MIDUSS®
Version 2

Reference Manual

Alan A. Smith Inc.

Published by:

Alan A. Smith Inc.

17 Lynndale Drive, Dundas, Ontario, Canada L9H 3L4
Tel: +1 (905) 628-4682
Fax: +1 (905) 628-1364
info@alanasmith.com
www.alanasmith.com

ISBN 0-921794-00-3

February 2004

Version 2.00 Rev200

Table of Contents

Chapter 1 An Overview of MIDUSS 1

Chapter 4 Design Options Available 109

Notes:

Chapter 1 An Overview of MIDUSS

The following general areas describe the overall purpose of the MIDUSS program, the structure of the main menu, major differences from previous versions of the program and a description of the functionality available in MIDUSS.

- Chapter 1 - Overview of MIDUSS
- Chapter 2 - Structure and Scope of the Main Menu
- Chapter 3 - Hydrology Used in MIDUSS
- Chapter 4 - Design Options Available in MIDUSS
- Chapter 5 - Hydrograph Manipulation.
- Chapter 6 - Working with Files
- Chapter 7 – Hydrological Theory
- Chapter 8 - Theory of Hydraulic Design.
- Chapter 9 - Displaying Results.
- Chapter 10 - Running in Automatic Mode
- Chapter 11 – MIDUSS Tools
- Appendix ‘A’ - References
- Appendix ‘B” – Installing MIDUSS
- Appendix ‘C’ – Malaysian Customization
- Appendix ‘D’ – What's New in MIDUSS Version 2

This manual created: January 26, 2004
Edited: February 11, 2004

MIDUSS Version 2.00Rev200

1.1 An Introduction to MIDUSS

The MIDUSS package was developed to help drainage engineers to design the hydraulic elements in a collection network of storm sewers or channels. The program does not make design decisions but rather carries out hydrological and hydraulic analyses and presents you with design alternatives. It is then left to you as the engineer to select values for the design variables and to decide on the acceptability of a design.

MIDUSS is highly interactive in use, and allows engineering judgment to be exercised at all stages of the design process. Moreover, this interaction lets you monitor each step of the process and take corrective action in the event of an error. With most commands, data is input in response to prompts, and you are free from the need to prepare lengthy data files prior to the design session. In many cases a design session will require to be repeated either:

1. To test a previously designed system under a different storm.
2. To continue or modify a previous design session.
3. To modify a hydrology or design parameter.

To simply continue a design session you may use the **File / Save Session** and the **File / Load Session**, but to make any changes you will find that Automatic Mode is invaluable.

1.2 Using Automatic Mode

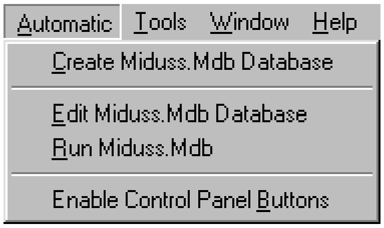
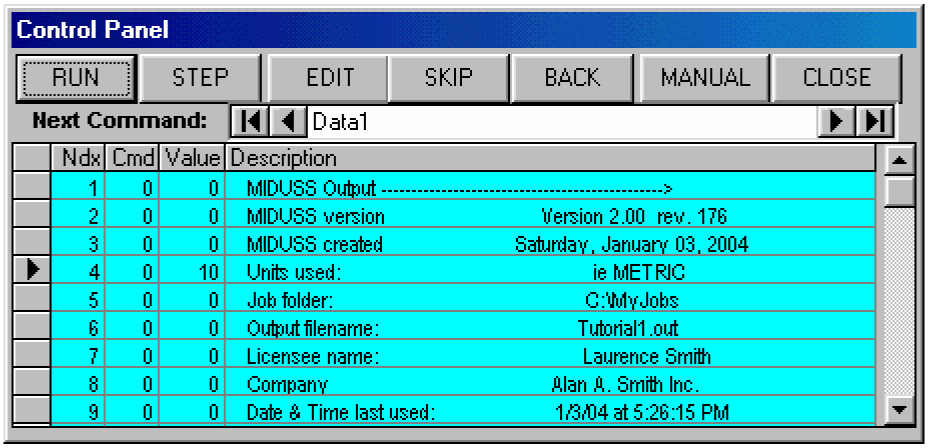


Figure 1-1 – The Automatic Menu

When a design session has to be repeated with a modification such as a different storm, it is possible to use an input file which contains a record of the previous session with all the commands and the relevant data. In MIDUSS this input file is in the form of a database which can be produced automatically from a previously created output file. Running MIDUSS in automatic mode eliminates the need to re-enter the commands and data from the keyboard. In this mode, however, design decisions with respect to pipes, ponds, channels or diversion structures can still be altered and changes can be made to the database. The use of files in this way is discussed in more detail later in this Help file. (see Chapter 10 - Running MIDUSS in Automatic Mode)



Next Command: Data1

Ndx	Cmd	Value	Description
1	0	0	MIDUSS Output -->
2	0	0	MIDUSS version Version 2.00 rev. 176
3	0	0	MIDUSS created Saturday, January 03, 2004
4	0	10	Units used: ie METRIC
5	0	0	Job folder: C:\MyJobs
6	0	0	Output filename: Tutorial1.out
7	0	0	Licensee name: Laurence Smith
8	0	0	Company Alan A. Smith Inc.
9	0	0	Date & Time last used: 1/3/04 at 5:26:15 PM

Figure 1-2 The Automatic Control Panel

The hydrology and hydraulics used in MIDUSS are based on well-established and accepted principles. The details of these techniques are described under the various command headings in Chapter 3 - *Hydrology Used in MIDUSS* and Chapter 4 - *Design*

Sections have been added which present the relevant background theory for the hydrology and hydraulics used in MIDUSS. These need not be studied in order to run MIDUSS but have been included for the interested reader or student.

The next section provides a general description of a typical design session A more detailed example is presented in the MIDUSS Tutorial manual

1.3 A Simple Example

1.3.1 A Typical Design Session

Assuming you have pressed the [Yes] button when the initial Disclaimer form is displayed, MIDUSS starts up with the main menu displayed at the top of the window. Depending on the size and resolution of your screen you will probably want to click on the ‘maximize window’ symbol at the right hand end of the title bar to make the MIDUSS window fill the screen.

With reference to the simple example shown in Figure 1-3, a typical session may be summarized by the sequence of steps described in the following topics. You should first define the system of units to be used. There are two ways to do this. You can use the **Edit / Units** menu and select either **Metric** or **Imperial**. For subsequent design sessions you will see a small form defining the current options selected for the units and three other options that are not likely to change often. It will often be faster to simply click on the [CONTINUE} command button on this form or occasionally alter your preferences. This new form is shown in Figure 1-5. The [Change] button provides the second way to define the units.

After setting the system of units, your first action should be to define the Output File to record the session.

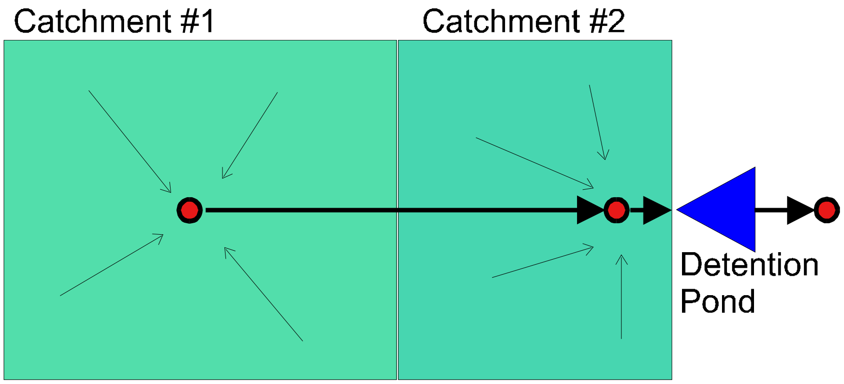


Figure 1-3 – A Simple Two-catchment System

1.3.2 Define the Output File

Figure 1-4 – Defining and Output File

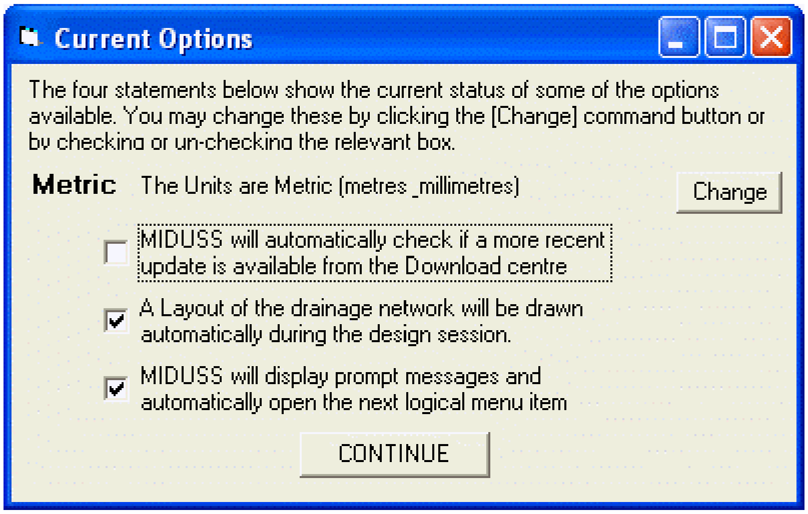
Using the **File / Open Output file** menu item, navigate through the standard Windows Common Dialog Box to select an existing file or create a new one in a directory of your choice. This directory will serve as the Job Directory for this session and the file will contain a record of the commands used, the data entered and some of the results. The Job Directory will also contain any files that you create such as storm or hydrograph files.

If you don't specify a particular folder and file, MIDUSS will use a default output file (called 'default.out') which resides in the MIDUSSData directory.

You must use the **File / Open Output file** menu item in order to enable several of the other items on the main menu.

After selecting the Job Directory the next logical step is to Select your Options

1.3.3 Select your Options



The four statements below show the current status of some of the options available. You may change these by clicking the [Change] command button or by checking or un-checking the relevant box.

Metric The Units are Metric (metres _millimetres)

☐ MIDUSS will automatically check if a more recent update is available from the Download centre

☑ A Layout of the drainage network will be drawn automatically during the design session.

☑ MIDUSS will display prompt messages and automatically open the next logical menu item

Figure 1-5 – Confirm or change the Current Options

MIDUSS offers a number of features most of which are optional. The most important is the system of units

Immediately following acceptance of the Disclaimer form you will be prompted to select the system of units via a form similar to Fig 1-5 above.

Your choice for units and other options will be remembered when the session is finished and MIDUSS will start the next session with the same selection.

Other important options at this point deal with: checking for available MIDUSS updates, automatic drawing of the drainage network and turning on or off prompt messages.

There are still more options in the Options menu under the Edit menu item. These will be explained in more details later in this Reference Manual. Figure 1-6 below shows these additional user preferences.

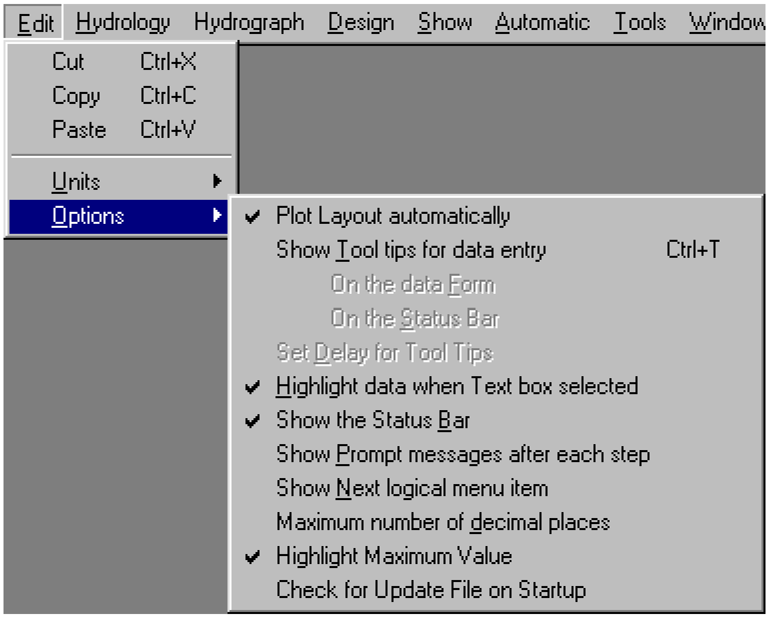


Figure 1-6 – The Edit / Options menu

Before you can proceed to any hydrological modelling you must set the Time parameters and this should be your next step.

1.3.4 Set the Time Parameters

In the **Hydrology** menu all of the options are initially disabled with the exception of the **Time parameters**. This command prompts you to set the time step, maximum storm duration and longest hydrograph duration you expect to use. All of these values are defined in minutes. Be generous with the estimate of storm duration and hydrograph length because if you find you want to define a storm longer than you originally thought a second use of the **Hydrology/Time parameters** command will cause the arrays storing rainfall and hydrographs to be re-initialized with loss of data.


TIME PARAMETERS

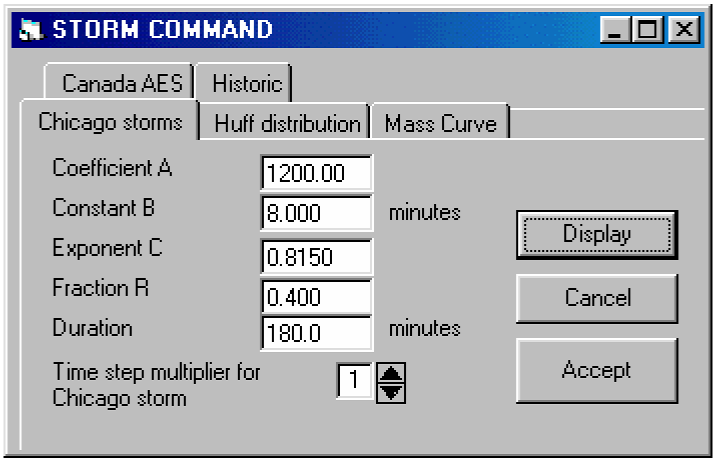
Time Step 5 minutes
Max. Storm length 180 minutes
Max. Hydrograph 1500 minutes

Figure 1-7 – The Time Parameters window

When you accept the time parameters the **Options/Units** menu item is disabled. This means that you cannot change the system of units in the middle of a design session. Setting the time parameters will also enable the **Storms** command and your next logical step will be to Define the Design Storm.

1.3.5 Define the Design Storm

Use the **Hydrology/Storms** command to define a single event design storm using one of the five methods shown in the figure and listed below.



Coefficient A 1200.00

Constant B 8.000 minutes

Exponent C 0.8150

Fraction R 0.400

Duration 180.0 minutes

Time step multiplier for Chicago storm 1

Figure 1-8 – The Storm Command window

- a Chicago storm,
- one of the four Huff distributions,
- any of the pre-defined mass rainfall distribution (*.mrd) patterns,
- a storm pattern as proposed by the Canadian Atmospheric Environment Service (AES), or
- a historic storm

Alternatively, you could import a previously created storm hyetograph file by means of the **File / Load file / Rainfall.**

Either method causes the **Hydrology /Catchments** command to be enabled. The rainfall defined in this way remains in force until the rainfall is redefined. Normally the storm is defined only once at the start of the session.

Once the storm is defined the next step is to generate the Runoff Hydrograph for the first catchment area.

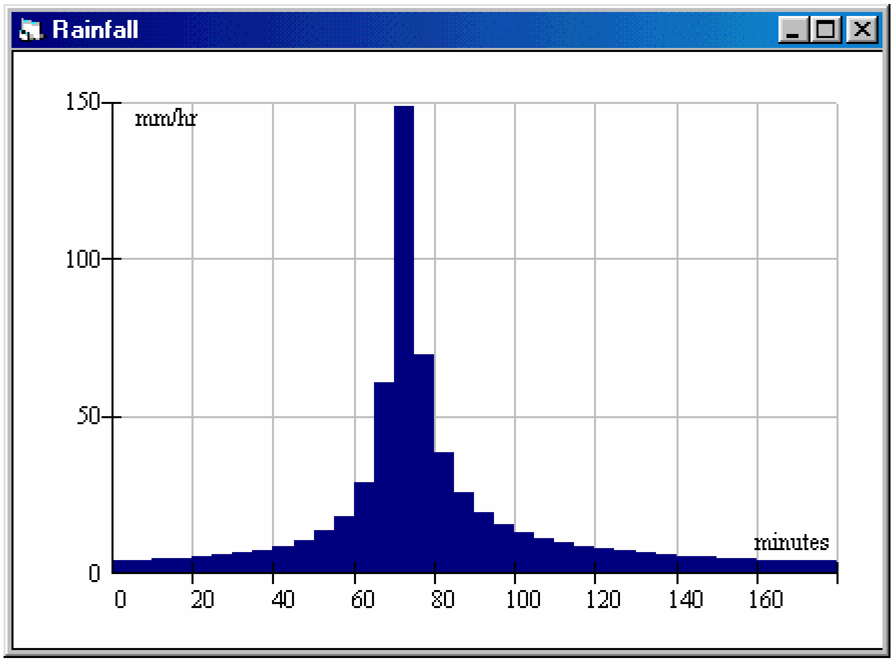
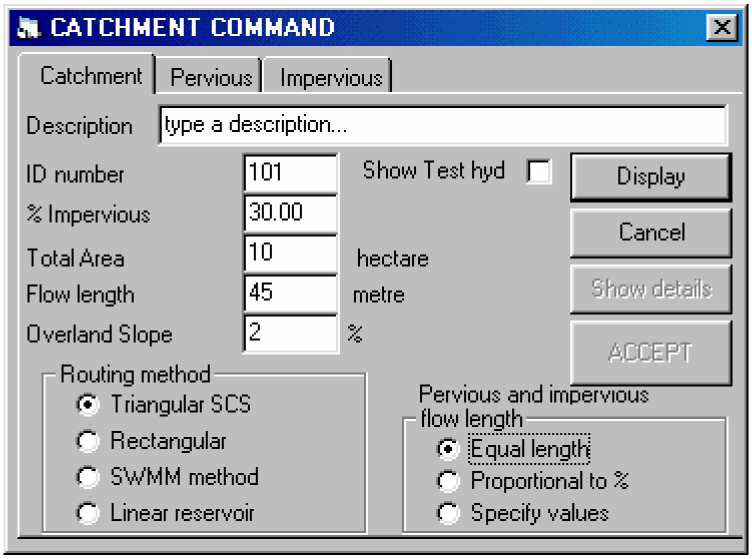


Figure 1-9 – The Chicago storm hyetograph

1.3.6 Generate the Runoff Hydrograph

Using the **Hydrology/Catchments** command you can now define the first catchment and produce the direct runoff hydrograph for the previously defined storm event. The runoff hydrograph is stored in the Runoff hydrograph array.



Description: type a description...
ID number: 101
Show Test hyd ☐
% Impervious: 30.00
Total Area: 10 hectare
Flow length: 45 metre
Overland Slope: 2 %

Routing method
- ● Triangular SCS
- ○ Rectangular
- ○ SWMM method
- ○ Linear reservoir

Pervious and impervious flow length
- ● Equal length
- ○ Proportional to %
- ○ Specify values

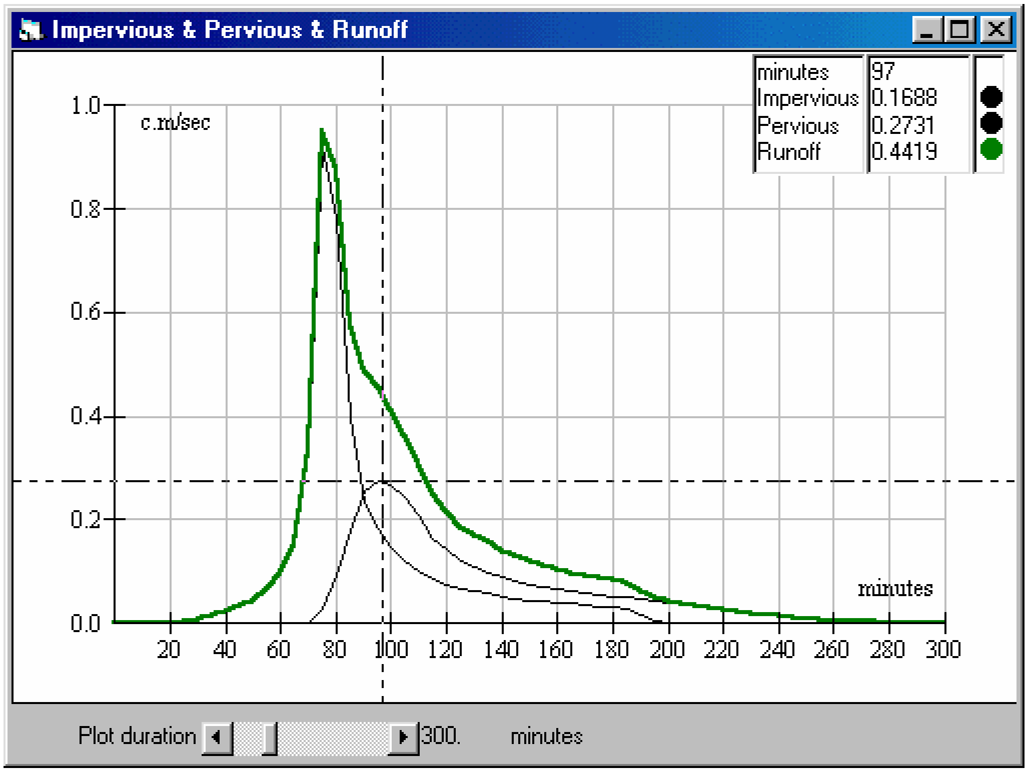
Figure 1-10 – The Catchment form has three tabs

The runoff is computed separately for the pervious and impervious fractions of the catchment and the two hydrographs are added. Information about the catchment runoff is provided in various ways;

- A table is displayed with the flow for each time step together with the peak flow and the total volume of runoff.
- A graphical display shows the runoff from the pervious and impervious fractions as well as the total runoff.
- A table of rainfall and runoff measures can be displayed by clicking on the [Show Details] button.
- Another summary table shows the history of peak flows in each of the four arrays for Runoff, Inflow, Outflow and Junction hydrographs, but this is not updated until you have accepted the results of the calculation by pressing the [Accept] key.

Accepting the computed runoff hydrograph causes the menu command **Hydrograph /Add Runoff** to be enabled. Using this the Runoff is added to the current Inflow hydrograph, which in this case is initially zero. The summary table of hydrograph peak flows is updated

Before you can use this hydrograph to design an element of the drainage network you must add the runoff to the current Inflow hydrograph. This is done by using the **Hydrograph/Add Runoff** command.



minutes	97	
Impervious	0.1688	●
Pervious	0.2731	●
Runoff	0.4419	●

Plot duration 300. minutes

Figure 1-11 – The Runoff hydrograph and its components.

1.3.7 Using the Add Runoff Command

The Hydrograph menu contains a number of options to manipulate flow hydrographs. These are initially disabled and become available to you as the pre-requisite steps are completed. Once a new runoff hydrograph has been created by means of the **Hydrology/Catchments** command the **Hydrograph/Add Runoff** command is enabled.

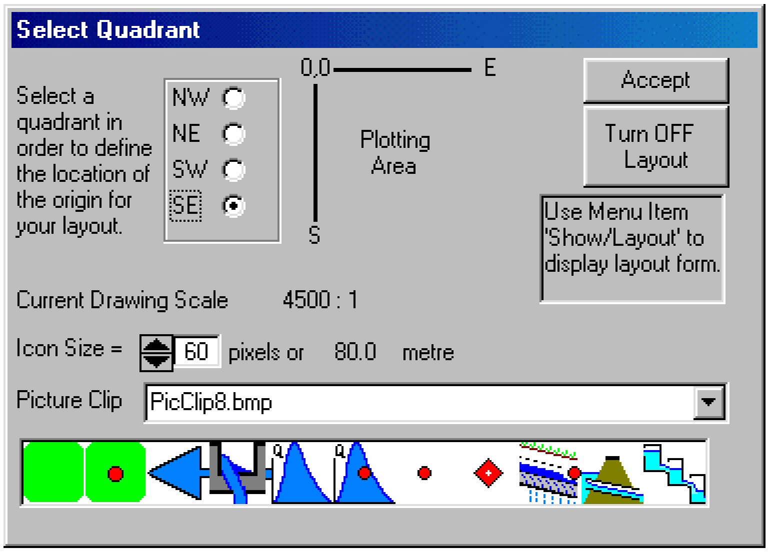
This command causes the last computed runoff hydrograph to be added to the current Inflow hydrograph. The result of this operation is shown in the summary of peak flows. The **Hydrograph/Undo** command is enabled by this action so that you can reverse the process should you wish to do so.

Now that the Inflow hydrograph has been updated the next logical step is to design an element of the drainage network such as a pipe or channel or other facility.

Prior to doing this MIDUSS wants to know the plotting direction you want to use for the Layout feature. This is known as Selecting a Quadrant.

1.3.8 Selecting a Quadrant

In the diagram below you can see that the origin (0,0) is set at the top left. Therefore the plotting of the layout will be towards the South-East. You can tell MIDUSS the direction you wish to use. At any point in the design session you can change the plotting direction and MIDUSS will re-calculate all the linked icons. Normally the SE direction is the most appropriate direction.



Select a quadrant in order to define the location of the origin for your layout.

NW ○
NE ○
SW ○
SE ●

Use Menu Item 'Show/Layout' to display layout form.

