SCS Unit Hydrograph method.

Date and time	Excess Rainfall (in)
01/01/2002 00:00	0
01/01/2002 00:10	0.6
01/01/2002 00:20	0.4
01/01/2002 00:30	0
01/01/2002 00:40	0
01/01/2002 00:50	0
01/01/2002 01:00	0

Hint:

1. You must create new basin, Meteorologic and control specifications file either in the same project or a new project.
2. You will need only one sub-basin element in the basin file with area and SCS hydrograph details. You only need transform method. Specify None for loss method (input provided is excess rainfall) and baseflow.
3. The input to Meteorologic file involves using the Time Series Data Manager from the Components menu. This will add Time-Series data folder in the watershed explorer and you will expand this folder to specify the rainfall hyetograph. When you edit the Meteorologic model in the component editor, you will use the Specified hyetograph (default method) method. You will then click on the specified hyetograph option in the watershed explorer and link the time series data to the Meteorologic model.
4. In the Control Specifications model, use 10 min time interval and the simulation time as 12 hours (end time = 12 hrs past start time).

To be turned in: A graph and a table showing the outflow hydrograph.