Current Drawing Scale 4500 : 1

Icon Size = 60 pixels or 80.0 metre

Picture Clip PicClip8.bmp

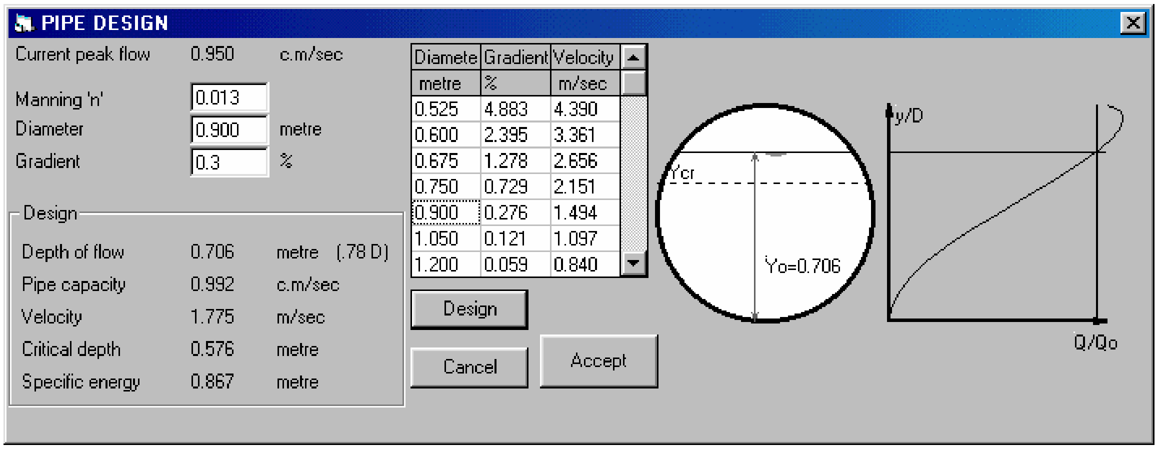
Figure 1-12 – Selecting a quadrant for the layout

Open the Design menu and select an item to design a pipe or channel .

1.3.9 Design a Pipe or Channel

Using one of the options in the **Design** menu, you can now design a pipe or channel to carry the peak flow of the Inflow hydrograph. For each of the **Design** menu items default data is displayed which you should change to suit your requirements. Your data values will become the new default values for the remainder of this session or until changed again.

Design of pipes or channels requires selection of a depth or diameter along with the gradient expressed as a percentage. Pressing the [Design] command button causes a uniform flow analysis to be displayed which shows the actual depth, the flow capacity of the conduit, the average velocity and the critical depth of flow to indicate whether the flow will be sub-critical or super-critical.



Current peak flow 0.950 c.m/sec

Manning 'n' 0.013
Diameter 0.900 metre
Gradient 0.3 %

Design
Depth of flow 0.706 metre (.78 D)
Pipe capacity 0.992 c.m/sec
Velocity 1.775 m/sec
Critical depth 0.576 metre
Specific energy 0.867 metre

Diamete	Gradient	Velocity
metre	%	m/sec
0.525	4.883	4.390
0.600	2.395	3.361
0.675	1.278	2.656
0.750	0.729	2.151
0.900	0.276	1.494
1.050	0.121	1.097
1.200	0.059	0.840

Figure 1-13 – A successful Pipe Design

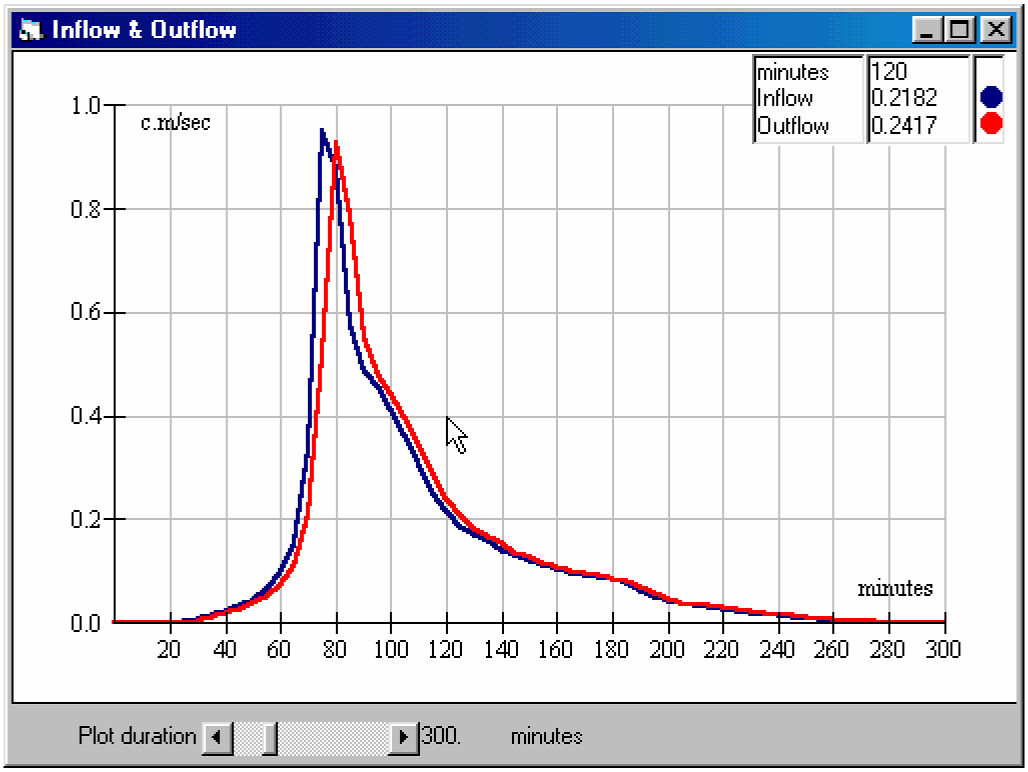
In the case of a pipe the design may result in surcharged conditions in which case the hydraulic grade (i.e. the slope of the energy line) is reported. Surcharged flow may result in only a portion of the runoff being captured so that the total runoff must be split between major (i.e. on the surface or street) and minor flows (i.e. in the pipe). If this happens you will be asked if you want to use the **Diversion** command to separate the inflow into the major and minor components.

Of course, options other than the **Pipe** and **Channel** design may be used but these simple cases are used for this introductory description.

When the design is accepted the **Design/Route** menu item is enabled which lets you route the flow hydrograph through a specified length of conduit.

1.3.10 Route the Hydrograph

The **Design/Route** menu command is used to route the hydrograph through a user-defined length of the most recently designed conduit. After accepting the result, the peak of the outflow hydrograph is displayed in the summary table of hydrograph peaks.



minutes	120	
Inflow	0.2182	●
Outflow	0.2417	●

Figure 1-14 – The Route command results in some lag

When an outflow hydrograph has been created by some routing operation you may choose from two possible courses of action. Either the outflow can be copied to the inflow array in order to continue to the next downstream link, or the outflow may be stored at a junction node to be combined with other flows at a confluence or junction node.

For this example you should assume that the outflow from the first conduit will form all or part of the inflow to the next downstream conduit. You can do this by using the **Hydrograph/Next Link** menu command in the **Hydrograph** menu.

1.3.11 Using the Next Link command

When a new outflow hydrograph is created, the **Hydrograph/Next Link** menu item is enabled. Using this command causes the outflow hydrograph to be copied to replace the inflow hydrograph and ready to receive the runoff contribution from the next sub-catchment. If no additional catchment runoff enters at this point you can simply use the inflow to design another link in the drainage network such as a detention pond, an exfiltration trench or a diversion structure

For this example a second catchment area must be defined and the runoff added to the total inflow. The procedure for modelling the second catchment is similar to the previous case. If the parameters describing the rainfall losses and infiltration are unchanged, the effective rainfall for the pervious and impervious areas will also be unchanged and only the calculation of runoff is required. One exception to this rule is when using the SWMM Runoff procedure to model the overland flow. Refer to Chapter 7 *Hydrological Theory* – The SWMM Runoff Algorithm for more information.

Once the runoff from the second catchment has been accepted you can use the **Add Runoff** command a second time to accumulate the flow in the inflow hydrograph.

1.3.12 Add Runoff Hydrograph #2

The **Hydrology /Catchment** is used a second time to generate the runoff hydrograph from the next catchment area. The **Hydrograph /Add Runoff** command causes this to be added to the Inflow hydrograph. Both changes are reflected in the summary table of peak flows.

For this simple example the next and final step is to design a detention pond to reduce the peak flow.

1.3.13 Design a Detention Pond

To reduce the peak of the Inflow hydrograph, the **Design/Pond** menu command is used to design a detention pond. The process requires the calculation of the Stage - Discharge - Storage Volume data for the pond. A number of options are available in the **Design/Pond** command to let you describe different parts of the outflow control device. These may comprise multiple orifices and weirs from which the Stage - Discharge curve is computed. Options are also available to help you calculate the Stage - Volume Storage curve for a few standard forms of storage geometry such as an idealized pond of rectangular shape, large size "super-pipes", wedge storage on graded parking areas or rooftop storage.

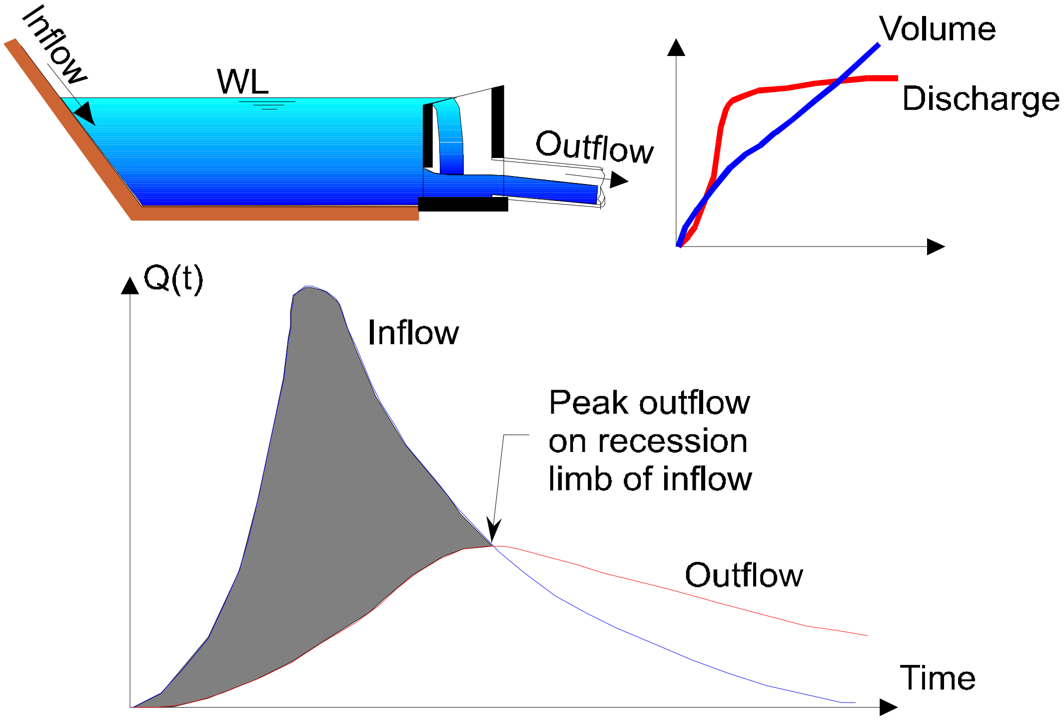


Figure 1-15 – A schematic of storage routing

When you use the **Design/Pond** command you can specify the desired peak outflow and MIDUSS will suggest values of storage and other parameters which will provide an initial design. You can then fine-tune the design until you are satisfied with the result. Since this is a trial and error procedure the Design Log feature of the **Design** options is useful in summarizing the progress of this iterative procedure.

The Outflow hydrograph is obtained by a storage routing procedure and the attenuated peak flow is displayed in the table of peak flows. MIDUSS will automatically adjust the storage routing time-step to ensure numerical stability in the vicinity of highly nonlinear Discharge - Storage Volume curves. Any such adjustment is reported but otherwise the process is transparent to the user.

This concludes this simple illustrative example but you may find it useful to review the summary of modelling procedure which follows. A more detailed example is presented in the Tutorial Manual.

1.3.14 Summary of Modelling Procedure

The total catchment area is subdivided into a series of sub- catchments, each of which generates an overland flow or runoff hydrograph. The hydrograph is assumed to enter the drainage network at a particular point or node associated with the sub-catchment. The drainage network is assumed to be a tree so that each node can have any number of inflow links but only one outflow link. For this reason, each link is given the same number as the node at its upstream end. These sub-catchments are processed in order, starting at the upstream limit of sewer branches and working in the downstream direction.

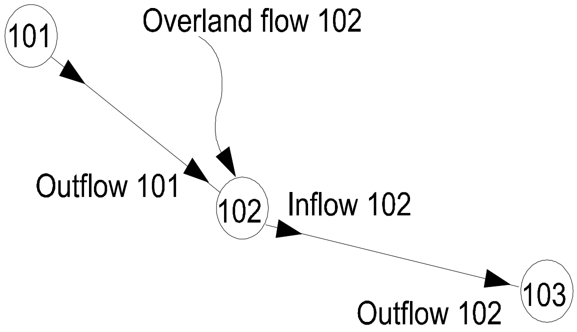


Figure 1-16 – Convention used for Numbering Nodes and Links

Figure 1-16 illustrates a node (102) which receives the outflow from the upstream link (101), adds to this the runoff from catchment (102) and creates the resulting inflow hydrograph to be carried by the downstream link (102). The node number is defined as the ID number of the catchment and sets the reference number for the element of the drainage network from this point to the next downstream node.

The links or branches of the network connecting the nodes are assumed to be pipes, channels, detention ponds, exfiltration trenches or other structures. As each runoff flow hydrograph is added to the flow in the system, a pipe, channel, pond, etc. can be proportioned to carry the peak inflow. Once this element has been designed the inflow hydrograph is routed through the link to produce an outflow hydrograph.

These steps are repeated, moving downstream, accumulating overland flow from each sub-catchment or storing an outflow hydrograph in order to design another branch meeting at a confluence point. Refer to Chapter 5 – *Hydrograph Manipulation* for more details.

Notes:.

Chapter 2 Structure and Scope of the Main Menu

Figure 2-1 – The Main Menu with all items enabled

When you first start MIDUSS a number of the items in the Main Menu will be grayed out or disabled. This indicates that some prerequisite information has not yet been defined. For example, the Storm command is not enabled until you have defined the time step and storm and hydrograph duration to be used. Similarly, the Catchment command cannot be used until a storm has been described. In the Design options, components such as a detention pond or ex-filtration trench require an Inflow hydrograph to have been created.

You will notice an apparent contradiction to this general rule. In the Pipe, Channel and Culvert commands, it is possible to use these without having previously computed the inflow hydrograph. If no inflow hydrograph exists you can still use the Pipe, Channel or Culvert command but you will have to enter a value for the peak flow instead of having that supplied as the peak flow of the inflow hydrograph. This allows you to use the Pipe and Channel commands for design purposes without having to go through any hydrological modelling.

File	Open files, print hardcopy, save and load files, and quit MIDUSS
Edit	Use the Clipboard, select units, and other preferences
Hydrology	Define storms, catchments etc. to model the rainfall-runoff process
Hydrographs	Manipulate hydrographs
Design	Design various facilities for stormwater management
Show	Choices to display the results in tabular or graphic form
Automatic	Steps to run in Automatic mode
Tools	Auxiliary applications to run from MIDUSS
Window	Options to show windows
Help	How to use Help, play Tutorial lessons

2.1 The File Menu

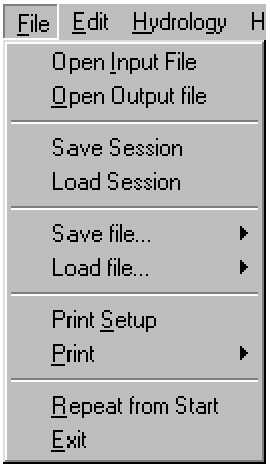


Figure 2-2 – The File Menu Options

Open Input File	Create an input database Miduss.Mdb using a previously created Output file for use in Automatic mode.
Open Output File	Specify and open an Output file, either creating a new file or overwriting an existing one.
Save Session	Save a file that preserves the current data and status of a MIDUSS session
Load Session	Reload a previously saved 'snapshot' of MIDUSS data in order to resume a design session.
Save File	Save and Write a hyetograph or hydrograph file
Load File	Read and Load a previously saved hyetograph or hydrograph file.
Printer Setup	Set printer parameters and preferences.
Print	Select a scaling factor and print the current screen contents
Repeat from Start	Automatically converts the current output file to an input database Miduss.Mdb and initiates a run in automatic mode. This lets you alter the storm – typically increasing the duration or reducing the average intensity – and then repeat the previously created design under the modified storm.
Exit	Close all files and exit from MIDUSS

Refer to The Automatic Mode later in this chapter or Chapter 10 *Running MIDUSS in Automatic Mode* for information on creating and using the Input database Miduss.Mdb.

2.1.1 Input File Command

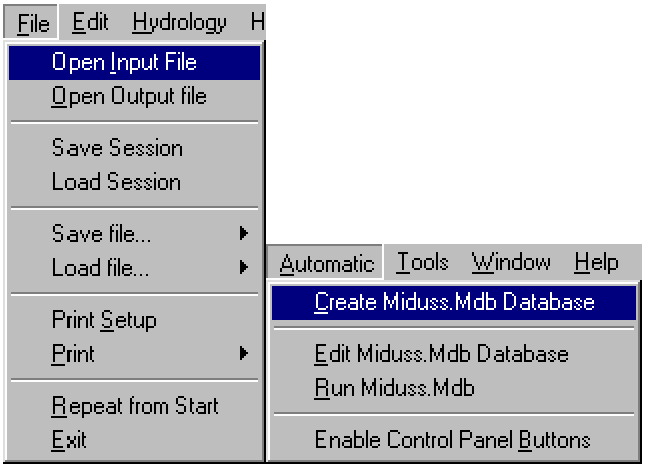


Fig 2-2a showing both File/Open Input file and Automatic / Create Miduss.Mdb Database

This command lets you create an input database called Miduss.Mdb from a previously created output file . Both the output file and the database reside in the current job folder.

The same process can also be carried out by using the menu item **Automatic / Create Miduss.Mdb Database**. Both menu items are shown above in Figure 2-2a.

If the job folder already contains a file called Miduss.Mdb a warning message is displayed that lets you either abort the command or overwrite the existing file.

2.1.2 Output File command

Figure 2-3 – Using the File Menu to define an Output File

During a MIDUSS run the commands used, the data entered and some of the results are always copied to an output file. If you do not specify an output file in a particular Job Directory the output goes to a file called Default.Out. This menu item allows you to specify an alternative to the default file.

You can choose between creating a new file or selecting an existing file. If you use the name of an existing file you will be warned that the file will be overwritten and the contents will be lost.

In both cases you select the directory and enter or select a file name by means of the standard File Open dialogue box. You can use long file names to relate the file to the design situation -

e.g. "PineView_5yr_post-dev.out"

2.1.3 Save Session

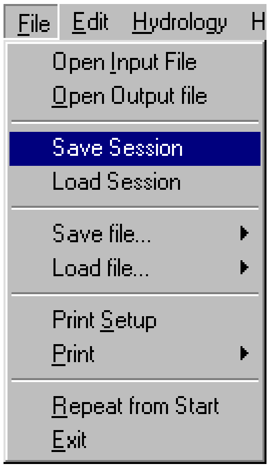
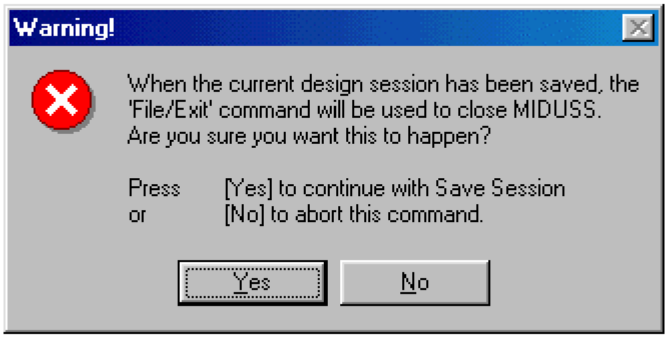


Figure 2-4 File/Save Session menu

This command saves the current state of all variables and arrays of data in a single binary file. This file is given the same name as the current output file with the additional extension '.BIN' added to it. Thus if the output file is called 'Test1.out' the binary file will be named 'Test1.out.bin'. The command is initially disabled (grayed out) when MIDUSS starts but is enabled once any significant data is entered.

The binary file also contains a copy of the output file as well as all data required to reconstruct the Layout diagram if one has been generated and the full contents of the Qpeaks summary table. Data used by the various items in the Tools menu is not saved in the binary file.

It is not possible to continue the MIDUSS design session after using the Save Session command. When you click the **File/Save Session** menu a warning message is displayed telling you that the **File/Exit** command will be automatically invoked after the session data has been saved. If you do not want to do this you can abort the command and continue with the MIDUSS session.


Warning!

When the current design session has been saved, the 'File/Exit' command will be used to close MIDUSS.
Are you sure you want this to happen?

Press [Yes] to continue with Save Session
or [No] to abort this command.

Figure 2-5 Warning message when Save Session clicked

2.1.4 Load Session

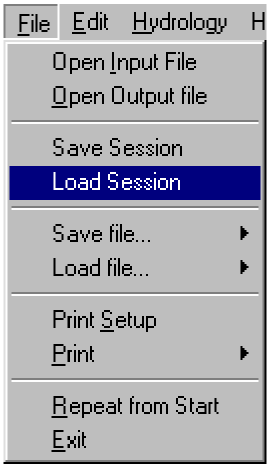
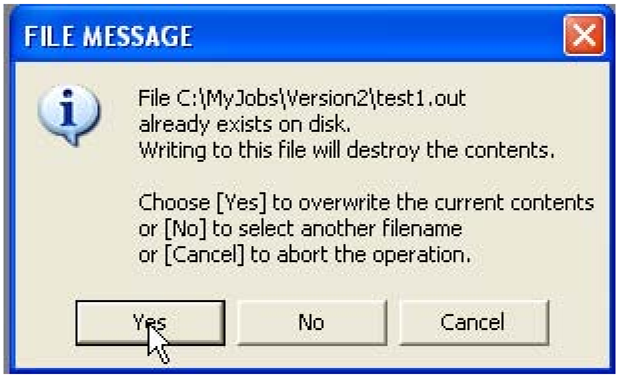


Figure 2-6 File/Load Session menu

This command must be used immediately after the main menu is displayed. You should not define an output file or start any hydrology modelling or design.

A File Open dialogue box is displayed showing all files with the extension '.BIN' in the job folder that was last used. You can, of course, navigate to any other folder but after a selected session has been loaded the folder will become your working job directory. This occurs, for example, if you use one of the 'Rescue' files that are distributed with the Tutorial manual.

Assume that you select a session file with the name 'Test1.out.bin'. This file was produced at the end of a session that created the output file 'Test1.out'. When you click the [Open] button of the File Open dialogue box you will see a warning advising you that an output file called Test1.out already exists and will be overwritten by the output file created by the **Load Session** command.


FILE MESSAGE

File C:\MyJobs\Version2\test1.out
already exists on disk.
Writing to this file will destroy the contents.

Choose [Yes] to overwrite the current contents
or [No] to select another filename
or [Cancel] to abort the operation.

Figure 2-7 Warning msg – output file will be overwritten

The message box gives you three choices to do one of the following.

- Accept this (the contents will initially be identical but you will probably be adding to the file)
- Select another name for the output file that will be created by the Load Session command and which will be subsequently added to during the continued design session.
- Abort the Load Session command

It is important to note the difference between using the Load Session command and using the output file to repeat the design session in Automatic mode. In the latter case, you can edit any of the data or actions taken prior to the point at which a session 'snapshot' may have been taken.

2.1.5 Save File

Saves a file containing a storm or effective rainfall hyetograph or any of a number of flow hydrographs.

2.1.6 Load File

Reads a file and loads a data array defining a rainfall hyetograph or flow hydrograph.

2.1.7 Print Setup

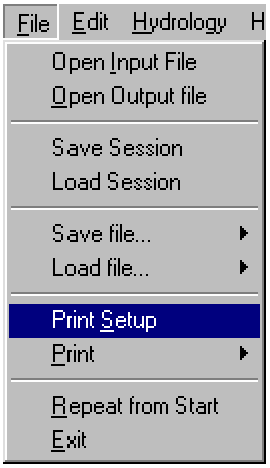


Figure 2-8 File/Print Setup menu

This command opens up the standard Windows dialog box to set up certain parameters for your printer. When you press [OK] a message box is displayed giving you the option to print either the full screen or the MIDUSS window. See the **Print Command** for further details.

2.1.8 Print command

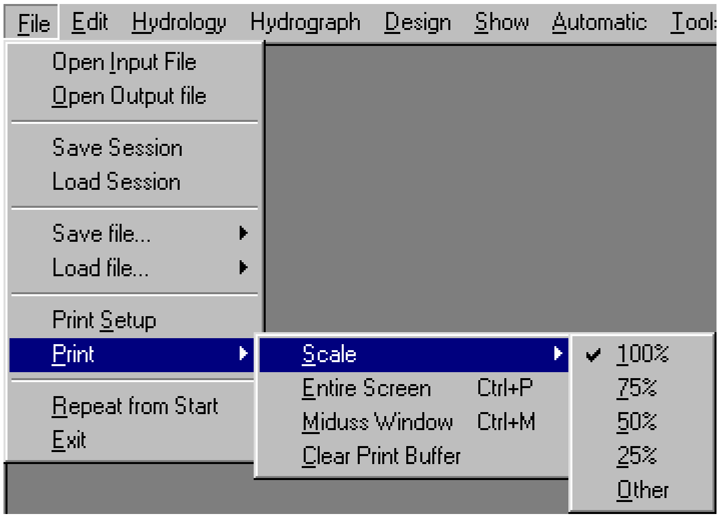


Figure 2-9 – Options available using the File/Print command

As illustrated in Figure 2-9, the print command contains a number of sub-commands which let you:

- Set the scale of the hardcopy relative to the page size
- Print the entire screen
- Print only the MIDUSS window
- Clear the buffer containing data to be printed

When you select the **Scale** command you can choose from a number of standard ratios such as 100%, 75%, 50% & 25%. If none of these is suitable you can specify another ratio which will be saved for future use during the current session. After a scale has been selected the **Print** menu item is modified to show the currently selected scale.

2.1.9 Repeat from Start

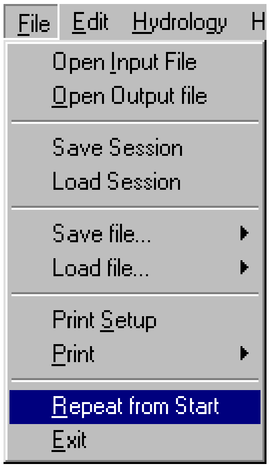


Figure 2-10 File/Repeat from Start menu

This command is intended to let you change the storm at some point in a design session and then repeat the entire design with this new storm up to the point where the storm is changed.

Typically the change in the storm is to increase the duration to allow for an increase in the time of concentration as the design proceeds down the drainage network. With many design storms, an increase in duration causes lower rainfall intensity. Thus pipes etc. designed for the original, more intense storm will be slightly over-designed.

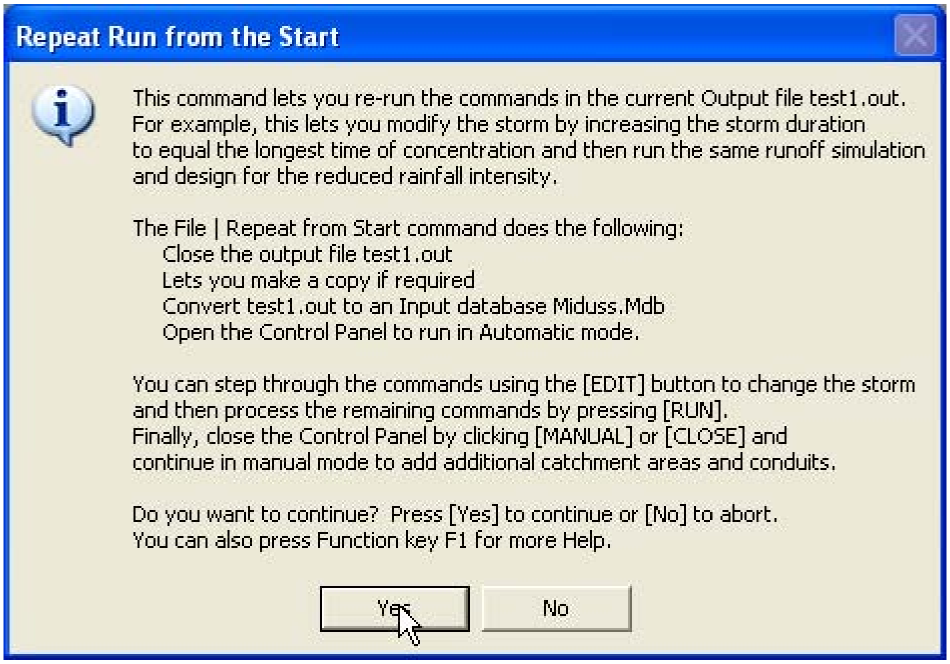
Normally, this command will be invoked after one or more upstream catchment areas have been analyzed and the resulting runoff has been routed down the drainage network to a point where the Time of Concentration is now greater than the duration of the current design storm.

This initial stage will allow design or sizing of drainage elements that must convey the runoff resulting from short, intense storms.

The Repeat from Start command allows you to redefine a storm with a longer duration and a reduced average intensity. However, once a new storm is defined, you cannot simply continue from this point in the drainage network as it would result in runoff hydrographs being combined which have been generated from different storm events.

Instead, once the storm is changed, the rainfall-runoff modelling must be repeated from the furthest upstream point. The runoff will be slightly reduced but no change should be made to the design decisions (e.g. pipe diameters) made with the initial storm.

The Repeat from Start command carries out all of the necessary steps automatically with the exception of defining the parameters of the modified storm.



This command lets you re-run the commands in the current Output file test1.out. For example, this lets you modify the storm by increasing the storm duration to equal the longest time of concentration and then run the same runoff simulation and design for the reduced rainfall intensity.

The File | Repeat from Start command does the following:

Close the output file test1.out
Lets you make a copy if required
Convert test1.out to an Input database Miduss.Mdb
Open the Control Panel to run in Automatic mode.

You can step through the commands using the [EDIT] button to change the storm and then process the remaining commands by pressing [RUN]. Finally, close the Control Panel by clicking [MANUAL] or [CLOSE] and continue in manual mode to add additional catchment areas and conduits.

Do you want to continue? Press [Yes] to continue or [No] to abort. You can also press Function key F1 for more Help.

Figure 2-11 large warning message

The warning message summarizes the various steps that will be carried out and gives you the opportunity to either continue or abort the process. The steps to be carried out are summarized below.

(1) The current output file must be closed but you can save a copy of the file with a different filename extension for reference. MIDUSS suggests a simple change by adding a letter ‘A’, ‘B’, etc. to the existing extension.

(2) After closing the current output file, it is re-opened as an input file and an Input Database Miduss.Mdb is created. No action is required by you but you will see briefly the progress window as the database is generated.

(3) The file containing the table of peak flows (Qpeaks.txt) must be closed because these flow values will be regenerated for the modified storm. A message is displayed for information only. Should you wish to keep a copy of the Qpeaks.txt file you can use the Show / Flow Peaks file command (or Ctrl+Q) and from Notepad save the file by another name.

(4) A message is displayed advising you that MIDUSS will now run in Automatic mode and suggesting that you use the [EDIT] button to advance to the Storm command and after modifying the storm you can press the [RUN] button to process the remainder of the input database. This message is also for information only.

(5) The Control Panel is opened with the new database loaded ready for a run in Automatic mode.

If you press the [Yes] button the above process will start. You can abandon the process by clicking on the [No] button.

If the Layout window is active the network drawing is deleted and reconstructed during the Automatic session.

2.1.10 File Exit

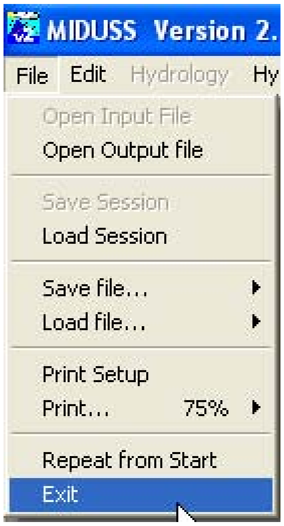


Figure 2-12 File/Exit menu

This command ends the MIDUSS session. Files created during the session will be closed and the current Options in effect will be saved for use as the initial defaults in the next session.

Two files may be worth saving before your next run of MIDUSS.

(a) If you did not specify a Job directory, the output will be stored in the default output file C:\Program Files\MIDUSS\default.out. If you want to keep this you must rename it or copy it to another file or another directory. The default output file will be overwritten at the start of the next MIDUSS session which does not use a specified Job directory.

(b) The file C:\Program Files\MIDUSS\Design.log will contain a record of any design commands which you used. This may be of use in interpreting the output file or for making comparisons during your next MIDUSS session.

Sometimes if an error occurs during the MIDUSS session you will see a message that a file Miduss.Log has been created. This will contain information of errors that were trapped by MIDUSS. It would be useful if you could forward this file to Alan A. Smith Inc. either by e-mail or by Fax so that the cause of the problem can be corrected.

2.2 The Edit menu item

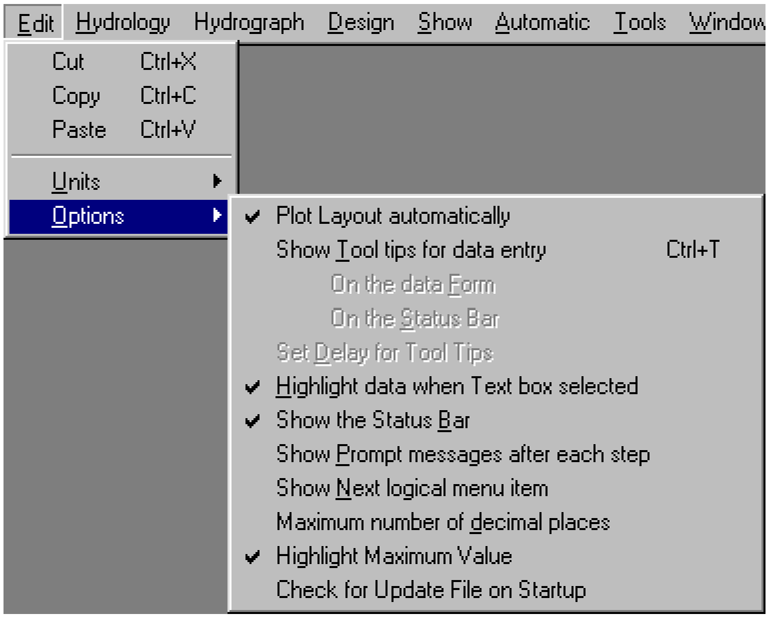


Figure 2-13 – The Edit Menu with Options selected

The Edit menu is illustrated in Figure 2-13 and shows three main choices available. These are:

Cut / Copy / Paste	Clipboard commands.
Units	Displays the system of units in force. Selection of units is done at the startup of MIDUSS when the option form Figure 1-5 form is displayed.
Options	A variety of interface choices are presented. A check mark beside a selection indicates that it is active.

2.2.1 Edit / Cut Copy Paste

The Cut, Copy and Paste commands let you use the Clipboard to transfer data between MIDUSS and any other Windows application that supports the use of the Clipboard in this way. Programs such as Microsoft Excel can be used to prepare data for use in MIDUSS by copying data from the spreadsheet to a data grid. Where possible, MIDUSS will adjust the incoming data to match the format of the MIDUSS data input grid.

The Edit commands Cut, Copy and Paste can be called using the normal Windows shortcut keystrokes, i.e.

Edit Cut	Ctrl + 'X'	Copy selected text to Clipboard and delete source
Edit Copy	Ctrl + 'C'	Copy selected text to Clipboard; source unchanged
Edit Paste	Ctrl + 'V'	Copy Clipboard text to destination cell.

If the Clipboard contains multiple cells the Paste operation checks to ensure that there is enough space available at the destination and then copies the text to the right and below the target cell. Data is copied by rows as defined in the destination grid.

In certain Design commands (e.g. Pond, Trench) the Main Menu is not visible when the Design window has the focus (i.e. has a highlighted title bar). The specialist menu in these commands has been modified to show the three Edit options for local use. The shortcut commands can also be used when the Main Menu is not visible.

2.2.2 Edit Units

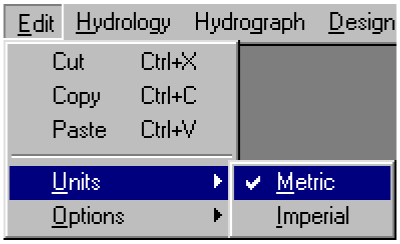
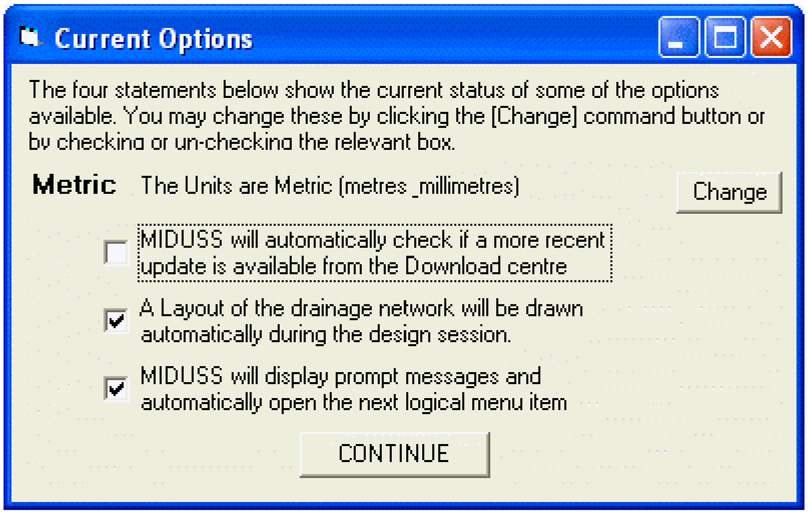


Figure 2-14 Edit/Units menu Metric selected

On selecting the Units item you will notice that the select choices are grayed out. There is a check mark beside the system of units currently in force.

To select the system of Units (either Metric or Imperial) you need to do so at the start of MIDUSS when the following screen appears.



Current Options

The four statements below show the current status of some of the options available. You may change these by clicking the [Change] command button or by checking or un-checking the relevant box.

Metric The Units are Metric (metres _millimetres) Change

☐ MIDUSS will automatically check if a more recent update is available from the Download centre

☑ A Layout of the drainage network will be drawn automatically during the design session.

☑ MIDUSS will display prompt messages and automatically open the next logical menu item

CONTINUE

Figure 2-15 Window showing 'Current Options'

On this screen the currently in force system of units is displayed. In this case Metric units are being used. To change to Imperial (US Customary) click the Change button.

In the metric system the unit of length is metre or millimetre depending on the variable being defined. In the Imperial system (also known as U.S. Customary units) length is defined in feet or inches.

In MIDUSS, only kinematic units are employed, i.e. only length, area, volume and time dimensions are used. The Table below shows the units employed for the various quantities used in hydrology and in the design of storm water management facilities.

Table 2.1

Variable dimensions used for SI and Imperial units

QUANTITY	**Imperial (US Customary)**	**Metric (SI units)**
Time	seconds, minutes or hours	
Rainfall depth	inches	millimetres
Rain intensity	inches/hour	millimetres/hour
Catchment	Acres	Hectares
Length	Feet	Metres
Diameter	Feet	Metres
Surface area	square feet	square metres
Velocity	feet/second	metres/second
Flow rate	cubic feet/sec	cubic metres/sec
Volume	cubic feet	cubic metres
	acre-ft	hectare-metre

When MIDUSS is first installed on your computer the default system of units is the metric system. However, if you change the selection your choice will be recorded in the system registry and will form the new default value for future sessions.

2.2.3 Edit Options

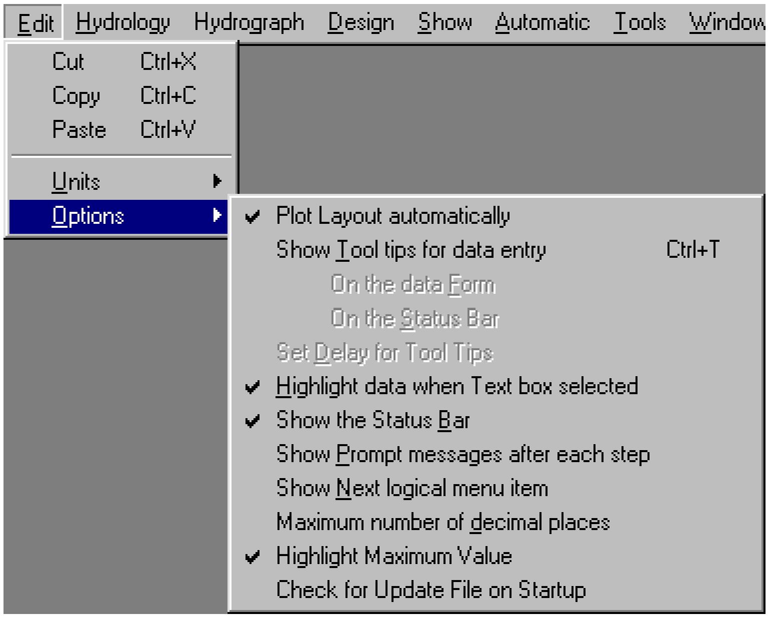


Figure 2-16 – The Edit Options Menu

As illustrated in Figure 2-16, the available options with the current release of MIDUSS are as follows.

Plot Layout automatically	When this option is selected MIDUSS will automatically build a Layout diagram showing the nodes and connecting links of the drainage network. The drawing area can be displayed by using the **Show/Layout** menu command. MIDUSS positions the elements in a selected coordinate system in accordance with the connectivity of the design, but you can manually adjust the position of elements (and their connected links) by selecting and dragging one or more nodes. If desired, a background diagram (such as a lotting fabric) can be imported to improve visualization of the drainage network. When you create a node that generates runoff or inflow you will be prompted to select a particular quadrant (SE, NE, NW or SW) that sets the location of the origin for the X,Y coordinates. Refer to the Show/Layout menu command or the video Tutorial “Using the Layout Feature” for more details. In addition to the aesthetic benefits, the Layout feature can be useful in identifying errors in connectivity in a complex network.
Show Text Box Tips.	Selection of this option triggers a display of a brief explanation of the data item when the mouse pointer is over a text box; sometimes a range of typical values is displayed. The tip can be displayed in a small yellow window adjacent to the text box or in the status bar by selecting one or other of the two options shown below.
On Form	
On Status Bar	
Set Delay for Text Box Tips	Specify the delay in milliseconds before the tip is shown.

	These menu choices are disabled (grayed out) if **Show Text Box Tips** is not selected. If **Show Text Box Tips** is selected, the active menu item is shown with a check-mark. You can also toggle between showing or not showing the Text Box Tips by pressing Ctrl-T. (i.e. Hold down the Control key and press 'T'). You will probably find the Text Box Tips option useful during your first few sessions with MIDUSS. When you are familiar with the data requirements it can be turned off.
Highlight data when Text box selected	When this option is selected it causes the current contents of a text box to be highlighted when the text box receives the focus. You may find this preferable since it allows you to start typing a value from the left character position instead of having to use the backspace key or an arrow key to position the text entry marker at which keystrokes will be entered. Note that this option does not apply to data entry in a grid.
Show Status bar	When this menu item is checked the status bar is enabled at the bottom of the MIDUSS window. The status bar contains information about the current menu selection, the job directory, input and output files.
Show Prompt messages after each step	This causes a brief message to be displayed after many commands to advise you on the action taken and suggest one or two logical next steps. When used together with the **Show Next logical menu item** option these options provide useful guidance if you are unfamiliar with the MIDUSS program.
Show Next logical menu item	After completion of a command, and after display of the **Show Prompt messages after each step** message if selected, this option causes the next logical menu item to be displayed with the mouse pointer positioned over the appropriate command as a guide to the user.
Maximum number of decimal places	In some displays of data or results you may require to increase the number of figures after the decimal point to get sufficient accuracy. Selecting this command opens a simple data entry window with a prompt to enter a single digit specifying the maximum number of places.
Highlight Maximum Value	In many commands a rainfall hyetograph or flow hydrograph is displayed in tabular form. Selecting this option causes the cell containing the maximum value to be coloured light cyan.
Check for Update file on Startup	When this option is selected MIDUSS will check for the availability of an update version that is more recent than the one that is running.

2.3 The Hydrology Menu

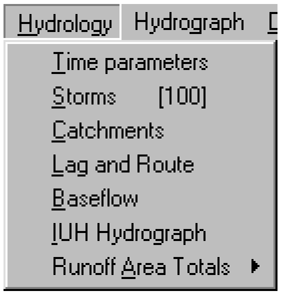


Figure 2-17 – Commands for Hydrologic simulation.

The Hydrology menu allows you to set the:

Time parameters	Set time step, maximum durations for storms and hyetographs.
Storms	Define a storm hyetograph of various types.
Catchments	Define catchment characteristics and calculate runoff.
Lag and Route	To allow very large sub-catchments to be modelled with realistic overland flow lengths
Base Flow	Add a constant flow rate to the inflow hydrograph (may be positive or negative).
IUH Hydrograph	Define a runoff hydrograph in terms of peak flow and time to peak (or duration).
Runoff Area Totals	Clear or Accumulate catchment areas.

2.3.1 Hydrology Time Parameters

Time Step 5 minutes
Max. Storm length 180 minutes
Max. Hydrograph 1500 minutes

Figure 2-18 – The default time parameters

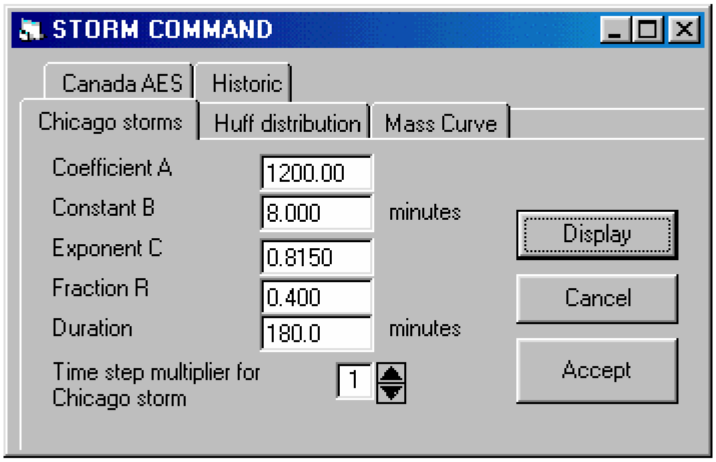
When you start MIDUSS this is the only item in the Hydrology Menu which is enabled. None of the other options can be used until you have specified the three time parameters shown here. Many of the items in the MIDUSS main menu are also disabled until the time parameters have been specified.

The specification of the time parameters should be preceded only by the specification of the system of units to be used and(if desired) the specification of a job-specific directory and output file.

The three time parameters are:

Time step	The time interval in minutes to be used for all rainfall runoff calculations and routing operations in pipes, channels, ponds or trenches. A value of 5 minutes is common but a smaller time step is seldom justified. Longer increments of 10 or 15 minutes may be used for long storms. Note that for some routing operations a sub-multiple of the time step is used to ensure numerical stability. If you intend to use the Chicago storm with a very small time step consider using the time step multiplier to avoid very large values of peak rainfall intensity.
Maximum storm duration	The longest storm duration which you expect to use during the current session. This sets the length of array used for hyetograph storage. A value of 3 to 6 hours (180 to 360 minutes) is common.
Maximum hydrograph length	The longest expected hydrograph duration for the current session. This sets the length of storage array used for hydrographs. A 24-hour hydrograph (1440 minutes) is not unreasonable.

2.3.2 Hydrology Storm



Coefficient A 1200.00
Constant B 8.000 minutes
Exponent C 0.8150
Fraction R 0.400
Duration 180.0 minutes
Time step multiplier for Chicago storm 1

Figure 2-19 – The five options available in the Storm command.

Use this menu option to specify a single event storm hyetograph. Typically this is done only once at the start of the session and applies to all of the catchment areas; however, there may be circumstances when a different storm may be required for a portion of the total area being modelled - e.g. higher elevation or a lagged, traveling storm.

The five tabs on the form provide five options to define four different types of design storm or a historic storm. In every case you are prompted to supply parameter values or use the default values displayed. Pressing the [Display] button causes a table of rainfall intensities and a graphical plot of the hyetograph to be displayed.

You can experiment by changing parameter values or even storm type and update the displays by pressing [Display]. When you are satisfied with the storm press the [Accept] button to cause the storm to be defined. This remains in effect until it is changed by a subsequent use of the command.

Note that you can use the **File / Load file / Rainfall** command to import a hyetograph file as an alternative to using the Storm command. Such files can be saved by means of the **File / Save file / Rainfall / Storm** command after a storm hyetograph has been defined – especially a long historic storm.

After you have accepted the design storm, an auxiliary window is opened prompting you to enter a string of up to 5 characters which is used as a descriptor of the storm just defined. This descriptor is used as part of the filename of any hydrographs that you may store as files during the current session. Refer to Hydrology/Storm Descriptor for more details.

See Chapter 3 *Hydrology Used in MIDUSS* Storm Command for more details.

2.3.3 Hydrology - Storm Descriptor

Supply up to 5 characters to serve as a file descriptor for hydrograph files created in this session.

005

Figure 2-20 – Defining a Storm descriptor

The window shown in Figure 2-20 is opened automatically after you have pressed the [Accept] button in the Storm command form. The string requested will be used as part of the name given to any hydrograph files which are generated in the current session using this design storm.

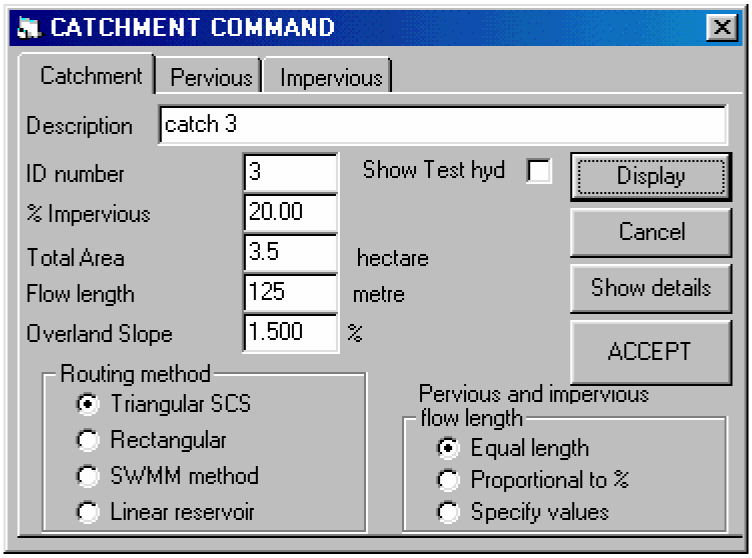
Typically you may enter the return period for the storm - e.g. "005" to denote a storm with a return interval of 5-years. When you save a hydrograph as a file the default extension used by MIDUSS is ".hyd". The filename which you provide has the Storm Descriptor added to this extension to give a modified extension of ".005hyd".

For example, if you specified a filename of "Pond7aOut" and the Storm Descriptor is "005" the final filename would be "Pond7aOut.005hyd".

Windows allows long filenames and also allows more than one 'period' separator. You can define the Storm Descriptor with a period at the end (e.g. "005."). Using the previous example, the resulting filename will be "Pond7aOut.005.hyd". However, you may find that this results in confusing abbreviations if the filename is displayed in a DOS window.

As a reminder of the descriptor that you have entered it is appended to the Storm item in the Hydrology menu. Refer to the **Hydrology** menu earlier in this chapter to see this (Figure 2-17).

2.3.4 Hydrology Catchment



Description: catch 3

ID number: 3

% Impervious: 20.00

Total Area: 3.5 hectare

Flow length: 125 metre

Overland Slope: 1.500 %

Show Test hyd ☐

Routing method
- ● Triangular SCS
- ○ Rectangular
- ○ SWMM method
- ○ Linear reservoir

Pervious and impervious flow length
- ● Equal length
- ○ Proportional to %
- ○ Specify values

Figure 2-21 – Data input form for the Catchment command

Once a design storm has been defined, you can use the Catchment command to generate the direct runoff hydrograph for a sub-catchment. The three tabs on the form are used as follows.

Catchment	Set the methods and parameters for the whole catchment and generate the overland flow hydrograph on the pervious, impervious and total areas.
Pervious	Set the rainfall loss model and the appropriate parameters to generate the effective rainfall on the pervious fraction of the area.
Impervious	Set the parameters for the impervious fraction and generate the effective rainfall on the impervious fraction of the area.
Description	This text entry box can contain a verbal description of the catchment area or merely an alpha-numeric identifier. A unique integer identifier ID number is still required. Both items appear in the output file. Each entry of an ID number is checked against a list of previously defined numbers to avoid duplication.
Show Test hyd	When this check box is selected (i.e. checked) the graphical display generated when the [Display] button is clicked includes the user-defined Test hydrograph as well as the hydrographs for total Runoff and the runoff components from the pervious and impervious fractions. This can be of value in calibrating runoff parameters to match an observed runoff hydrograph

The command contains a number of options for the overland routing method, the relative flow length on the pervious and impervious fractions of the catchment and the method to be used to estimate rainfall losses by infiltration, interception or surface depression storage.

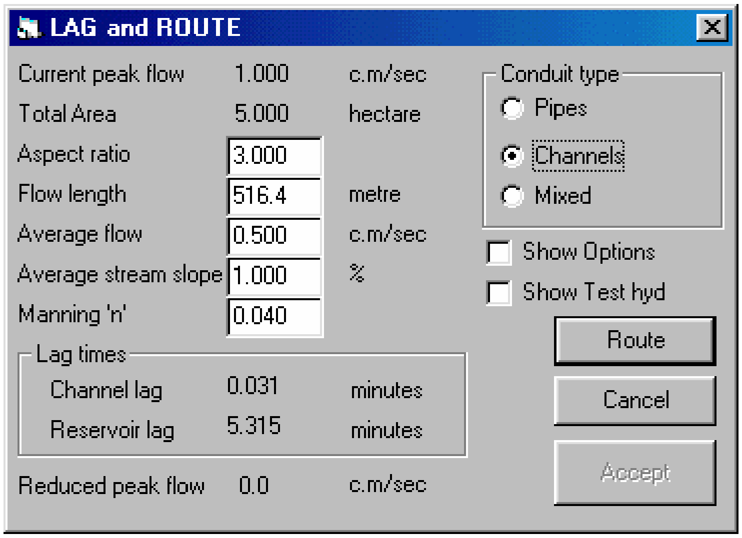
Before calculating the runoff it is necessary to compute the effective rainfall on both the impervious and pervious surfaces. This can be done explicitly by selecting the appropriate part of the form or it can be done automatically if you select the runoff calculation option.

Special conditions apply if you select the SWMM Runoff algorithm as the overland routing method since it uses a surface water budget method rather than an effective rainfall approach.

The command is described in full in the Hydrology section of this Help System. Refer to Chapter 3 *Hydrology Used in MIDUSS* Catchment Command for more details.

Note that you can use the **File / Load file / Hydrograph** command to import a hydrograph file as an alternative to using the Catchment command. Such files can be saved by means of the **File / Save file / Hydrograph** command after a runoff hydrograph has been defined.

2.3.5 Hydrology - Lag and Route



LAG and ROUTE

Current peak flow 1.000 c.m/sec
Total Area 5.000 hectare
Aspect ratio 3.000
Flow length 516.4 metre
Average flow 0.500 c.m/sec
Average stream slope 1.000 %
Manning 'n' 0.040

Lag times
Channel lag 0.031 minutes
Reservoir lag 5.315 minutes

Reduced peak flow 0.0 c.m/sec

Conduit type
○ Pipes
● Channels
○ Mixed

☐ Show Options
☐ Show Test hyd

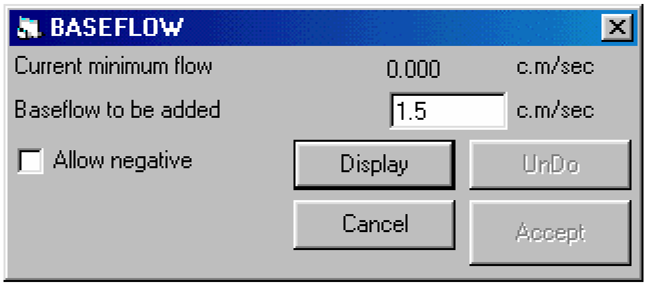
Figure 2-22 – Data input for the Lag and Route command

Calculating the runoff from a large catchment (e.g. more than 100 hectares or 250 acres) is complicated by the fact that the travel time of the runoff from sub-areas close to the outflow point will be much shorter than from sub-areas at the furthest upstream regions of the watershed or sewer-shed. This causes the incremental runoff hydrographs to have peak flows which are spread over time with the result that the peak outflow is significantly less than the sum of the constituent peaks.

MIDUSS incorporates a Lag and Route procedure which can provide an approximation to this effect. Although it is empirical and thus only approximate it is preferable to using unrealistically long overland flow lengths to provide an appropriate amount of peak attenuation. The method assumes that a hypothetical linear channel and linear reservoir are located at the outflow point of the catchment. The result is that the peak outflow is lagged in time and reduced in value.

The key to success is, of course to assign the correct lag times to the linear channel and linear reservoir. The empirical procedure used is described in Chapter 7 *Theory of Hydrology*. The procedure to use the Lag and Route command is described in Chapter 3 *Hydrology Used in MIDUSS* - Lag and Route Command .

2.3.6 Hydrology - Baseflow



Current minimum flow 0.000 c.m/sec

Baseflow to be added 1.5 c.m/sec

☐ Allow negative

Figure 2-23 – Adding a constant baseflow to the Inflow hydrograph

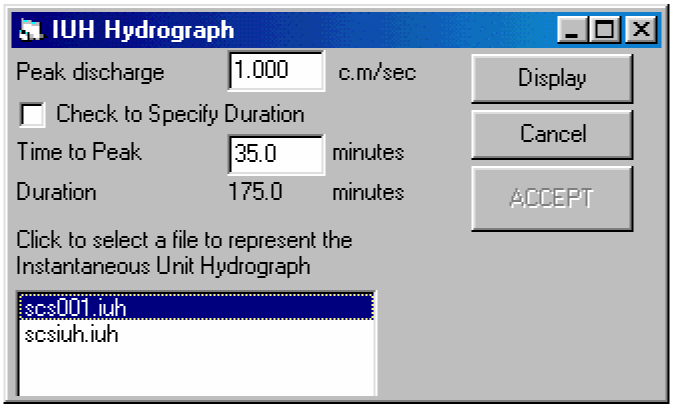
Even if the first element of the effective rainfall is finite, the direct runoff hydrograph starts with an initial value of zero. If you want to simulate the effect of baseflow in a drainage system you can add a constant flow value to the Inflow hydrograph.

The Baseflow command is enabled only after an Inflow hydrograph has been generated.

Refer to Chapter 3 *Hydrology Used in MIDUSS* - Baseflow Command for more details.

2.3.7 Hydrology IUH Command

This command lets you generate a runoff hydrograph with a desired peak flow and with either a specified time to peak or duration. The command is shown below.



Peak discharge 1.000 c.m/sec

☐ Check to Specify Duration

Time to Peak 35.0 minutes

Duration 175.0 minutes

Click to select a file to represent the Instantaneous Unit Hydrograph

scs001.iuh
scsiuh.iuh

Figure 2-24 – The Default values in the IUH window

When using a pre-defined IUH curve it is probably more usual to define the desired peak discharge and the time to peak. The hydrograph duration is then dependent on the skew-ness of the IUH curve. If the computed duration is greater than the maximum hydrograph length defined with the Times parameters, the length of the IUH hydrograph is truncated and a warning message is displayed.

However, if the box labeled “Check to Specify Duration” is checked, the form is modified to let you specify the duration and the time to peak is then calculated as some fraction of the duration defined by the shape of the IUH function.

A file defining an IUH function must be selected for the [Display] command to be enabled. Pressing [Display] then computes a runoff hydrograph and displays it both graphically and in tabular form.

Different IUH files can be prepared by the user by copying or editing one of the sample files provided with MIDUSS.

2.3.8 Hydrology Runoff Area Totals

An in the Hydrology menu causes MIDUSS to keep a running total of the Catchment areas defined and also the total of impervious surface.

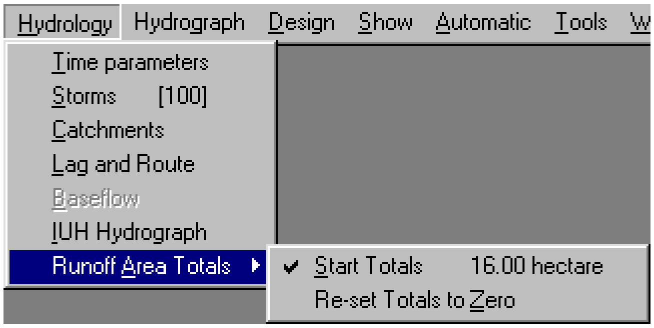
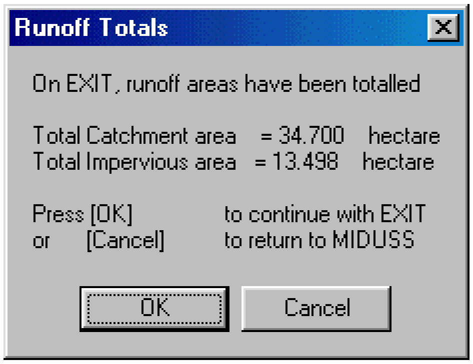


Figure 2-25 – The Runoff Area Totals menu.

The default condition when MIDUSS starts is to accumulate totals but this can be toggled off if desired. When totals are being calculated the Start Totals command is shown with a check mark. The current total area is also displayed against this menu item.

If separate totals are required for different branches in the same drainage network, the totals can be reset to zero by selecting (clicking) the **Reset Totals to Zero** command.

When you use the **Files / Exit** command a small summary message is displayed. More detail is shown in the output file



On EXIT, runoff areas have been totalled

Total Catchment area = 34.700 hectare
Total Impervious area = 13.498 hectare

Press [OK] to continue with EXIT
or [Cancel] to return to MIDUSS

Figure 2-26 – Runoff Area Totals are displayed on Exit.

2.3.9 Hydrology - Retrieving the Previous Storm

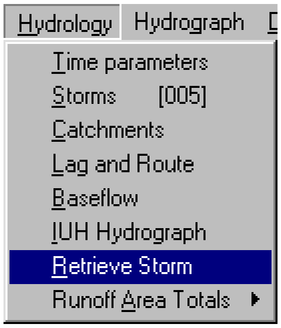


Figure 2-27 – The Hydrology menu after specifying a second storm

If you specify a second design storm during a MIDUSS design session the previous storm hyetograph will be overwritten with the new storm and this new hyetograph will be used for subsequent uses of the Catchment command. However the first storm is kept in memory and can be recovered if required.

Accepting the second storm causes the Hydrology menu to have a new item added at the bottom as illustrated in Figure 2- 27. Using the Retrieve Storm command causes the first storm to be restored but the second storm is not saved. This command may be useful in two cases.

- You mistakenly define a new storm
- You need to change the storm for only a few sub-catchments.

If you need to switch between two or more storms several times the simplest solution is to store the hyetograph as a file. See Chapter 6 *Working with Files* for details.

2.4 The Hydrograph Menu

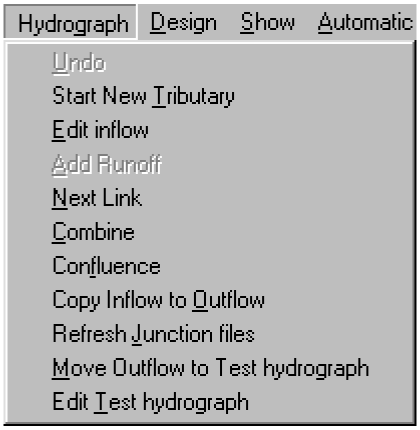


Figure 2-28 – Hydrograph operations available in MIDUSS

This menu item allows you to manipulate hydrographs in a number of different situations. Not all of the options may be enabled unless the appropriate pre-requisites are satisfied, e.g. the Combine command is available only after an Output hydrograph has been generated.

Undo	To restore the last modified hydrograph(s).
Start New Tributary	To edit or set the Inflow hydrograph to zero.
Edit inflow	Lets you create or modify an inflow hydrograph.
Add Runoff	Add the Runoff to the Inflow hydrograph.
Next Link	Copy the Outflow to the Inflow hydrograph.
Combine	Add the Outflow hydrograph to a junction.
Confluence	Move the Junction hydrograph to the Inflow.
Copy Inflow to Outflow	Copy Inflow hydrograph to Outflow.
Refresh Junction files	Refresh Junction files (e.g. delete unwanted files).
Move Outflow to Test hydrograph	An alternative way to load an observed hydrograph file.
Edit Test hydrograph	To create or modify a hydrograph of observed data/

2.4.1 Hydrograph Undo

This menu item is enabled only if a hydrograph has been written to the backup hydrograph. This is done in a number of instances to allow you to reverse an operation. For example, you may have used the Add Runoff command which causes the Runoff hydrograph to be added to the current inflow hydrograph. You may wish to 'Undo' this action and restore the previous Inflow hydrograph. In such a case, the Undo menu item will be enabled and pressing this command will restore the Inflow hydrograph to its previous state.

2.4.2 Hydrograph Start

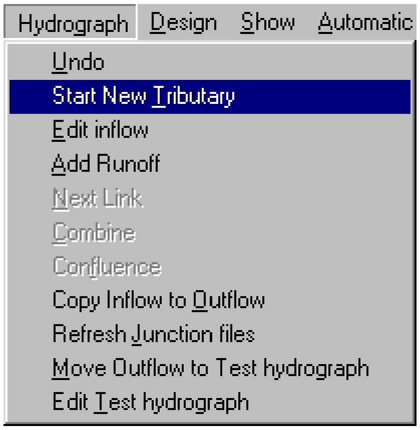


Figure 2-29 – The Hydrograph/Start New Tributary menu

This first option is generally used after you have stored an outflow hydrograph at a Junction node (by using the Combine command) and therefore wish to start a new tributary of the drainage network. If you fail to zero the Inflow hydrograph, the runoff from the first catchment of the new tributary will be added to the furthest downstream Inflow hydrograph of the previous branch.

2.4.3 Hydrograph Edit Inflow

This command may be used to input a hydrograph that represents observed data for comparison with a simulated hydrograph. Alternatively, you may wish to modify an existing inflow hydrograph.

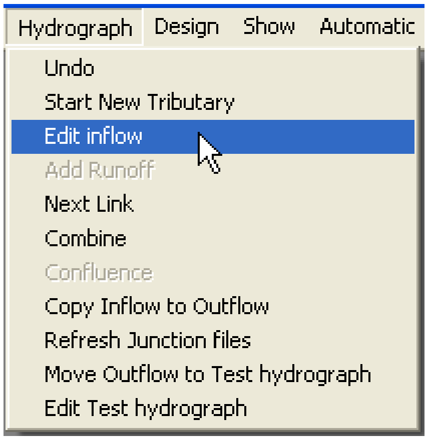
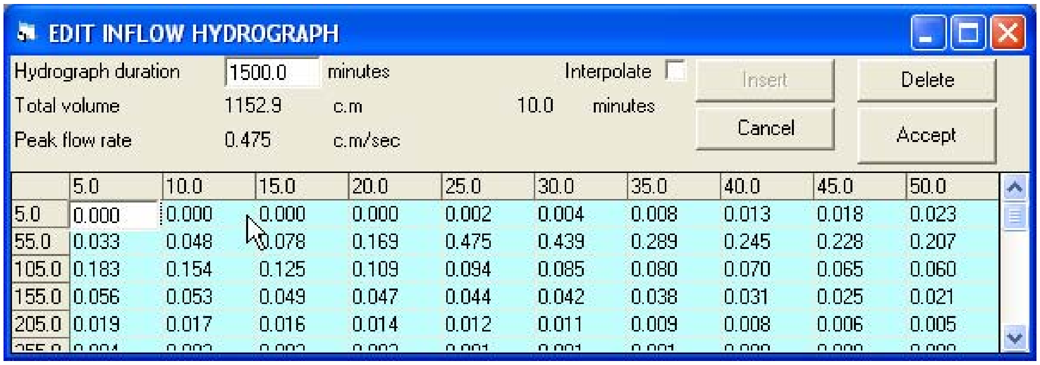


Figure 2-30 – The Hydrograph/Edit inflow menu

Figure 2-31 below shows the current inflow hydrograph loaded into the editing grid. Individual cell values can be modified using the keyboard. Cells can be Inserted or Deleted with automatic adjustment of the hydrograph length, maximum flow value and volume. A graphical plot is also displayed and is continuously updated as editing proceeds. Data can be transferred between the grid and the Clipboard using the **Edit / Copy** and **Edit / Paste** commands.



Hydrograph duration 1500.0 minutes

Total volume 1152.9 c.m

Peak flow rate 0.475 c.m/sec

Interpolate ☐ 10.0 minutes

	5.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	45.0	50.0
5.0	0.000	0.000	0.000	0.000	0.002	0.004	0.008	0.013	0.018	0.023
55.0	0.033	0.048	0.078	0.169	0.475	0.439	0.289	0.245	0.228	0.207
105.0	0.183	0.154	0.125	0.109	0.094	0.085	0.080	0.070	0.065	0.060
155.0	0.056	0.053	0.049	0.047	0.044	0.042	0.038	0.031	0.025	0.021
205.0	0.019	0.017	0.016	0.014	0.012	0.011	0.009	0.008	0.006	0.005

Figure 2-31 – Editing of the inflow hydrograph is done on a Grid form

2.4.4 Hydrograph Add Runoff

This command causes the Runoff hydrograph to be added to the current Inflow hydrograph so that a new pipe, channel or other drainage element can be designed to carry the increased flow. A message box is displayed to inform you what action has been taken and a new record is added to the database displayed in the Peak Flows table in the bottom right corner of the MIDUSS window showing the change in the peak of the Inflow hydrograph.

Runoff from last catchment area has been added to the Inflow hydrograph

Figure 2-32 – Hydrograph operations provide an explanatory message

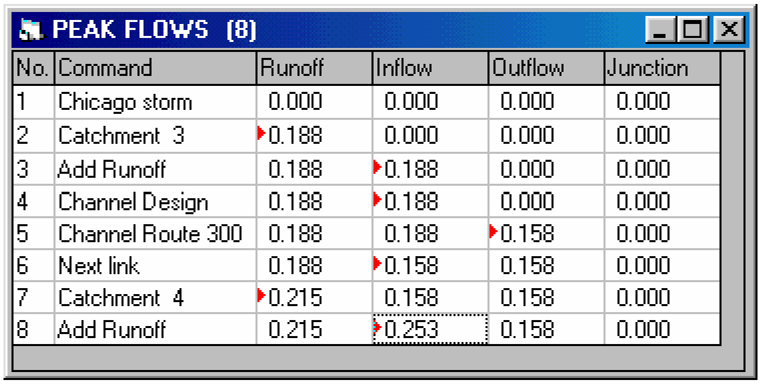
Note that in general the time to peak for the Runoff and Inflow hydrographs will not be the same. For this reason, the peak of the sum of two hydrographs will usually be less than the sum of the individual peaks.

The original value of the Inflow hydrograph is stored in the Backup hydrograph array and the Undo command is enabled. This allows you to reverse the action and restore the Inflow to its previous state. See the Hydrograph - Undo command for details.

2.4.5 Hydrograph Next Link

This command causes the Outflow hydrograph to be copied to the Inflow hydrograph overwriting the previous contents. This command is used when an Outflow hydrograph has been created by a routing process and you want to continue downstream to the next reach of conduit or channel, etc.

Once the Inflow has been updated in this way you may wish to define another catchment area and add the runoff from this area to the inflow before continuing with the design of the drainage network. Thus a typical sequence of commands might be represented by the following fragment of the peak flow summary table.

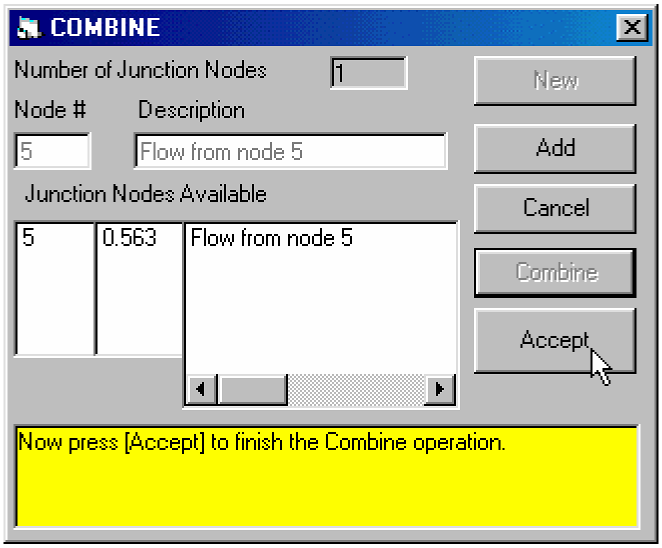


No.	Command	Runoff	Inflow	Outflow	Junction
1	Chicago storm	0.000	0.000	0.000	0.000
2	Catchment 3	▸0.188	0.000	0.000	0.000
3	Add Runoff	0.188	▸0.188	0.000	0.000
4	Channel Design	0.188	▸0.188	0.000	0.000
5	Channel Route 300	0.188	0.188	▸0.158	0.000
6	Next link	0.188	▸0.158	0.158	0.000
7	Catchment 4	▸0.215	0.158	0.158	0.000
8	Add Runoff	0.215	▸0.253	0.158	0.000

Figure 2-33 – The peak flow summary table after the Add Runoff command.

As with other hydrograph operations, the Inflow hydrograph is stored in the Backup hydrograph before it is overwritten thus allowing you to use the Undo command to reverse the operation and restore the Inflow to its original state.

2.4.6 Hydrograph Combine



Number of Junction Nodes: 1

Node #: 5

Description: Flow from node 5

Junction Nodes Available

5	0.563	Flow from node 5

Now press [Accept] to finish the Combine operation.

Figure 2-34 – The Hydrograph/Combine form offers step-by-step advice

The Combine command is used to store or accumulate one or more Outflow hydrograph at a junction node. The first time you store a hydrograph at a junction node, the peak flow is shown in the Junction or temporary column of the Peak Flows table, the hydrograph is stored in the current Junction or Temporary array and a file is created with the name Hydnnnnn.jnc where 'nnnnn' is the number of the junction node.

The next logical step is to clear out or set to zero the Inflow hydrograph in preparation for computing the flow in another branch of the drainage network. This branch will eventually terminate at the same junction node or at a different one. If it is added to the same node, the peak flow, the Junction hydrograph and the Junction hydrograph file are all updated.

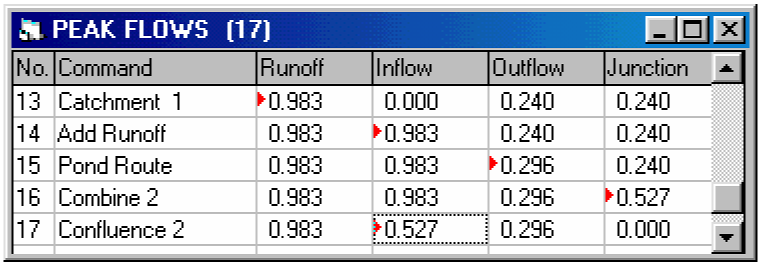
If a different Junction node is used the peak flow and the Junction hydrograph are overwritten and a new Junction hydrograph file is created. Since junction flows are always stored as a file it is possible to have any number of Junctions nodes active at one time

A more detailed description of the use of Junction nodes is provided in Chapter 5 – *Hydrograph Manipulation* topic The Combine Command.

2.4.7 Hydrograph Confluence

The Confluence command is used in conjunction with the Combine command and allows you to recover the accumulated flow at a junction node and copy it to the Inflow hydrograph, overwriting the previous contents. In this way, the command lets you continue the design of the drainage network downstream of a junction node.

The illustrated table has been copied from the summary of peak flows and shows a design sequence in which two tributaries are accumulated at a confluence node.



No.	Command	Runoff	Inflow	Outflow	Junction
13	Catchment 1	▸0.983	0.000	0.240	0.240
14	Add Runoff	0.983	▸0.983	0.240	0.240
15	Pond Route	0.983	0.983	▸0.296	0.240
16	Combine 2	0.983	0.983	0.296	▸0.527
17	Confluence 2	0.983	▸0.527	0.296	0.000

Figure 2-23 – Sequence of operations adding two branches at a Junction

In carrying out the operation the Inflow hydrograph is first copied to the Backup hydrograph to allow use of the **Hydrograph / Undo** command, the junction flow in the Peak Flows window is set to zero, the Junction array is set to zero and the Junction file is deleted. All of these actions can be reversed if you subsequently use the **Undo** command.

Before continuing with the design of the drainage network downstream of the junction node you may define a local catchment and add the runoff to the inflow if this is appropriate.

A more detailed description of the use of Junction nodes is provided in Chapter 5 – *Hydrograph Manipulation* in the topic The Confluence Command.

2.4.8 Hydrograph Copy To Outflow

This command causes the current inflow hydrograph to be copied to the Outflow hydrograph overwriting the current contents. This may be useful if the current inflow is to be added to a new or existing junction node without the need to use a Design option such as the Pond command or the Pipe & Route commands.

The action can be reversed by means of the **Hydrograph/Undo** command.

2.4.9 Hydrograph Refresh

This command is enabled if there are one or more junction files in the current Job directory and is intended to let the user selectively delete junction files which are no longer required.

2.4.10 Hydrograph Move Outflow to Test hydrograph

Moves a hydrograph that has been imported into the Outflow hydrograph array to the Test Hydrograph array

Several commands in MIDUSS have a checkbox which causes a user-defined Test hydrograph to be displayed graphically with the other hydrographs computed by the command. This allows some calibration of the MIDUSS model to improve the agreement between a hydrograph observed in the field and the hydrograph computed by MIDUSS.

A useful method to import an observed hydrograph is to open the Test Hydrograph window with the **Hydrograph / Edit Test Hydrograph** command and then run (for example) Microsoft Excel.

Enter the observed values in appropriate units in the Excel spreadsheet using 10 columns. Then use the **Edit / Copy** and **Edit / Paste** commands to copy the block of cells from the spreadsheet to MIDUSS.

2.4.11 Hydrograph Edit Test Hydrograph

Displays the current contents of the Test hydrograph in both tabular and graphical mode and allows the user to edit the individual cell-values. In addition, cells can be Inserted or Deleted with automatic adjustment of the hydrograph length, maximum flow value and total volume.

In several places in MIDUSS you will find a checkbox which causes a user-defined Test hydrograph to be displayed graphically with the other hydrographs computed by the command. This assists in calibration of the MIDUSS model to improve the agreement between a hydrograph observed in the field and the hydrograph computed by MIDUSS.

A useful method to import an observed hydrograph is to open the Test Hydrograph window with the **Hydrograph / Edit Test Hydrograph** command and then run (for example) Microsoft Excel.

Enter the observed values in appropriate units in the Excel spreadsheet using 10 columns. Then use the **Edit / Copy** and **Edit / Paste** commands to copy the block of cells from the spreadsheet to MIDUSS.

2.5 The Design Menu

Figure 2-36 – Design tools available in MIDUSS

The Design functions are generally all enabled once an Inflow hydrograph has been created. One exception is the Route command which requires that a Pipe or Channel conduit has been designed. Another difference is that the Pipe, Channel and Culvert commands are always enabled to allow you to enter a flow value and design a pipe or channel for that flow without having to define a storm and generate a flow hydrograph.

2.5.1 Design Pipe

The Pipe command lets you design a pipe to carry the peak flow of the current Inflow hydrograph. If no hydrograph has been calculated you can specify a desired flow directly by entering it in the text box. For the specified peak flow you will be shown a table of diameters, gradients and average velocities that represent feasible designs. You can either choose one of these diameter-gradient pairs by double clicking on a row in the table or you can enter explicit values for diameter and gradient.

MIDUSS carries out a uniform flow analysis and reports the actual depth and velocity and also the critical and specific energy depths. You can experiment by changing either the pipe roughness (i.e. the Manning 'n') or the diameter or gradient and press the [Design] button to see the results. When satisfied with the design press the [Accept] button to copy the data and results to the Output file.

2.5.2 Design Channel

MIDUSS lets you design channels with two types of cross-section to carry the current peak flow in the Inflow hydrograph. If no hydrograph has been calculated you can enter a flow value directly by entering it in the text box. The cross-section can be:

(1) A general trapezoidal shape defined by a base width and left and right side slopes.

(2) An arbitrary shape defined by up to 50 pairs of coordinates.

In both cases a table of depth, gradient, velocity values is displayed which represent feasible designs. You can select from this list by double clicking on a row of the table or you can specify a total depth and gradient explicitly. MIDUSS carries out a uniform flow analysis for the given flow, roughness and geometry and reports the depth of flow, the average velocity and the critical depth in the channel.

You can experiment by changing the data and pressing the [Design] button. When satisfied, press the [Accept] button to save the data and results to the output file.

2.5.3 Design Route

Once a drainage conduit has been designed - either a pipe or channel - you can route the Inflow hydrograph through a reach of specified length to obtain the Outflow hydrograph at the downstream end.

For each trial design MIDUSS checks that the time step and reach length are acceptable to ensure stability in the routing process. If the time step is too long it will be reduced to an appropriate sub-multiple. If the reach length is too long it also will be subdivided. In both cases the information is reported on the screen but no further action is required by the user.

The result of the routing operation is displayed in both graphical and tabular form. Once you are satisfied with the result you can press the [Accept] button and the peak flow summary table will be updated.

2.5.4 Design Pond

MIDUSS helps you to design a detention pond to achieve a desired reduction in the peak flow of a hydrograph. The current peak flow and the total volume of the inflow hydrograph are reported and you are prompted to specify the desired peak outflow. MIDUSS estimates the maximum storage requirement to achieve this.

The storage routing through the pond requires a table of values defining the outflow discharge and the storage volume corresponding to a range of stage or depth levels. You can enter this data directly into the grid if you wish, but it is usually easier to use some of the features of the Pond command to automate this process.

The outflow control can be designed using multiple orifices, weir controls or outflow pipes. The Stage - Storage values can be estimated for different types of storage facility. These may be a multi-stage pond with an idealized rectangular plan shape and different side slopes in each stage; one or more "super-pipes" or oversized storm sewers; wedge storage formed on graded parking lots; or a combination of these types of storage.

Rooftop storage can also be modelled to simulate controlled flow from the roof of a commercial development.

2.5.5 Design Trench

The Trench command lets you proportion an exfiltration trench to provide underground storage for flow peak attenuation and also to promote return of runoff to the groundwater. The trench usually consists of a trench of rectangular or trapezoidal cross-section filled with clear stone with a voids ratio of around 40% and with one or more perforated pipes to distribute the inflow along the length of the trench.

The exfiltration trench splits the inflow hydrograph into two components. One of these is the flow which infiltrates into the ground water; the balance of the inflow is transmitted as an outflow hydrograph. Obviously an exfiltration trench requires reasonable porosity of the soil and a water table below the trench invert.

The design involves several steps including definition of the trench and soil characteristics, definition of the number, size and type of pipes in the trench and description of the outflow control device comprising, orifice, weir and outflow pipe controls as used in the Pond command.

More detailed information is contained in the section 4-8 on **Exfiltration Trench Design.**

2.5.6 Design Diversion

A diversion structure allows the inflow hydrograph to be split into two separate components, the outflow hydrograph and the diverted flow hydrograph. Below a user-specified threshold flow all of the inflow will be transmitted to the outflow hydrograph. When the inflow exceeds the threshold value, the excess is divided in proportion to a specified fraction $F < 1.0$.

For example, if the inflow is 25 cfs and the threshold is 5 cfs the excess flow is 20 cfs. Now if the fraction $F = 0.8$ meaning that 80% of the excess flow is diverted the diverted flow will be 16 cfs and the outflow will be 9 cfs.

Instead of specifying the diverted fraction F you can define this implicitly by specifying the desired peak outflow. MIDUSS will then work out the necessary fraction to be diverted.

The diverted flow hydrograph is written to a file so that it may be recovered at a later time and used to design the necessary conduit or channel.

Use of the diversion command is the only instance in which the topology of the network changes from a tree to a circuited network.

Refer to Chapter 4 *Design Options Available* – Diversion Structure Design for more information.

2.5.7 Design Culvert

The Culvert command can be used with a constant (i.e. steady) discharge or with an inflow hydrograph.

A variety of barrel cross-sections can be selected including pipes, rectangular box sections, horizontal or vertical elliptical sections of pipe arch sections. Multiple barrels can be defined but all barrels will have the same attributes.

If the inflow is described by a hydrograph, completion of the design for the peak flow causes the [Route] command button to be enabled. The available storage upstream of the embankment through which the culvert passes is defined by the slope of the flood plains on either side of the channel and the slope and geometry of the channel leading to the culvert. A table is constructed with the stage, discharge and volume storage upstream of the culvert assuming that the upstream water surface is approximately horizontal. This allows the inflow hydrograph to be routed through the ponded reach with the result that the peak flow through the culvert is less than the peak of the inflow hydrograph. The design can then be updated for the reduced flow or left unchanged.

The results are displayed both graphically and in tabular form as in other flood routing commands.

If the culvert is surcharged either by a high flow rate or by a high tail-water depth it may happen that the flow is split into two components – through the culvert and over the weir formed by the embankment. In this case the weir flow is reported separately and the depth over the crest of the embankment is shown.

2.5.8 Design Cascade

On-site storage for commercial development frequently requires some underground storage facility to handle runoff from frequent storm event in order to reduce the frequency with which parking lot storage may be an inconvenience.

The Cascade command lets you route the current inflow hydrograph through a short cascade of storage cells formed from a variety of cross-sectional shapes such as pipes, rectangular boxes, horizontal and vertical elliptical pipes and pipe arch sections.

The current version of the command is limited to only two elements. The data entry section is self-explanatory to some extent. The form displays a number of columns equal to the number of elements. The bottom row shows a drop-down list from which the type of cross-section can be selected.

If any of the three special pipe sections are used (e.g. elliptical or pipe arch) a second drop-down list on line 3 of the form is activated to let you browse through a set of commercially available sizes. These are shown in metric or imperial sizes depending on the choice of units.

Apart from the overall dimensions of each element, you must define the invert level and an orifice diameter and coefficient of contraction of the outflow control from each cell. The orifice invert is assumed to be the same as the invert of the upstream element.

Each iteration of the analysis requires two steps – first click on [Setup Current Data] to enable the [Route] command and then click [Route] to carry out the analysis.

The second row of the grid shows the peak values of the Inflow, outflow from each cell and maximum depth in each element. If a cell is surcharged, the data box containing the Height is highlighted to warn you that more storage or a larger orifice is required.

2.6 The Show Menu

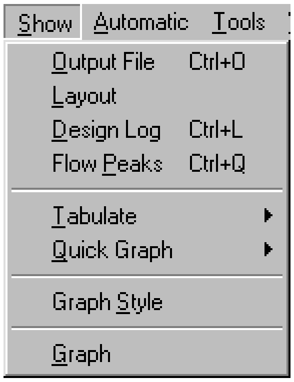


Figure 2-37 – Options available for displaying your results

The options in this menu allow you to display the results of your hydrologic modelling and design. Results for inclusion in a report can be displayed in tabular format or in graphical form. For each of these choices you can specify the specific data items to be displayed. The menu items are not enabled until there is data available to be displayed.

2.6.1 Show Output File

This command causes the Microsoft programs Notepad or WordPad to be opened with the current Outflow file. You can review the contents of the file, print all or part of the file or save it with a name other than that initially assigned to the file. (You can use the same filename but in a different directory).

However, you cannot make changes to the file since it must be re-opened when the Notepad window is closed.

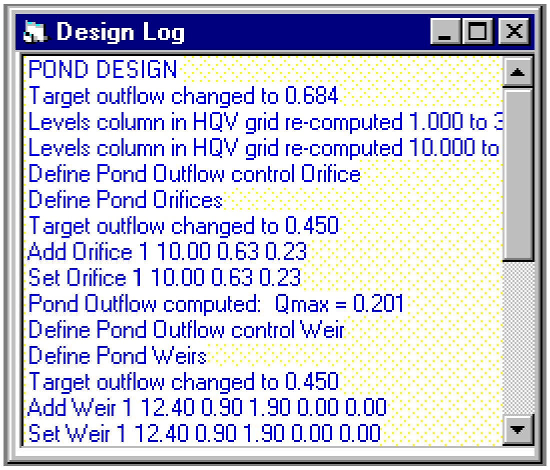
2.6.2 Show Layout

This command provides a visual display of the elements added to the drainage network. The procedure is carried out automatically in MIDUSS and Catchments, Pipes, Channels and other stormwater control devices are drawn on the Layout as they are designed. However, you can modify the layout by dragging one or more icons without violating the continuity of the drainage network.

2.6.3 Show Design Log

This command lets you review the contents of the current Design Log file. The Microsoft editor Notepad is opened with the current Design Log. As with the Output file, you can review it, print it or save it with a different path or name but you cannot change the contents.

When the Notepad window is closed the Design Log file is re-opened in Append mode so that additional records written to the Design Log will be added.



```
POND DESIGN
Target outflow changed to 0.684
Levels column in HQV grid re-computed 1.000 to 3
Levels column in HQV grid re-computed 10.000 to
Define Pond Outflow control Orifice
Define Pond Orifices
Target outflow changed to 0.450
Add Orifice 1 10.00 0.63 0.23
Set Orifice 1 10.00 0.63 0.23
Pond Outflow computed:  Qmax = 0.201
Define Pond Outflow control Weir
Define Pond Weirs
Target outflow changed to 0.450
Add Weir 1 12.40 0.90 1.90 0.00 0.00
Set Weir 1 12.40 0.90 1.90 0.00 0.00
```

Figure 2-38 – A typical Design Log during a Pond design

Each of the six options in the Design menu causes a Design Log window to be opened in the top right corner of the screen. This is used to record a summary of the steps taken during a design process. For simple elements such as a pipe design, the data requirements are limited to the roughness and to successive trials with different diameters and/or slopes. However, for a more complex drainage element such as a detention pond, there are many options which can be employed and the trial and error process can be complicated. In these cases the Design Log can be useful as a reference to earlier trials and also serves as a reminder should you be interrupted during the design process.

The current design log can be printed out at any time for convenience of reference.

At the end of the design of each element the Design Log is copied to a file which can be saved or printed out at the end of the MIDUSS session. The total Design Log can also be viewed by using the Show/Design log menu item. Refer to the Show menu for more details.

2.6.4 Show Flow Peaks

If for some reason you have closed the Window showing the summary of flow peaks, it will be restored automatically with the next command that modifies the hydrographs. The records for each row of the peak flow summary table are contained in a text file called Qpeaks.txt which resides in the MIDUSS current job folder. Normally you will have no need to refer to this but should you wish to do so it can be viewed and printed or saved by another name or path by using the Microsoft editor Notepad which can be run from the Tools menu or from the desktop after the MIDUSS session has been completed.

2.6.5 Show Tabulate

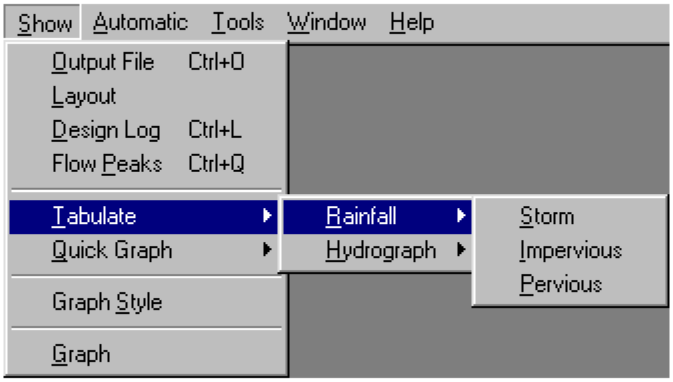


Figure 2-39 – Some of the options available with the Show/Tabulate command

A tabular display of each rainfall hyetograph or flow hydrograph is displayed - usually in the lower left corner of the MIDUSS window - as the MIDUSS session proceeds. This command lets you display the table for any of the three hyetographs or any hydrograph arrays which are currently in use.

2.6.6 Show Quick Graph

With each step of the MIDUSS session a graphical display is opened in the top right corner of the MIDUSS window. This command lets you open a similar graphical window to display any of the current rainfall hyetographs or flow hydrographs.

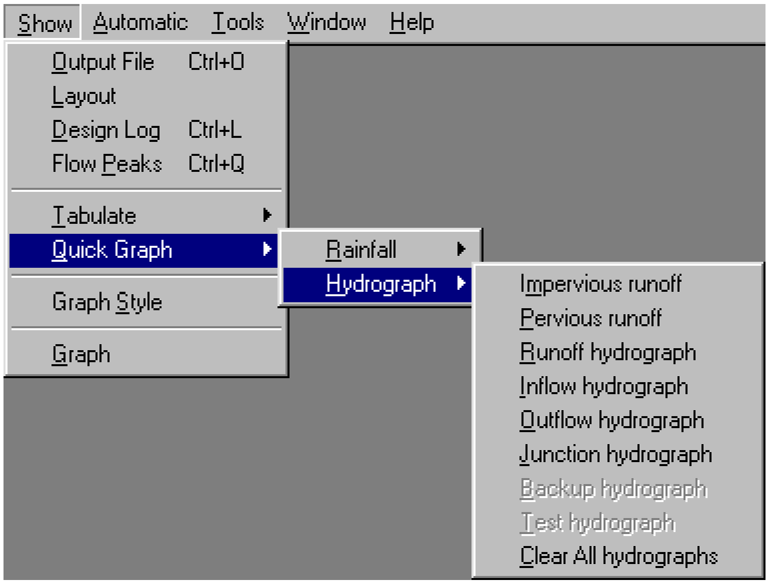


Figure 2-40 – Some of the Options available with the Show/Quick Graph command

2.6.7 Show Graph

This command lets you construct a graphical display comprising several hydrographs and hyetographs. If a rainfall hyetograph is plotted first it may be drawn either on the bottom axis or along the top edge of the plotting window.

However, if one or more hydrographs have been plotted, the addition of a hyetograph will be automatically placed on the top edge with the scale reading downwards.

The scale for both time and ordinate value can be set when the first object is plotted but subsequent objects are plotted to the same scale.

You can annotate the graphic by adding text, arrows, lines, rectangles or circles.

The graphic can be saved as a file or a previously saved graphic file can be imported to the empty graphic window. To conveniently store the graphic for addition of other objects later in the MIDUSS session the form can be iconized from a menu item and restored by re-invoking the **Show/Graph** command.

The colour, line thickness and fill pattern can be selected by the user. The preferred styles are automatically saved as default values when the MIDUSS session is ended.

2.7 The Automatic Menu

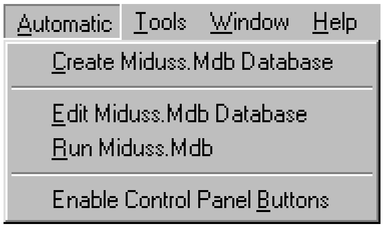


Figure 2-41 – Commands available for running in Automatic mode

The four options in this menu allow you to run MIDUSS in Automatic mode. When MIDUSS runs in normal, Manual mode the commands, data and some of the results are copied to an output file. To use this file for input in Automatic mode it must be converted into a Database file. The data base can be reviewed and edited prior to running in Automatic mode.

2.7.1 Automatic - Create Input Database

When running in Automatic mode MIDUSS uses a specially created Database file called Miduss.Mdb which resides in the current job directory. This file can be created from an existing Output file by means of this menu command.

When you use this command a standard File Open dialogue box is opened. You should navigate to the appropriate directory and select the Output file that you wish to use in creating the Input Database file. During processing a small window opens to show the number of records processed and the number of commands read.

You can immediately make use of the Edit Input Database or Run Input Database commands.

See Chapter 10 – *Running MIDUSS in Automatic Mode* Creating the Input Database MIDUSS.Mdb for more details.

2.7.2 Automatic- Edit Input Database

This menu command lets you review and edit the contents of the current Input Database file 'MIDUSS.Mdb' in the job folder. Editing is limited to changing data values in a field of a particular record. It is not possible to insert new commands in the Database.

If you want to make major changes such as adding or deleting an entire command you can do this by interrupting a run in Automatic mode and entering the necessary manual commands. You may do this also by editing the Output file in a text editor such as Notepad. However, you should attempt such editing only after you are fully conversant with the sequence and format of the required data.

2.7.3 Automatic - Run Input Database

This command causes a control panel to be displayed which contains a grid and several command buttons. The grid shows the current contents of the database file Miduss.Mdb

The command buttons let you run the input database automatically in three different modes.

(1) EDIT mode lets you see the result of each command and you may make any changes to the data prior to pressing the [Accept] key on the command form. Any changes will be reflected in the new Output file being created.

(2) STEP mode advances the input file one command at a time but you do not have the opportunity to make changes to the data.

(3) RUN mode causes the commands to be processed sequentially without a pause until any one of the STEP, EDIT or MANUAL buttons is pressed or until the end of file is reached. RUN mode may also be interrupted if surcharge or overflow conditions are encountered.

(4) SKIP moves the current record forward to just before the next command.

(5) BACK move the current record back to just before the previous command.

(6) MANUAL terminates the automatic processing to allow you to carry out design steps in manual mode. However, a ‘bookmark’ is inserted into the database to let you resume automatic processing from that point.

(7) CLOSE closes the input database and the control panel.

See Chapter 10 – *Running MIDUSS in Automatic Mode* Using the Automatic Control Panel for further details.

2.7.4 Automatic - Enable Control Panel Buttons

Sometimes when running in Automatic mode you may find that all of the command buttons are disabled or ‘grayed out’. This command allows you to re-enable the command buttons to let you continue in either automatic or manual mode.

2.8 The Tools Menu

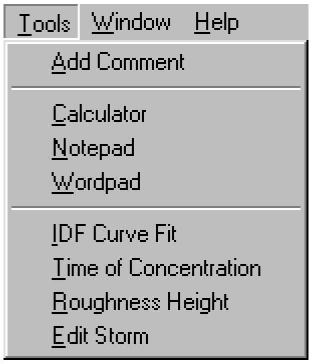


Figure 2-42 – Choices available in the Tools menu

The Tools menu provides access to the Microsoft Calculator, the Notepad text editor or the WordPad editor from within MIDUSS.

In Addition, a menu item is provided to let you add comments to the currently defined output file.

Other options are available as shown in Figure 2-42.

2.8.1 Tools - Add Comment

When you run MIDUSS there is always an Output file, whether one is specified explicitly by the user in a Job Directory or the default output file 'Default.Out'. This menu command lets you add explanatory comments to the output file.

You simply type in the Text box using normal text editing features such as Backspace, Delete or mouse controls. The font used is a uniformly spaced Courier font to allow columns to be aligned if desired. The text box entry provides automatic word wrap but this may not correspond exactly to the location of new lines in the output file.

The double quote character (“) cannot be included in a comment and is automatically trapped and converted to a single quote.

The comment window can be re-sized by dragging the right side or the bottom of the window. The text entry box will also be re-sized. You can enter as many lines of comment as you wish. A vertical scroll bar appears when there are more lines than the text box can display.

The maximum line length which is written to the output file is 60 characters and you cannot use a ‘word’ or string of characters more than 59 characters in length.

Another use for the **Add Comment** command is to add a visual description to the MIDUSS window before printing hardcopy of the screen. If you don’t want this added to the output file press [Cancel] to close the window before accepting the current design object.

2.8.2 Tools - the Microsoft Calculator

Clicking on this menu item causes the Microsoft Calculator Accessory to be opened. This may be useful if some hand calculation is required during a MIDUSS session.

When MIDUSS starts up it attempts to locate the directory where Calculator resides and stores this for later use. However, if the file Calculator.exe cannot be found for any reason you will be prompted to locate this manually in a standard File Open dialogue box. Once located, the directory will be stored in the registry for future use.

When you finish using Calculator you should close it in the normal way rather than merely clicking on the MIDUSS window which will place the Calculator window behind the MIDUSS window and probably out of sight. This may lead to multiple instances of Calculator being opened simultaneously.

2.8.3 Tools - the Microsoft Notepad editor

This command opens the Microsoft text editor Notepad. You can load, view and edit text files with this facility. Notice however, that files which are currently in use by MIDUSS - such as the current output file - cannot be changed. You can, however, print out the file in whole or in part or save it with a different filename.

When you save a file from Notepad, check to see if Notepad has added the default extension ".txt" to the filename which you have specified.

When MIDUSS starts up it attempts to locate the directory where Notepad resides and stores this for later use. However, if the file Notepad.exe cannot be found for any reason you will be prompted to locate this manually in a standard File Open dialogue box. Once located, the directory will be stored in the registry for future use.

When you finish using Notepad you should close it in the normal way rather than merely clicking on the MIDUSS window which will place the Notepad window behind the MIDUSS window and probably out of sight. This may lead to multiple instances of Notepad being opened simultaneously.

2.8.4 Tools - the Microsoft WordPad editor

This command causes the Microsoft text editor Wordpad.exe to be opened. You can load, view and edit text files with this facility with more flexibility than is possible with Notepad. Notice however, that files which are currently in use by MIDUSS - such as the current output file - cannot be changed. You can, however, print out the file in whole or in part or save it with a different filename.

When MIDUSS starts up it attempts to locate the directory where WordPad resides and stores this for later use. However, if the file Wordpad.exe cannot be found for any reason you will be prompted to locate this manually in a standard File Open dialogue box. Once located, the directory will be stored in the registry for future use.

When you finish using WordPad you should close it in the normal way rather than merely clicking on the MIDUSS window which will place the WordPad window behind the MIDUSS window and probably out of sight. This may lead to multiple instances of WordPad being opened simultaneously.

2.8.5 Tools - IDF Curve Fit

The IDF Curve Fit tool manipulates data describing an Intensity-Duration-Frequency for a particular geographical locality and can be used in two modes:

1. To compute the ‘a’, ‘b’ and ‘c’ parameters of a Chicago hyetograph that most closely approximates a set of observed data.

2. To compute the IDF curve for user-supplied values of the three coefficients and compare this with observed data.

2.8.6 Tools - Time of Concentration

The Time of Concentration Tool helps you calculate Tc using the sum of up to three components of travel time. These are:

1. Flood wave travel time of overland flow

2. Travel time in relatively small collector channels or gutters

3. Travel time in a storm conduit such as a circular pipe or a channel of general trapezoidal cross-section.

For the overland flow you can select one of two equations: Friend’s equation or the Kinematic equation.

Either metric or U.S.Customary (Imperial) units can be used depending on the units selected in running MIDUSS

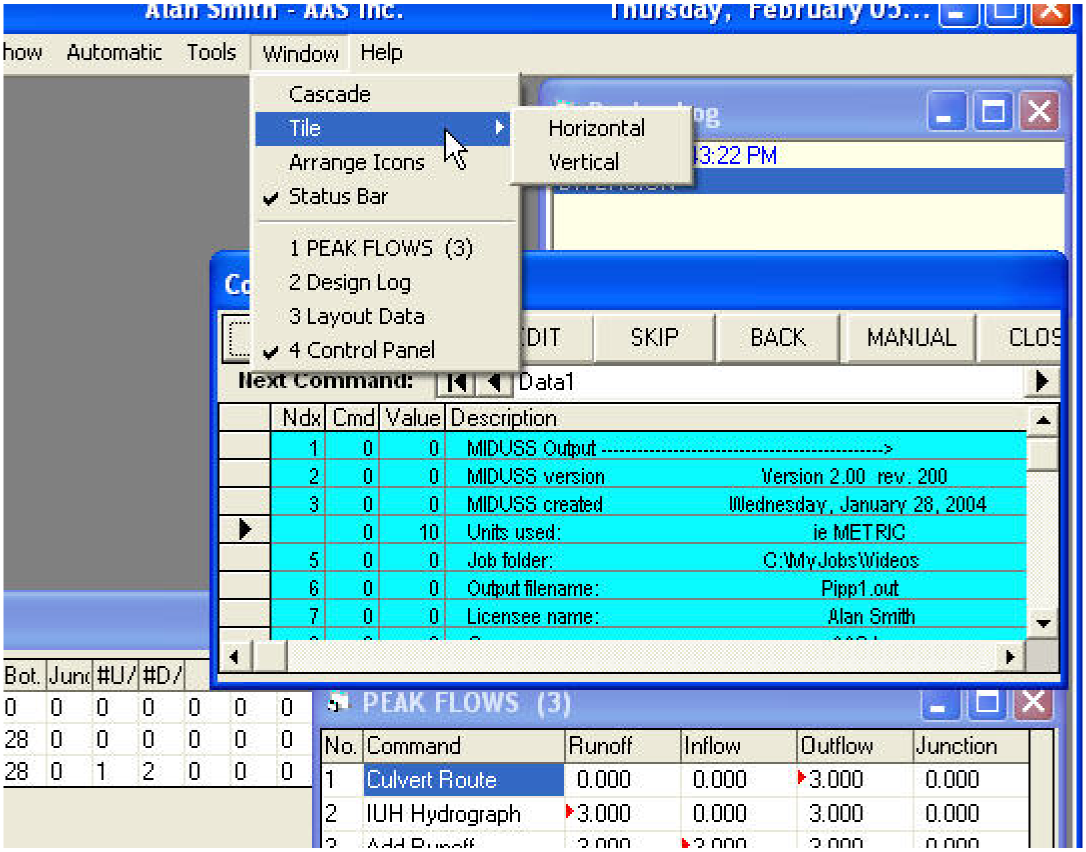
2.8.7 Tools - Roughness Height

MIDUSS design routines use the Manning ‘n’ to describe surface roughness. Users who prefer to define roughness in terms of the equivalent roughness height can use the Roughness Height tool to convert from roughness height (in either mm or inches) to Manning ‘n’.

2.8.8 Tools - Edit Storm

One of the options in the Storm command is to use a pre-defined curve known as a Mass Rainfall Distribution curve. These files are given the extension *.MRD and define the fraction of rainfall depth R(t)/Rtot as a function of the ratio of elapsed time over total storm duration. Typical examples are for the various Huff storm quartiles and the SCS hyetographs. The Edit Storm tool lets you modify an existing MRD file or construct a new one.

2.9 The Windows Menu



Ndx	Cmd	Value	Description	
1	0	0	MIDUSS Output --->	
2	0	0	MIDUSS version	Version 2.00 rev. 200
3	0	0	MIDUSS created	Wednesday, January 28, 2004
	0	10	Units used:	ie METRIC
5	0	0	Job folder:	C:\MyJobs\Videos
6	0	0	Output filename:	Pipp1.out
7	0	0	Licensee name:	Alan Smith

Bot.	Junc	#U/	#D/			
0	0	0	0	0	0	0
28	0	0	0	0	0	0
28	0	1	2	0	0	0

No.	Command	Runoff	Inflow	Outflow	Junction
1	Culvert Route	0.000	0.000	3.000	0.000
2	IUH Hydrograph	3.000	0.000	3.000	0.000

Figure 2-43 – You can arrange windows in different ways

The Window menu lets you modify the way the windows are displayed and also lists the windows which are currently open.

2.9.1 Windows - Cascade

This causes all of the currently open windows to be displayed in the standard Windows cascade style.

2.9.2 Windows - Tile

This causes all of the currently open windows to be displayed in the standard Windows tile arrangement.

2.9.3 Windows - Arrange Icons

Positions icons on the desktop.

2.9.4 Windows - Status Bar

This command has the same effect as the **Show Status Bar** command in the **Options** menu.

2.10 The Help Menu

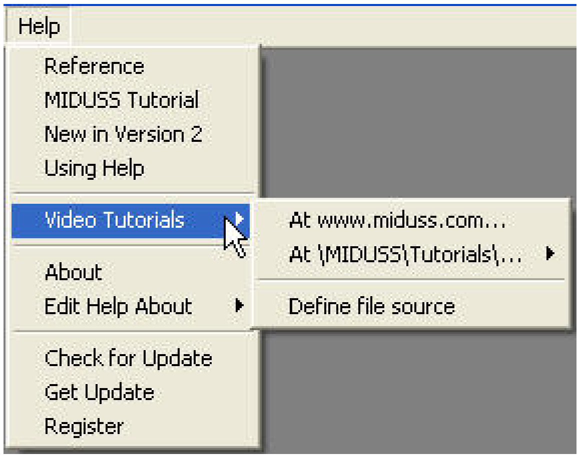


Figure 2-44 – The standard Help menu is available

This menu has the normal choices available in most Windows applications.

Reference	This help file provides detailed theory and practice on the features available in MIDUSS. This help file has the same content as the printed Reference manual.
MIDUSS Tutorial	This help topic goes through the design of a small drainage network. Many of the MIDUSS features are used. This tutorial has the same content as the printed Tutorial Manual as well as some of the Video Tutorials.
New in Version 2	This help file provide details of the additional features released with Version 2.
Using Help	This help file explains how to use the standard Windows-based help system.
Video Tutorials	Provides audio visual videos detailing how to use MIDUSS. Many of the lessons follow the example presented in the MIDUSS Tutorial manual and help file. The MIDUSS CD contains a folder called Tutorials which holds a number of audio-visual lessons on the main features and operations of MIDUSS. These can be accessed directly from the CD or from the **Help / Video Tutorials** menu command. Your computer must have a sound card in order to hear the audio track. Many of the lessons are taken from the steps in the MIDUSS Tutorial.
About	Provides information on your MIDUSS license including Serial numbers and maintenance expiry date.

Edit Help About	Lets you change some of the About details to reflect your organization.
Check for Update	Starts a small program that visits the www.miduss.com web site to determine if there is a more recent MIDUSS update available.
Get Update	Opens your internet browser at a location on the MIDUSS web site where you can logon and download the latest MIDUSS files.
Register	Access the registration page on the web site.

Chapter 3 Hydrology used in MIDUSS

This part of the MIDUSS Help System describes the hydrology commands used to model the rainfall runoff process and generate the hydrographs for which your stormwater management facilities will be designed.

The hydrology incorporated in MIDUSS is based on relatively simple and generally accepted techniques. Apart from setting the time parameters there are five commands to control the fundamental operations.

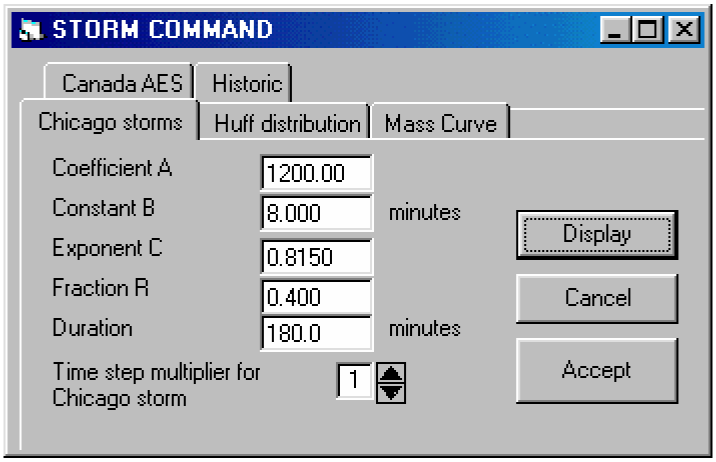
STORM	This command allows you to define a rainfall hyetograph either of the synthetic, design type or a historic storm. You should remember that as an alternative to using the STORM command, a previously defined rainfall hyetograph may be read in from a file, by means of the **File / Load file / Rainfall** command.
CATCHMENT	The Catchment command lets you define a single sub-catchment and computes the total overland flow hydrograph for the currently defined storm. The runoff hydrographs from the pervious and impervious areas are computed separately and added to give the total runoff. The roughness, degree of imperviousness, overland flow length and surface slope of both the pervious and impervious fraction are defined in this command. In addition you can choose from different rainfall loss models. The effective rainfall on these two fractions is computed and stored for future use. Different methods for routing the overland flow are available.
LAG and ROUTE	This command is useful for modelling the runoff from very large sub-catchments without having to resort to specifying unrealistically long overland flow lengths. The command computes the lag time in minutes of a hypothetical linear channel and linear reservoir through which the runoff hydrograph is routed. Typically this results in a smaller, delayed runoff peak flow.
BASE FLOW	This command lets you specify a constant positive (or negative) value of base flow to be added to (or subtracted from) the current inflow hydrograph. The command is enabled only after an inflow hydrograph has been defined.
IUH HYDROGRAPH	This lets you define a hydrograph based on a peak flow value and time to peak applied to an Instantaneous Unit Hydrograph.

This chapter describes how to use these commands and discusses the hydrological techniques in general terms. More information on the background theory can be found in Chapter 7 - *Theory of Hydrology*. If you need further information refer to any standard text on the subject. A number of references for this purpose are provided in Appendix ‘A’. You may also find it useful to subscribe to one or more of the relevant discussion groups which are available on the Internet.

MIDUSS offers a choice between four alternative methods for routing the overland flow and three different models for estimating infiltration and rainfall losses. In general these different methods will result in significantly different results. MIDUSS may therefore be used to compare methods and to examine the sensitivity of the resulting runoff hydrograph to the methods used. This flexibility means, however, that the engineer must exercise some care and consistency in the selection of procedures and parameter values for a particular application.

An important distinction must be made between the three overland flow routing methods which are based on the notion of effective rainfall and the SWMM Runoff method which uses a surface water budget approach which computes both runoff and rainfall loss as a function of rainfall. The difference is particularly noticeable when the runoff from the pervious fraction is significant This is discussed in more detail in Chapter 7 – *Hydrological Theory*, Rainfall-Runoff Models..

3.1 Storm Command



Coefficient A 1200.00
Constant B 8.000 minutes
Exponent C 0.8150
Fraction R 0.400
Duration 180.0 minutes
Time step multiplier for Chicago storm 1

Figure 3-1 – Five design storms are available

The rainfall hyetograph can be defined in five ways using this command. You may choose to define either:

(1) a synthetic design storm from four different types available,

or (2) a historic rainfall record based on observed data.

The form for the Storm command offers five alternate Tabs and selection is made by clicking on one of these. Select from the first four for a design storm or choose the fifth option for a historic storm.

Chicago Storm Hyetograph

Huff Storm

Mass Rainfall Distribution

Canadian AES 1-hour Storm

Historic Storm

After you have defined or accepted the parameters for the selected storm type you can press the [Display] command button to compute the storm hyetograph. This is displayed in both a graphical and tabular form. Since these display features are common to all types of design storm they are described first.

You can experiment by changing one or more of the parameter values and press [Display] again. When you are satisfied with the design storm you should press the [Accept] command button to store the hyetograph. The design storm will remain unchanged until you change it by a second use of the Storm command or by importing a hyetograph file.

The description of each of the design storms referenced above describes how these options can be used. A more detailed description of some of the design storm options is contained in Chapter 7 *Hydrological Theory*.

3.1.1 Graphical Display of the Storm

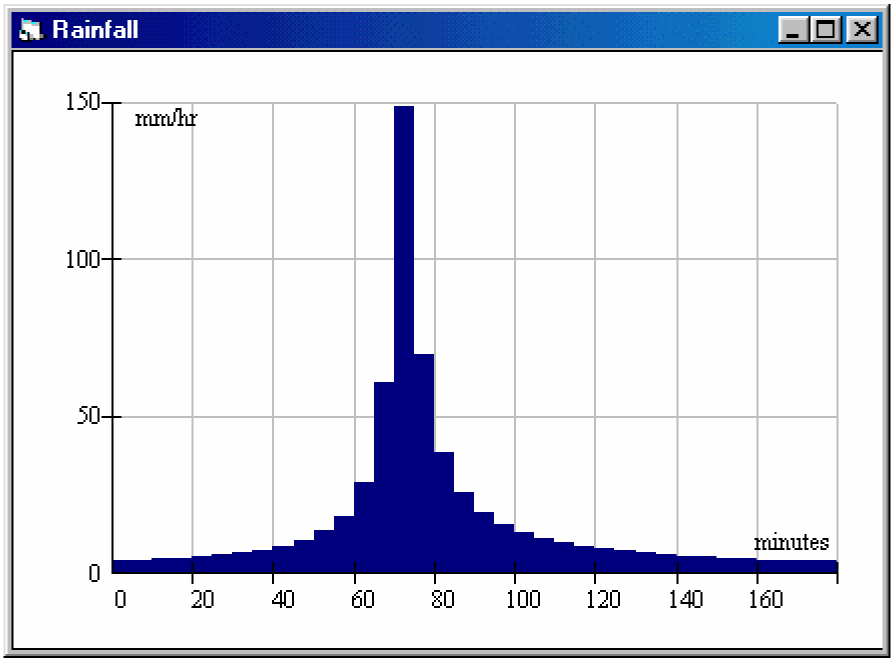


Figure 3-2 – A typical plot of the design storm.

A graph of the storm hyetograph is presented when you press the [Display] command button. The graph illustrated in Figure 3-2 is for a Chicago hyetograph. You can test the sensitivity of the storm to changes in any of the parameters by altering the data and pressing [Display] again. The storm will not be stored until the [Accept] command button is pressed.

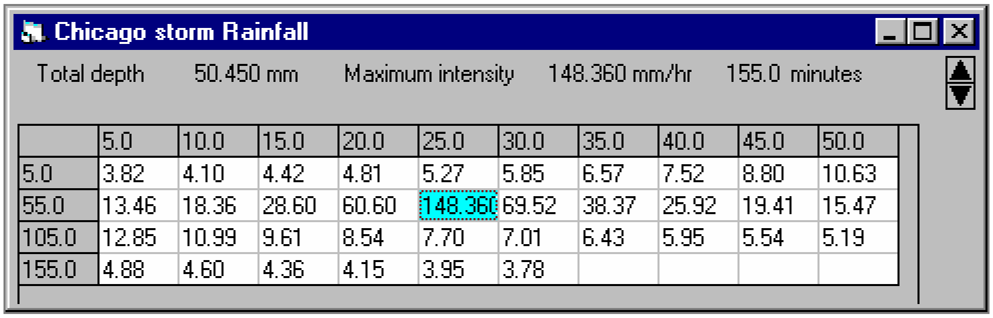
A similar graphical display is also shown for hydrographs following any of the commands which result in some change in the hydrograph data.

When you move the mouse pointer over the graph a small window displays the time and either a rainfall intensity or a hydrograph flow value corresponding to the position of the vertical cross-hair. If you hold down the secondary (right) mouse button a pair of cross-hairs will follow the mouse pointer. The position of the horizontal cross-hair has no significance but the vertical cross-hair should intersect the time axis at the time value shown in the small data window.

You can position the graph window by clicking on the title bar and dragging the window to a new location. You can also resize the graph window by dragging any corner of the graph window. The graph is automatically scaled to fit the window. The graph window can also be shown in full screen mode or reduced to an icon by clicking on the window size controls at the top-right corner of the window.

If you click with the primary mouse button anywhere in the plot area, the window is restored to its default size and position in the upper right corner of the MIDUSS window.

3.1.2 Tabular Display of Storm Hyetograph



Total depth 50.450 mm Maximum intensity 148.360 mm/hr 155.0 minutes

	5.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	45.0	50.0
5.0	3.82	4.10	4.42	4.81	5.27	5.85	6.57	7.52	8.80	10.63
55.0	13.46	18.36	28.60	60.60	148.360	69.52	38.37	25.92	19.41	15.47
105.0	12.85	10.99	9.61	8.54	7.70	7.01	6.43	5.95	5.54	5.19
155.0	4.88	4.60	4.36	4.15	3.95	3.78				

Figure 3-3 – A tabular display of Mass Curve storm hyetograph.

When the [Display] command button is pressed a table of rainfall intensities is displayed in the lower left corner of the screen. When you move the mouse pointer over the table, the time in minutes corresponding to the cell beneath the pointer is displayed in the top-right corner of the grid. The Spin Button on the top right corner lets you vary the number of figures after the decimal point between a minimum of 0 and a maximum of 5.

A similar tabular display is also shown for hydrographs following any of the commands which result in the generation of a new set of hydrograph data.

Basic statistics are shown in the header of the table defining:

- For hyetographs, the total depth of rainfall and the peak intensity.
- For hydrographs, the total volume and the peak flow rate.

By altering the data for the command and pressing the [Display] button again the table is updated. In the Storm command, this is useful for checking the sensitivity of the design storm to changes in the parameters or even in the type of design storm. For other commands a similar capability exists for sensitivity analysis.

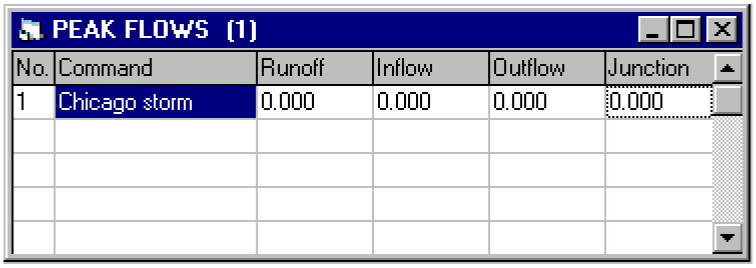
If the number of rows of data is greater than the number of rows in the table you can do one of two things.

- use the scroll bar on the right side of the table
- click on and drag the top edge of the form to increase the height of the form.

The second method is useful if you need to print out the entire contents of the table.

3.1.3 Accepting the Storm

The [Accept] command button is initially disabled when you first use the Storm command. When you press the [Display] button to show the graph and tabular display, the [Accept] button is enabled. However, if you change any of the parameter values, the [Accept] button is again disabled until the [Display] button is used to refresh the storm hyetograph.

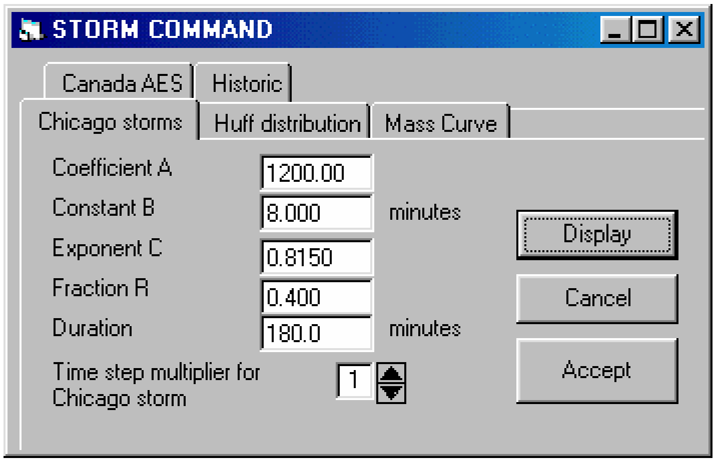


No.	Command	Runoff	Inflow	Outflow	Junction
1	Chicago storm	0.000	0.000	0.000	0.000

Figure 3-4 – The peak flow summary table with a single record.

Once you are satisfied with the storm you can cause it to be stored by pressing the [Accept] button. The graph and tabular display windows are closed and the small summary window shown in Figure 3-4 is displayed in the lower right corner of the screen. This table will be updated with hydrograph information as you define sub-catchments and design components. For this initial display only the storm has been defined and no runoff hydrographs have been generated.

3.1.4 Chicago Hyetograph



Coefficient A: 1200.00
Constant B: 8.000 minutes
Exponent C: 0.8150
Fraction R: 0.400
Duration: 180.0 minutes
Time step multiplier for Chicago storm: 1

Figure 3-5 – The Chicago storm tab of the Storm command

When you select the Chicago hyetograph tab the data entry form shown in Figure 3-5 is displayed. The synthetic hyetograph computed by the Chicago method is based on the parameters of an assumed Intensity - Duration - Frequency (IDF) relationship, i.e.

$$i = \frac{a}{(t_d + b)^c} \qquad [3.1]$$

where i = average rainfall intensity (mm/hr or inch/hr)

t_d = storm duration (minutes)

a,b,c = constants dependent on the units employed and the return frequency of the storm.

The asymmetry of the hyetograph is described by a parameter r (where $0 < r < 1$) which defines that point within the storm duration t_d at which the rainfall intensity is a maximum.

When data has been entered press the [Display] command button to see a graphical plot of the storm hyetograph and a table of rainfall intensities. You can change any value and press [Display] again to see the effect of the change. When you are satisfied with the storm, press [Accept] to define the storm.

Although widely used in North America, the Chicago storm has been criticized because it combines the characteristics of many different events and does not necessarily provide a reasonable simulation of an actual storm. In particular, the peak intensity is very sensitive to the time step employed. For that reason some engineers prefer to use a multiple of the time step in the vicinity of the peak. The time step multiplier can be used for this purpose. The volume of rainfall is unaffected but the peak intensity is significantly reduced. The best way to understand this is to experiment by increasing the multiplier to 2 or 3 and press [Display] to see the effect.

If you have access to data defining either depth or average intensity for different times, you can use the IDF Curve Fit Tool to estimate the values of a, b and c which give the closest fit to the observed IDF data. See Chapter 11 *MIDUSS Tools* Sec 11.3.

3.1.4.1 The 'a' coefficient

The value of the 'a' coefficient depends on (i) the return interval in years of the storm and (ii) the system of units being used. e.g.

Years	Metric	Imperial
2	250	10
5	400	17
10	700	25
25	950	36
50	1250	50
100	1800	75

Use these default values only as a last resort.

3.1.4.2 The b-constant

This constant in minutes is used to make the log-log correlation as linear as possible. Typical values range from 2 to 12 minutes. A value of zero for this parameter represents a special case of the IDF equation where

$$[3.2] \qquad i = \frac{a}{t_d^c}.$$

In general, this results in poor agreement between observed values of intensity and duration and those represented by the IDF equation.

3.1.4.3 The c-exponent

This parameter is usually less than 1.0 and is obtained in the process of fitting the data to the power expression. Values are usually in the range 0.75 to 1.0

3.1.4.4 The r-peak fraction

This parameter is the fraction of the storm duration to the point of maximum rainfall intensity (e.g. in Figure 3-2 the value of r is 0.4). Values are usually in the range 0.25 to 0.6 but any value less than 1.0 may be used. Notice that on pervious surfaces a high (i.e. late) value of r will result in a higher runoff peak since the ground tends to be more saturated when the peak intensity occurs.

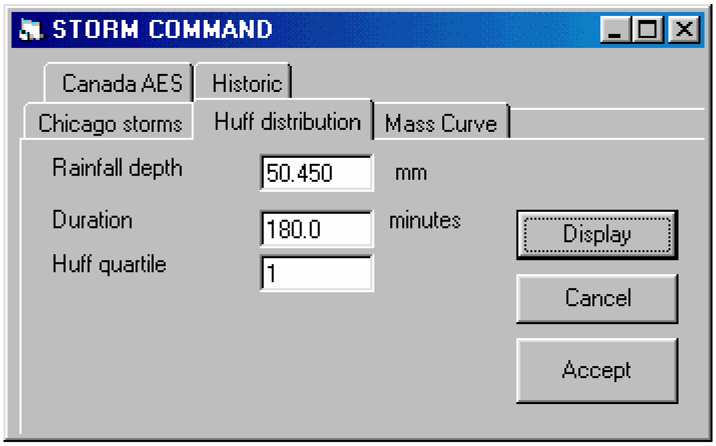
3.1.4.5 Duration

This defines the duration of the storm in minutes. It must be not greater than the maximum storm duration defined in the time parameters. Usually some multiple of the time step is used.

3.1.4.6 Time step multiplier

If a very small time step is used with the Chicago hyetograph it can result in a very high peak rainfall intensity. You can use an integer multiplier to use a small time step but avoid the unrealistically high peak. Experiment to see the difference.

3.1.5 Huff Rainfall Distribution



Rainfall depth 50.450 mm
Duration 180.0 minutes
Huff quartile 1

Figure 3-6 – The Huff distribution tab of the Storm command

Selection of the Huff Distribution tab on the Storm command form causes the form shown in Figure 3-6 to be displayed. The non-dimensional rainfall distribution patterns suggested by Huff were divided into four groups in which the peak intensity occurs in the first, second, third or fourth quarter of the storm duration. Within each group the distribution was plotted for different probabilities of occurrence. MIDUSS uses the median curve for each of the four quartile distributions. (See Chapter 7 *Hydrological Theory*; Derivation of the Huff Storm).

To define a storm of this type you must provide values for the total depth and duration of the storm and select the quartile distribution required. Refer to the topics which follow for a brief explanation of each parameter. When data has been entered press the [Display] command button to see a graphical plot of the storm hyetograph and a table of rainfall intensities. You can change any parameter value and press [Display] again to see the effect of the change. When you are satisfied with the storm, press [Accept] to define the storm.

Canadian users who want to employ the Atmospheric Environment Service 1-hour or 12-hour distribution suggested by Hogg can do so with the Mass Rainfall Distribution option.

3.1.5.1 Huff Quartile

The Huff quartile takes a value of 1, 2, 3 or 4 and defines the quartile of the storm duration in which the maximum intensity occurs. Refer to Chapter 7 *Hydrological Theory* for background information on the Huff distribution or the dimensionless quartile curves.

See Sec. 7.1.3 Derivation of the Huff Storm for background information or Sec. 7.1.5 P(t)/Ptot for Four Huff Quartiles for the data used for this distribution.

3.1.5.2 Rainfall depth

For this storm type you must specify the total depth of rainfall in millimetres or inches. This will depend on climatic factors and the return interval in years. It is common to use the Intensity - Duration - Frequency curve for the area from which the total depth can be estimated for a given storm duration and return interval in years.

The values below are typical for some regions in Southern Ontario, Canada.

Return period	Depth		% of 5 year
(years)	(mm)	(inch)	(%)
2	38.0	1.50	78%
5	48.6	1.90	100%
10	55.7	2.20	115%
25	64.5	2.55	133%
50	71.1	2.80	146%
100	77.6	3.05	160%

See also Gumbel Distribution Sec. 3.1.5.3.

3.1.5.3 Gumbel Distribution

Estimates for extreme rainfall events can be expressed in terms of the average and standard deviation of the annual maximum series. Thus:

[3.3] $$x_T = \mu_x + K_T \sigma_x$$ where

x_T = magnitude of the T year event

μ_x = mean of the annual maximum series

σ_x = standard deviation of the annual maximum series

K_T = frequency factor which depends on return period T

The Gumbel (double exponential) distribution is often used to describe the frequency factor for extreme rainfall. This is expressed as:

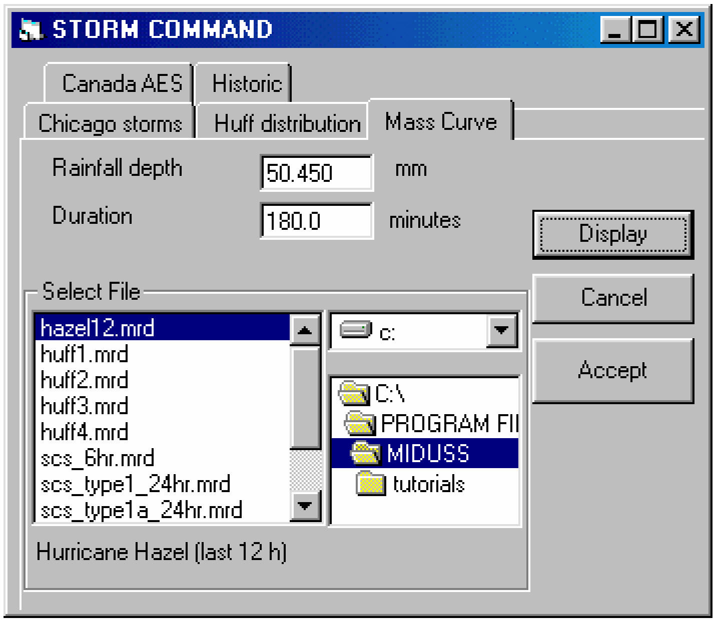
[3.4] $$K_T = \frac{-\sqrt{6}}{\pi}\left(0.5772 + \ln.\ln\frac{T}{T-1}\right)$$

e.g.

T (years)	2	5	10	25	50	100
K	-0.164	0.719	1.305	2.044	2.592	3.137

Values for the mean and standard deviation of the annual maximum series can often be estimated from rainfall frequency maps. See Applied Hydrology (Chow, Maidment and Mays), Section 12.2 *Extreme Value Distribution* for a good discussion of these distributions.

3.1.6 Mass Rainfall Distribution



Rainfall depth 50.450 mm

Duration 180.0 minutes

hazel12.mrd
huff1.mrd
huff2.mrd
huff3.mrd
huff4.mrd
scs_6hr.mrd
scs_type1_24hr.mrd
scs_type1a_24hr.mrd

Hurricane Hazel (last 12 h)

Figure 3-7 – The Mass Curve distribution tab of the Storm command

MIDUSS contains a number of files with the extension '.**mrd**' which stands for Mass Rainfall Distribution. These files reside in the MIDUSS directory (typically **C:** \ **Program Files** \ **MIDUSS** \..). You can use one of these pre-defined patterns or create a special one for your own use to define a customized non-dimensional mass rainfall distribution curve similar to the patterns used for the Huff storms.

The data form shown above is opened when you select this option. The default location of the *.mrd files is indicated by the Drive (e.g. C:\) and Directory (e.g. C:\Program Files\MIDUSS\) and all files with the extension .mrd are listed in the Files List box. (Hint: If the file List box shows ‘All Files’, double click on the drive and then the directory to show only files with the extension *.mrd)

By selecting (i.e. clicking on) one of the files a brief description is displayed below the list box. In addition to the selected non-dimensional distribution you must specify the total depth and duration for the storm. These are discussed below. When data has been entered press the [Display] command button to see a graphical plot of the storm hyetograph and a table of rainfall intensities. You can change any of the values or the distribution pattern and press [Display] again to see the effect of the change. When you are satisfied with the storm, press [Accept] to define the storm.

Irrespective of the number of points used to describe the distribution, the rainfall intensities are discretized in terms of the time step defined for the MIDUSS session using linear interpolation. The interpolated values are then scaled up by the total depth of rainfall. More details are provided in Chapter 7 *Hydrological Theory*. Derivation of the User Defined Distribution.

3.1.6.1 Drive for *.MRD file

When you install MIDUSS all the files of type *.MRD are located in the MIDUSS directory. If this is typically C:\Program Files\MIDUSS\ then the Drive for the *.MRD files will be 'C:' If you create special mass rainfall distribution files - either an edited copy of the pre-defined ones, or a customized one of your own choosing - you may wish to store these in a particular job directory. You can use the Drive list box to navigate to where your *.MRD files are located.

3.1.6.2 Directory for *.MRD file

When you install MIDUSS all the files of type *.MRD are located in the MIDUSS directory. If this is typically C:\Program Files\MIDUSS\ then the Directory for the *.MRD files will be 'C:\Program Files\MIDUSS\'. If you create special mass rainfall distribution files you may wish to store these in a particular Job directory. You can use the Directory list box to navigate to where your *.MRD files are located.

3.1.6.3 Select *.MRD file

Once the drive and directory of the path have been selected you can click on the particular *.MRD file to be used. The description in the text box below helps to ensure you have made the right choice. The file will not be processed until you press the [Display] command button. Note that the [Display] button is not enabled until a file has been selected.

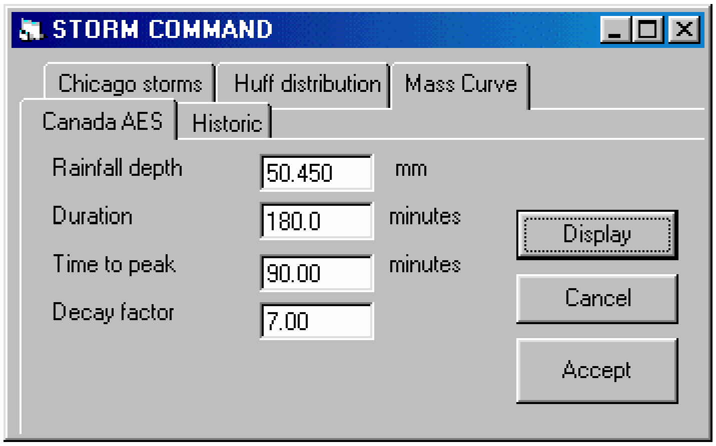
3.1.6.4 Customized *.MRD files

If you create a customized mass rainfall distribution file you should copy the format of one of the pre-defined files as illustrated below

```
SCS 6 hour distribution   # 1 - Description
51                        # 2 - Number of values N
0.000                     # 3 - Initial zero value
0.008
0.016                       Intermediate values
0.024                       defining (N-1)
0.032                       increments.
0.040
  :                       You can use
0.976                       any number of
0.984                       points > 2.
0.992
1.000                     # N+2 - Final value of 1.0
```

The **Edit Storm** command in the Tools menu provides a simple procedure to either edit and existing *.MRD file or create a new one. See Section 11.7 *Editing a Storm*.

3.1.7 Canadian AES 1-hour Storm



Rainfall depth 50.450 mm

Duration 180.0 minutes

Time to peak 90.00 minutes

Decay factor 7.00

Figure 3-8 – The Canadian AES tab of the Storm command

The form shown in Figure 3-8 is used to define a simple two parameter design storm which has a linear rising portion followed by an exponentially decreasing curve. The possibility of reversing the linear and exponential segments is suggested in the original publication (Watt *et al*) but this option is not currently supported in MIDUSS. A definition sketch is shown in Chapter 7 *Hydrological Theory*, Canadian 1- hour storm derivation

The parameter values required for this option are the depth (mm or inches) and duration (minutes) of the rainfall the time to peak intensity (minutes) and the decay coefficient K.

You should note that the proposal by Watt *et al* is intended to be used only for 1-hour storms since the data used for the work was limited to this duration. However, MIDUSS allows you to define other values of the duration. Be careful if suggested values for the time to peak are taken from the original reference since these are intended specifically for 60 minute storms. For that reason, suggested values in Chapter 7 *Hydrological Theory*, 'Suggested tp values for locations in Canada' show time to peak in minutes and also as a fraction of the duration but care should be taken in using these.

When the data has been entered press the [Display] command button to see a graphical plot of the storm hyetograph and a table of rainfall intensities. You can change any of the values and press [Display] again to see the effect of the change. When you are satisfied with the storm, press [Accept] to define the storm.

3.1.7.1 AES Time to Peak

Suggested values of time to peak are intended for 60 minute storms. For other values of duration use the values of **tp /Duration** with caution.

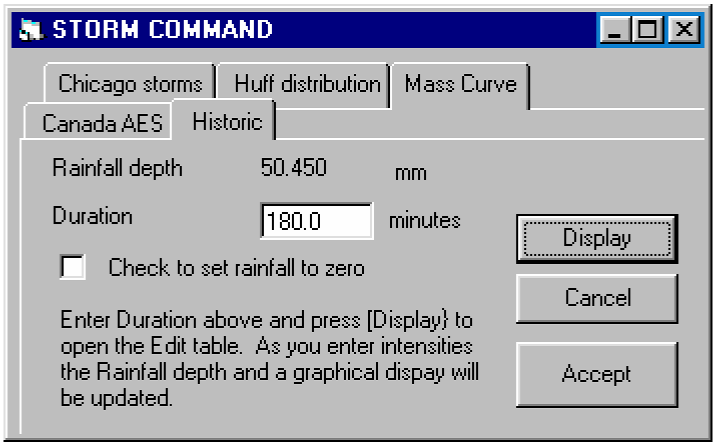
Location	Tp (minutes)	Tp/Duration
Yukon	20	0.33
B.C.(coast)	28	0.47
B.C.(interior), Prince George	13	0.22
Alberta	17-18	0.29
Saskatchewan	23-24	0.39
Manitoba (Brandon, Churchill)	31	0.52
Manitoba (Winnipeg)	25	0.42
Ontario (Timmins, Thunder Bay)	24-25	0.41
Ontario (Ottawa, Kingston, Windsor)	26-27	0.44
Ontario (Toronto, Sudbury)	21	0.35
Quebec (Montreal)	27	0.45
Quebec (Val D'Or, Quebec City)	23	0.38
New Brunswick (Fredericton)	17	0.28
Nova Scotia, Newfoundland	26-28	0.45

3.1.7.2 AES Decay factor

Values of the decay coefficient taken from the original publication are shown below.

Province	K value
B.C.(coastal region)	5
Yukon, New Brunswick, Nova Scotia, Newfoundland	6
B.C.(interior), Alberta, Saskatchewan, Manitoba, Ontario, Quebec	7

3.1.8 Historic Storm



Rainfall depth 50.450 mm

Duration 180.0 minutes

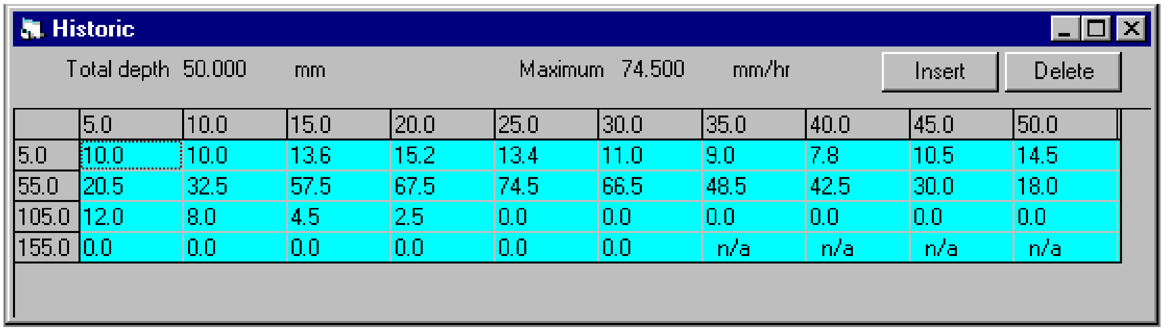
☐ Check to set rainfall to zero

Enter Duration above and press [Display} to open the Edit table. As you enter intensities the Rainfall depth and a graphical dispay will be updated.

Figure 3-9 – The Historic storm tab of the Storm command

This storm option lets you define an observed or historic rainfall event. The form illustrated here prompts you to specify the duration in minutes. Depth, intensity and distribution are defined by entering intensity values in a Table that is opened when you press the [Display] button.

If you set a check mark in the box labeled 'Check to set rainfall to zero' the table will be opened with all values set to zero. Otherwise, the intensity values for the currently defined storm are copied into the table where they can be edited. Notice that when you are using any of the other four options in the Storm command, the rainfall intensities displayed do not become the "currently defined storm" as soon as you have pressed the [Display] button. Thus, if you wish to generate (say) a 3rd quartile Huff storm and then edit it using the Historic storm option you must first use the ‘Huff distribution’ tab, define depth, duration and the desired quartile and then click [Display]. If you now select the ‘Historic’ tab the Huff rainfall will be imported into the Historic storm table for editing.



Total depth 50.000 mm

Maximum 74.500 mm/hr

	5.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	45.0	50.0
5.0	10.0	10.0	13.6	15.2	13.4	11.0	9.0	7.8	10.5	14.5
55.0	20.5	32.5	57.5	67.5	74.5	66.5	48.5	42.5	30.0	18.0
105.0	12.0	8.0	4.5	2.5	0.0	0.0	0.0	0.0	0.0	0.0
155.0	0.0	0.0	0.0	0.0	0.0	0.0	n/a	n/a	n/a	n/a

Figure 3-10 – The Historic storm data entry grid

The table shown above was generated by first creating and 'Displaying' a 3rd quartile Huff storm with a duration of 120 minutes and a total depth of 50 mm. Then the Historic tab was selected, the duration increased to 180 minutes and with the 'Check to set rainfall to zero' checkbox left empty (unchecked). Note the extra 12 cells (60 minutes) with zero values.

You can navigate around the table using the arrow keys or by clicking the mouse pointer on a cell of the table. You will notice a slightly heavier outline around the selected cell. You can type a new value into the current cell and move to the next cell with an arrow key. This will:

- overwrite the cell contents

- change the total depth in the table and the peak intensity if the new value is large enough.

- change the total depth in the **Storm/Historic** form

- update the graphic display of the Historic storm.

You can use the [Delete] button to delete the current cell and move all of the cells after this point back in time by one time step. The [Insert] button opens up an empty cell in front of the currently defined cell and moves all of the cells forward by one time step.

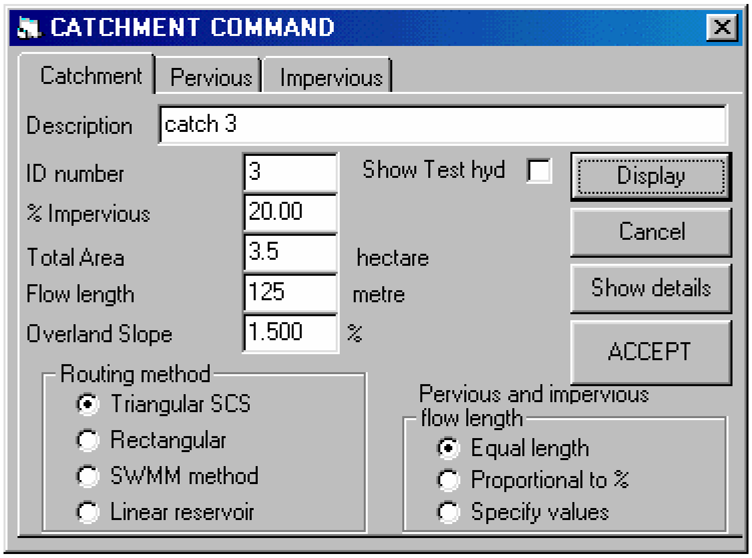
When you are satisfied with the historic storm press the [Accept] key in the **Storm/Historic** form.

3.1.8.1 Check to set Rainfall to Zero

If this checkbox is left empty the Historic storm table will be opened to show the rainfall intensity values for the currently displayed. The currently defined storm is one which has been created and accepted by a previous use of the Storm command or a storm hyetograph which has been imported by use of the **File / Load file / Rainfall** command.

If a check is entered in the box by clicking on it the values in the Historic storm table are all set to zero.

3.2 Catchment Command



Description: catch 3

ID number: 3

% Impervious: 20.00

Total Area: 3.5 hectare

Flow length: 125 metre

Overland Slope: 1.500 %

Show Test hyd ☐

Routing method
● Triangular SCS
○ Rectangular
○ SWMM method
○ Linear reservoir

Pervious and impervious flow length
● Equal length
○ Proportional to %
○ Specify values

Figure 3-11 – The total Catchment tab of the Catchment command

The Catchment command allows you to describe a sub-catchment and generate the runoff hydrograph for the design storm previously defined in the Storm command. The pervious and impervious fractions of the catchment are modelled separately and the two hydrographs are then added together. The process generally involves the following steps.

Define the total catchment area and percent impervious etc.

Select a method to define overland flow length on the pervious and impervious areas

Select a model to estimate rainfall losses

Compute the effective rainfall hyetograph for the impervious fraction

Compute the effective rainfall hyetograph for the pervious fraction

Select a model for routing the overland flow

Compute the runoff hydrographs (pervious, impervious and total)

The available options for overland flow routing are:

Combine effective rainfall with a triangular response function

Combine effective rainfall with a rectangular response function

Combine effective rainfall with a response function defined by a linear reservoir

Compute runoff from a surface water budget as in the SWMM Runoff block

The same rainfall loss model is used for both pervious and impervious areas. The models available to estimate rainfall losses are:

The SCS method   (not available for the SWMM Runoff routing option)

The Horton equation (moving curve method)

The Green and Ampt method

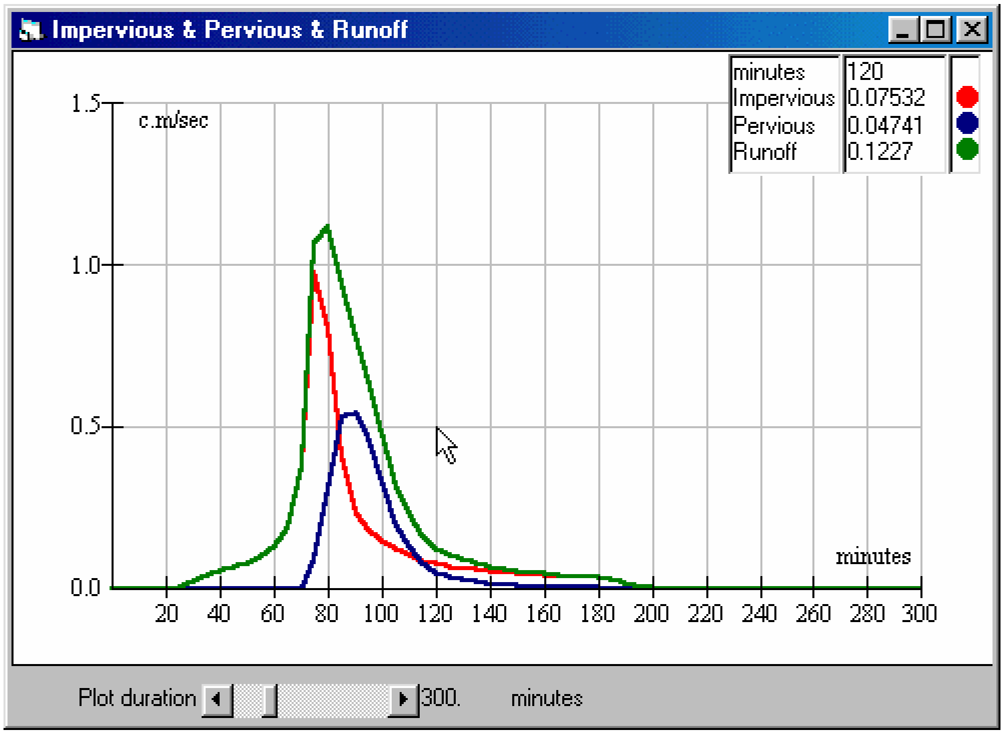
On the Pervious (Tab #2) form you can also click on any of the three options for infiltration method to see the data requirements and get specific information for each of the parameters.

The results displayed by the catchment command are described in Reviewing the Catchment Results

3.2.1 Reviewing the Catchment Command Results

The results of the catchment command can be reviewed in a number of ways.

A graphical display in the top right corner shows the runoff from the pervious and impervious areas and the total runoff from the catchment. This time to peak for the two components may be different and it is not uncommon to see a total runoff hydrograph which exhibits two peaks. The graph window contains some additional features which help you to interpret the plot. The topic Graph Window Features (Sec. 3.2.2.) describes these in more detail.



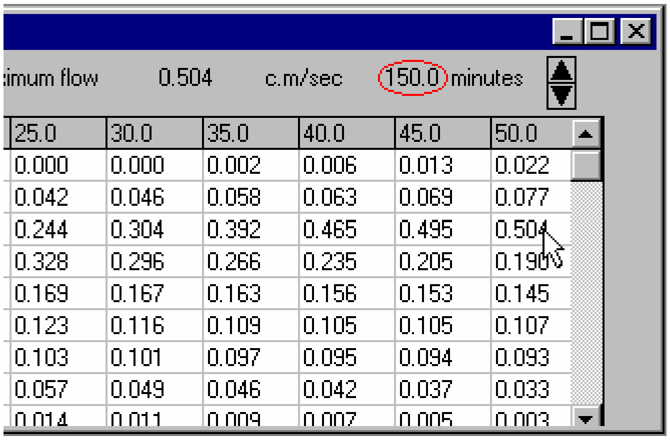
minutes	120	
Impervious	0.07532	🔴
Pervious	0.04741	🔵
Runoff	0.1227	🟢

Plot duration 300. minutes

Figure 3-12 – A typical plot of hydrograph components

A tabular display of the total hydrograph is shown in the lower left corner of the screen. If the [Display] button is pressed from either the Pervious or Impervious tabs on the form, the effective rainfall on the relevant area is displayed in the table. An exception to this is when the SWMM Runoff algorithm is used which uses a surface water budget rather than the effective rainfall.

The fragment of table shown here has been increased in height by 'dragging' the top edge upwards to display as many rows as you wish. Also, the figure shows the cursor over the right hand cell in the third row. This causes the time for that cell to be displayed as circled in red.

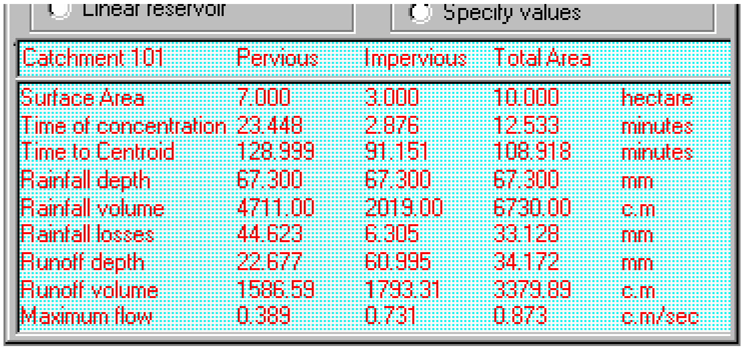


imum flow 0.504 c.m/sec 150.0 minutes

25.0	30.0	35.0	40.0	45.0	50.0
0.000	0.000	0.002	0.006	0.013	0.022
0.042	0.046	0.058	0.063	0.069	0.077
0.244	0.304	0.392	0.465	0.495	0.504
0.328	0.296	0.266	0.235	0.205	0.190
0.169	0.167	0.163	0.156	0.153	0.145
0.123	0.116	0.109	0.105	0.105	0.107
0.103	0.101	0.097	0.095	0.094	0.093
0.057	0.049	0.046	0.042	0.037	0.033
0.014	0.011	0.009	0.007	0.005	0.003

Figure 3-13 – Part of the tabular display of the Runoff hydrograph

Some summary statistics can be displayed by pressing the [Show Details] button. A typical display is shown in Figure 3.14 below.

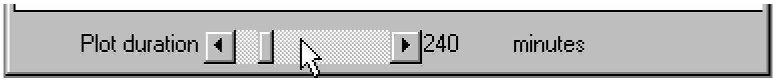


Catchment 101	Pervious	Impervious	Total Area	
Surface Area	7.000	3.000	10.000	hectare
Time of concentration	23.448	2.876	12.533	minutes
Time to Centroid	128.999	91.151	108.918	minutes
Rainfall depth	67.300	67.300	67.300	mm
Rainfall volume	4711.00	2019.00	6730.00	c.m
Rainfall losses	44.623	6.305	33.128	mm
Runoff depth	22.677	60.995	34.172	mm
Runoff volume	1586.59	1793.31	3379.89	c.m
Maximum flow	0.389	0.731	0.873	c.m/sec

Figure 3-14 – The Catchment command provides details as an option.

The final step is to Accept the results of the catchment command .

3.2.2 Graph Window Features

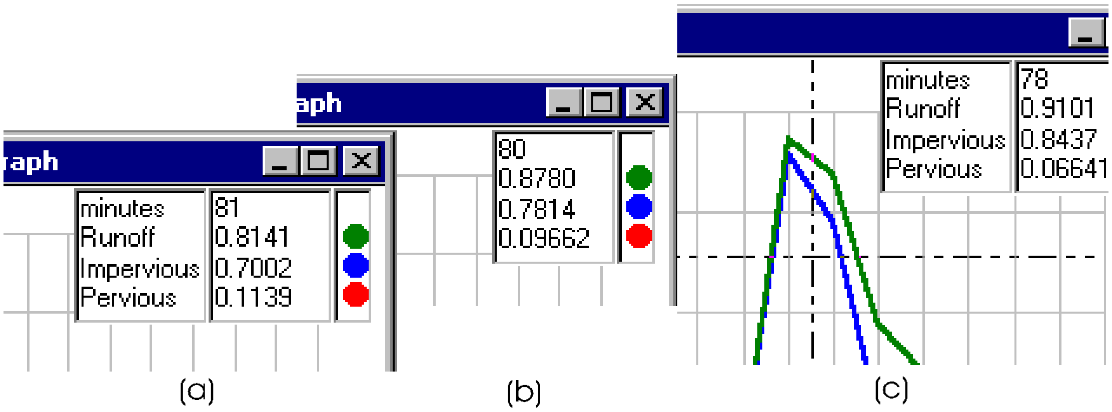


Plot duration 240 minutes

Figure 3-15 – The displayed time-base of the hydrograph plot can be adjusted.

At the bottom of the graph window a slider control lets you vary the time base which is plotted in the window. You can adjust the time by clicking either on the bar or on the arrow at the extremity. Clicking on the right increases the time (thus compressing the plot); clicking on the left reduces the time. Click on the arrows for greater control. The time base is displayed to the right of the slider

Figure 3-16 illustrates some of the details which are available in any plot of a hyetograph or hydrograph



(a)

minutes	81	
Runoff	0.8141	●
Impervious	0.7002	●
Pervious	0.1139	●

(b)

80	
0.8780	●
0.7814	●
0.09662	●

(c)

minutes	78
Runoff	0.9101
Impervious	0.8437
Pervious	0.06641

Figure 3-16 – Some details in the plot window

Figure 3-16(a) shows the default state of the legend which is displayed whenever the mouse pointer is over the plotting area. The small windows show respectively the time and name of the functions displayed, the values for each of these and a colour code keyed to the plotted functions. Typically the legends contain two or more rows of information.

By clicking with the primary (left) mouse button on the window showing the coloured circles, the name legend can be toggled off or on as shown in Figure 3-16(b). The values displayed change dynamically when the mouse pointer is moved. Moving the mouse out of the left side of the plot window causes the legend windows to be hidden.

If you hold down the secondary mouse button while moving the mouse, a pair of cross-hairs is displayed as in Figure 3-16(c). The cross-hairs are removed when the cursor is moved without the secondary button held down.

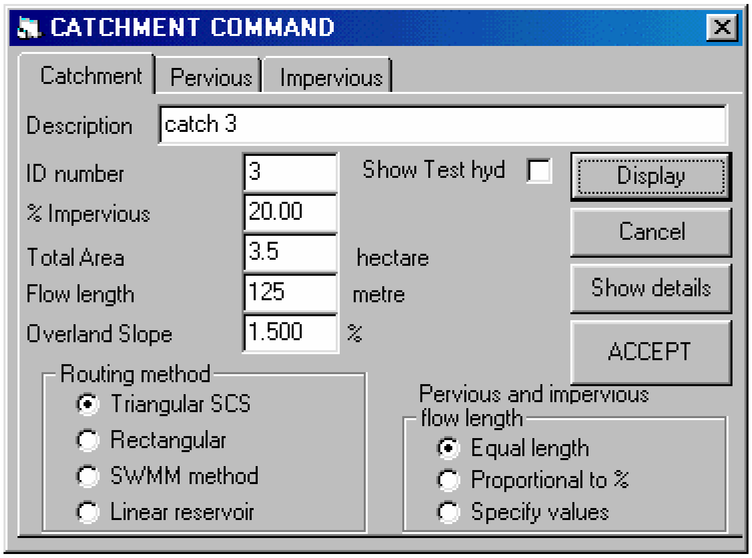
Clicking with the primary mouse button in the legend window containing the numerical values can alter the type of grid display. The options are selected in the following revolving sequence – no grid, horizontal, vertical, or both. The style in use when you exit MIDUSS is remembered and used as the default in the next session.

3.2.3 Accepting the Catchment Command

You can change any of the parameters in the catchment form and press the [Display] button to update the graph, the table and the "details" window. Each time a parameter is altered the [Accept] button is disabled and is re-enabled only when the [Display] button is pressed to refresh the screen. This prevents you from mistakenly 'accepting' a display which has not been updated.

When you are satisfied with the result you can press the [Accept] button to save the data and results to the output file. All of the catchment windows are closed and the peak flow summary table is updated by adding a row with the new runoff hydrograph.

3.2.4 Data for the Total Catchment



Description: catch 3
ID number: 3
Show Test hyd ☐
% Impervious: 20.00
Total Area: 3.5 hectare
Flow length: 125 metre
Overland Slope: 1.500 %

Routing method
● Triangular SCS
○ Rectangular
○ SWMM method
○ Linear reservoir

Pervious and impervious flow length
● Equal length
○ Proportional to %
○ Specify values

Figure 3-17 – Data required for the total Catchment area

As illustrated, the data required comprises a verbal description, an ID number for the catchment, the percentage of impervious area, the total catchment area and the length and average slope (as %) of the overland flow surface.

As you might expect, the total area and the percentage of impervious surface are the most important parameters in determining the volume of runoff and the peak flow of the runoff hydrograph.

On this form you can select the overland routing method from:

The SCS triangular response function

A rectangular response function

The SWMM Runoff algorithm

A response function equivalent to the response of a linear reservoir to an instantaneous unit input (sometimes referred to as a Dirac δ-function),

3.2.4.1 Catchment Description

This text entry box can contain a verbal description of the catchment area or merely an alpha-numeric identifier. A unique integer identifier ID number is still required. Both items appear in the output file. Each entry of an ID number is checked against a list of previously defined numbers to avoid duplication.

3.2.4.2 Catchment ID Number

This number is used to identify the sub-catchment being defined. Each entry of an ID number is checked against a list of previously defined numbers to avoid duplication. Use a positive integer of not more than 1,999,999,999.

3.2.4.3 Catchment Show Test Hyd

When this check box is selected (i.e. checked) the graphical display generated when the [Display] button is clicked includes the user-defined Test hydrograph as well as the hydrographs for total Runoff and the runoff components from the pervious and impervious fractions. This can be of value in calibrating runoff parameters to match an observed runoff hydrograph

3.2.4.4 Percent Impervious

The percentage of impervious area can be any positive value from 0% to 100%. This parameter is second only to the total area in importance in determining the runoff.

3.2.4.5 Total Catchment Area

This is the total contributing area including both pervious and impervious fractions. The units must be either hectares (metric or SI) or acres (imperial or US Customary). The area is the most significant parameter in determining peak and volume of runoff.

3.2.4.6 Overland Flow length

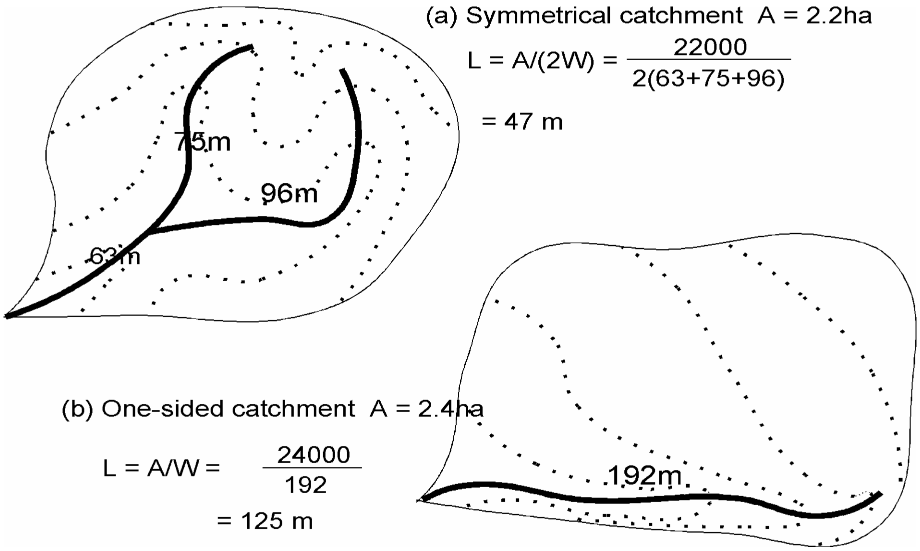


Figure 3-18 – Approximate ways of estimating overland flow length

The overland flow length is the average overland flow length in metres or feet from the edge of the sub-catchment to the main drainage conduit (i.e. pipe, gutter or channel). An approximate estimate for a symmetrical area such as Figure 3-18(a) is:

$$[3.5] \qquad Length = \frac{Area}{2 \times Channel Length}$$

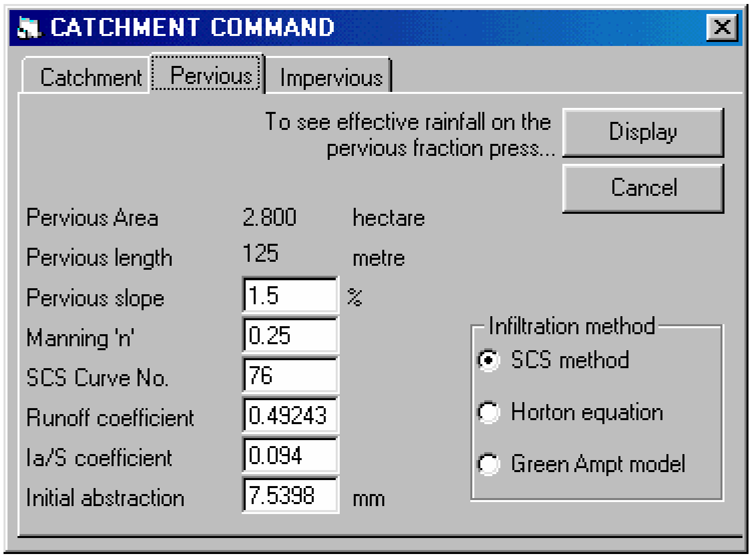
A one-sided catchment as in sketch (b) may often result from highway or railway construction. The length can be approximated as:

[3.6] $$Length = \frac{Area}{Channel\,Length}$$

3.2.4.7 Overland Slope

The slope of the overland flow is the average surface slope from the edge of the catchment to the main conduit or channel estimated along a line of greatest slope (i.e. normal to the contours). Do <u>not</u> use the maximum height difference in the sub-catchment divided by the length of the main drainage channel.

3.2.5 Data for the Pervious Area



To see effective rainfall on the pervious fraction press...

Pervious Area 2.800 hectare
Pervious length 125 metre
Pervious slope 1.5 %
Manning 'n' 0.25
SCS Curve No. 76
Runoff coefficient 0.49243
Ia/S coefficient 0.094
Initial abstraction 7.5398 mm

Infiltration method
● SCS method
○ Horton equation
○ Green Ampt model

Figure 3-19 – Data required for the Pervious fraction (SCS method selected)

Figure 3-19 above displays the catchment data form when the Pervious tab has been clicked.

The form shows the parameters required to define the rainfall losses when the SCS Infiltration model is selected. The required infiltration parameters will change with the selection of different infiltration methods.

The area of the pervious fraction is indirectly defined in terms of the total area and the percentage of impervious surface. You can change this from the Catchment tab.

Definition of the Pervious flow length can be done either explicitly or can be indirectly defined by the option choice for Pervious and Impervious flow length in the Catchment tab form. If the ‘Specify values’ Option button is selected in the ‘Pervious and impervious flow length’ frame (see Figure 3-17) the data fields for Pervious length and Impervious length change to text boxes to allow data entry. (See Section 3.2.5.2)

3.2.5.1 Pervious Manning 'n'

The values listed below are typical for overland sheet flow on pervious surfaces with various types of vegetation. These are not suitable for flow in channels.

Surface	**Manning ‘n’**
Dense growth	0.4 – 0.5
Pasture	0.3 – 0.4
Lawns	0.2 – 0.3
Bluegrass sod	0.2 – 0.5
Short-grass prairie	0.1 – 0.2
Sparse vegetation	0.05 – 0.13
Bare clay/loam soil	0.013 – 0.03

3.2.5.2 Pervious Flow length

There are three options to specify the overland flow length on the pervious and impervious areas.

- Use the same length for both
- Make the length proportional to the percentage of the area
- Specify the lengths

These are illustrated in terms of an idealized rectangular catchment in Figure 3-20.

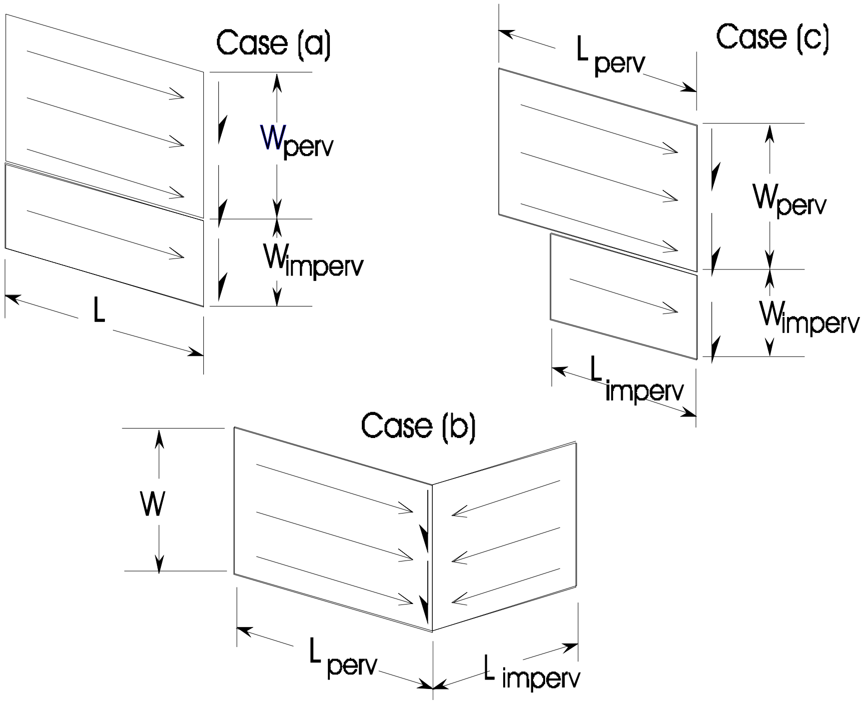


Figure 3-20 – Options for defining overland flow length

For the second option the pervious flow length is set equal to the overland flow length defined in the Catchment tab. The impervious flow length is then calculated as

$$[3.7] \qquad L_{impervious} = L_{pervious} \frac{\%I}{(1-\%I)}$$

As an example assume L = 100 m and %I = 20%, then the impervious flow length is 100 x 0.2/0.8 or 25 m. and the pervious flow length is 100m.

If you select the third option text boxes are opened to allow you to specify any value.

3.2.5.3 Pervious Slope

The default value is the same as the average overland slope for the total area. However, you can specify a different value if you wish.

3.2.5.4 Pervious Data for SCS Infiltration

When the SCS Infiltration option is selected you have some choice as to how the required data values are entered. The basic parameter is the SCS Curve number which depends on the soil type and land use.

As an alternative, you can define a value for the volumetric runoff coefficient. This will cause the equivalent SCS CN value to be computed for the current rainfall event and displayed. Likewise, entry of a SCS CN value causes the equivalent runoff coefficient to be computed and displayed.

Another important parameter is the depth of the initial abstraction. This can be specified explicitly or you can provide a coefficient to define the initial abstraction as a fraction of the storage potential S which is a function of CN. The relationship is given below for both metric and U.S. Customary units.

[3.8] $$S = \frac{1000}{CN} - 10$$

S is in inches for U.S Customary units.

[3.9] $$S = 25.4\left(\frac{1000}{CN} - 10\right)$$

S is in millimetres for metric/SI units

3.2.5.5 Pervious SCS Curve number

CN depends on Soil Type, Antecedent Moisture and Land Use.

See Section 3.2.5.6 for Soil Types classification.

See Section 3.2.5.7 for Dry and Wet CN values for variation from normal conditions.

	Soil type			
Land Use	**A**	**B**	**C**	**D**
Cultivated land with no conservation treatment	72	81	88	91
Cultivated land with conservation treatment	62	71	78	81
Pasture in poor condition	68	79	86	89
Pasture in good condition	39	61	74	81
Woodland - poor cover	45	66	77	83
Woodland - good cover	25	55	70	77
Park land - >75% grass	39	61	74	80
Park land - 50-75% grass	49	69	79	84

In some texts you may see values of *CN* quoted as a function of the percentage of impervious area. These are usually calculated as a weighted average assuming *CN*impervious = 98 and *CN*pervious equal to the value for ‘Pasture in good condition’ for the various soil types A, B, C or D. This is often done using an equation of the form:

[3.10] $$CN_{equiv} = \left(\%I\ CN_{imperv} + \left(100 - \%I\right)CN_{perv}\right)/100$$

where %I is the percentage of impervious area.

Values of *CN* estimated in this way are intended to be applied to the **total** catchment assuming other parameters to be the same for both pervious and impervious areas. Many programs (including MIDUSS) compute the runoff from the pervious and impervious fractions separately and then add the two hydrographs. In such cases, it is most important that you **do not use** a composite value of *CN* since this would 'double count' the impervious fraction and greatly exaggerate the runoff prediction.

3.2.5.6 SCS Soil Types

The following four classifications of soil are used.

Type A	Deep, very well drained sand or gravel
Type B	Moderately well drained soil with medium texture
Type C	Fine soil with an infiltration impeding layer
Type D	Clay; soil over rock; soil with a permanent high water table

3.2.5.7 Dry and Wet CN values

With normal moisture conditions the Curve number is defined as *CN2*. For very dry or very wet antecedent conditions the corresponding values *CN1* and *CN3* can be expressed approximately as simple functions of *CN2*.

Dry:

$$[3.11] \qquad CN1 = CN2 - 2.45(100 - CN2)^{0.62}$$

$$S1 = 2.3\,S2$$

Wet:

$$[3.12] \qquad CN3 = CN2 + 0.60(100 - CN2)^{0.953}$$

$$S3 = 0.4\,S2$$

3.2.5.8 Runoff coefficient

The runoff coefficient in MIDUSS is defined in volumetric terms,

i.e. C = Runoff depth/Rainfall depth

In the SCS Infiltration method the time history of runoff depth (or the effective rainfall) is computed as:

[3.13] $$Q(t) = \frac{(P(t) - I_a)^2}{(P(t) + S - I_a)}$$

where *Q(t)* = accumulated depth of runoff to time t

P(t) = accumulated depth of rainfall to time t

Ia = initial abstraction

3.2.5.9 Ia/S Coefficient

This ratio indirectly defines the Initial Abstraction as a fraction of the potential storage depth S. Values of 0.05 to 0.1 are reasonable. The value of 0.2 originally recommended by SCS is now considered by many engineers to be too high. It is often easier to define the initial abstraction directly as a depth. If you do this MIDUSS will compute the corresponding value of *Ia/S* as the ratio of the initial abstraction Ia and the storage potential S (see equations 3.8, 3.9).

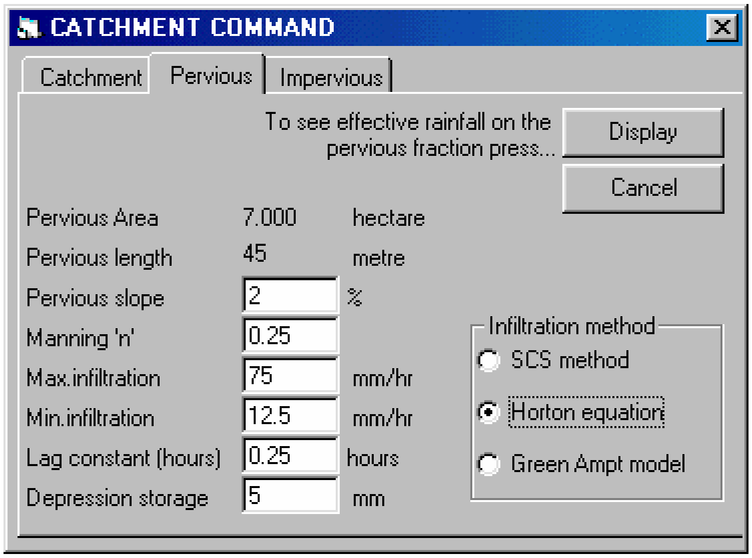
a function of *S*.

3.2.5.10 Initial Abstraction

For impervious surfaces this value may range from 0.5 - 1.5 mm (0.02 - 0.06 inch) depending on the type and steepness of the surface. It is roughly equivalent to the surface depression storage.

For pervious areas the initial abstraction may be between 5 - 10 mm (0.2 - 0.4 inch) depending on vegetative cover and tree canopy.

3.2.5.11 Pervious Data for Horton Equation



To see effective rainfall on the pervious fraction press...

Pervious Area	7.000	hectare
Pervious length	45	metre
Pervious slope	2	%
Manning 'n'	0.25	
Max.infiltration	75	mm/hr
Min.infiltration	12.5	mm/hr
Lag constant (hours)	0.25	hours
Depression storage	5	mm

Infiltration method
○ SCS method
● Horton equation
○ Green Ampt model

Figure 3.21 – Data required for the pervious fraction (Horton method selected)

The Horton Infiltration option requires four parameters to compute the rainfall losses. These are:

The initial infiltration rate fo

The final infiltration rate fc

The exponential decay time constant K

and The surface depression storage Ysd

For a more complete understanding of the method you should refer to the section in Chapter 7 *Hydrological Theory*; The Horton Equation.

3.2.5.12 Horton fo for Pervious Areas

	Soil Group and Type	**mm/h**	**inch/h**
A	sand/gravel, sandy loam	250	10.0
B	Silty loam	200	8.0
C	Sand-clay-loam	125	5.0
D	Clay, soil over rock	75	3.0

Values are for dry soil. Allow for pre-wetting

3.2.5.13 Horton fc for Pervious Areas

	Soil Group and Type	**mm/h**	**inch/h**
A	sand/gravel, sandy loam	25	1
B	Silty loam	13	0.5
C	Sand-clay-loam	5	0.2
D	Clay,	3	0.1
	Shallow soil over rock	0	0.01

3.2.5.14 Horton Lag Constant

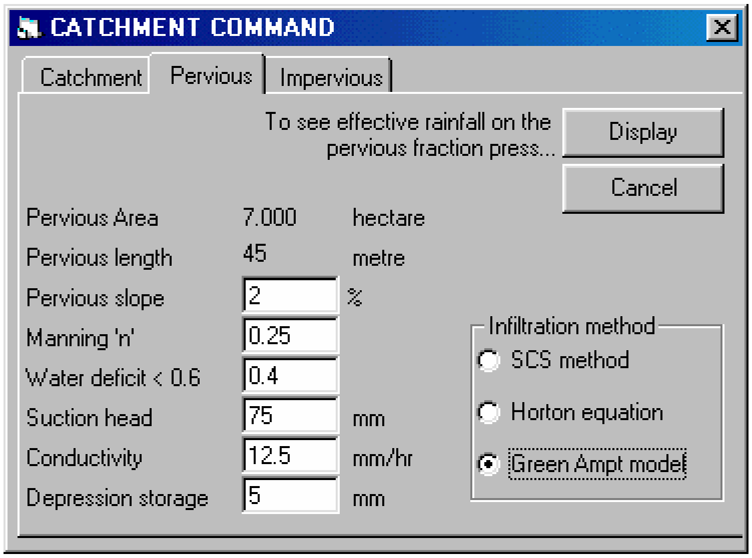
For pervious surfaces the lag K may vary from 0.25 to 0.5 hours depending on the soil type. The lag K has little or no physical significance. For impervious surfaces the lag K will be very short, e.g. 0.05 hours .If fo = 0.0 set K =0 also. Notice that in MIDUSS, the lag is specified as a time in hours whereas in many other models which use the Horton equation the lag coefficient is expressed as the reciprocal of the lag and is often expressed in units of 1/sec or $\sec^{-1}$. Thus a lag of 0.25 hours is equivalent to a coefficient of (1/900 sec) or 0.001111 $\sec^{-1}$.

3.2.5.15 Depression Storage

For pervious surfaces this may be 5 - 10 mm (0.2 - 0.4 inch) depending on surface type and slope. In general, steep surfaces retain less depth.

For impervious surfaces the value may be 0.5 - 1.5 mm (0.02 - 0.06 inch) depending on surface type and slope. Steep slopes retain less depth.

3.2.5.16 Pervious Data for Green & Ampt Method



To see effective rainfall on the pervious fraction press...

Pervious Area 7.000 hectare
Pervious length 45 metre
Pervious slope 2 %
Manning 'n' 0.25
Water deficit < 0.6 0.4
Suction head 75 mm
Conductivity 12.5 mm/hr
Depression storage 5 mm

Infiltration method
○ SCS method
○ Horton equation
● Green Ampt model

Figure 3.22 – Data required for the pervious fraction (Green and Ampt method selected)

The Green & Ampt Infiltration option requires four parameters to compute the rainfall losses. These are:

The initial soil moisture deficit *M*

The suction head across the wetting front *S*

The hydraulic conductivity in the soil *K*

and The surface depression storage *Ysd*

For a more complete understanding of the method you should refer to the sections in Chapter 7 *Hydrological Theory* which deal with the Green and Ampt method and also the Green and Ampt parameter Evaluation.

3.2.5.17 Water deficit

This is the difference between the initial water content of the surface layers of the soil and the saturated water content after the wetting front has passed through a layer. It is a dimensionless number normally less than 0.6. For a fully drained specimen of the following soil types the maximum effective porosity is as shown. The actual initial soil moisture deficit will depend on the antecedent rainfall.

Soil Type	M
sand	0.417
loamy sand	0.401
sandy loam	0.412
loam	0.434
silt loam	0.486
sandy clay loam	0.330
clay loam	0.309
silty clay loam	0.432
sandy clay	0.321
silty clay loam	0.423
clay	0.385

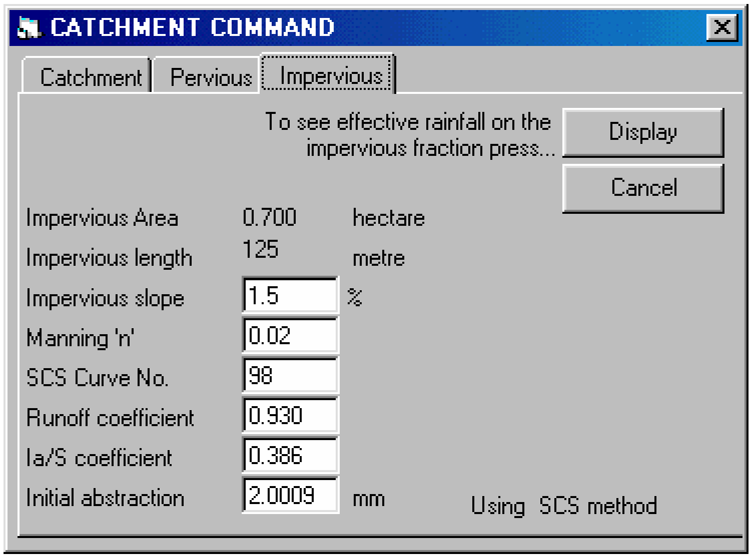
3.2.5.18 Suction Head

Soil type	Suction head S	
	inch	mm
Sand	1.949	49.5
Loamy sand	2.413	61.3
Sandy loam	4.335	110.1
Loam	3.500	88.9
Silt loam	6.567	166.8
Sandy clay loam	8.602	218.5
Clay loam	8.220	208.8
Silty clay loam	10.748	273.0
Sandy clay	9.410	239.0
Silty clay loam	11.504	292.2
Clay	12.453	316.3

3.2.5.19 Soil Conductivity

Soil type	Hydraulic conductivity	
	inch/h	mm/h
Sand	4.638	117.8
Loamy sand	1.177	29.9
Sandy loam	0.429	10.9
Loam	0.134	3.4
Silt loam	0.256	6.5
Sandy clay loam	0.060	1.5
Clay loam	0.039	1.0
Silty clay loam	0.039	1.0
Sandy clay	0.024	0.6
Silty clay loam	0.020	0.5
Clay	0.012	0.3

3.2.6 Data for the Impervious Area



To see effective rainfall on the impervious fraction press...

Impervious Area 0.700 hectare
Impervious length 125 metre
Impervious slope 1.5 %
Manning 'n' 0.02
SCS Curve No. 98
Runoff coefficient 0.930
Ia/S coefficient 0.386
Initial abstraction 2.0009 mm

Using SCS method

Figure 3-23 – Data required for the impervious fraction (SCS method selected)

Figure 3-23 displays the catchment data form when the Impervious tab has been clicked.

The form shows the parameters required to define the rainfall losses when the SCS Infiltration model is selected. The required infiltration parameters will change with the selection of infiltration model. To change the infiltration option you must use the Option buttons in the Infiltration method frame on the Pervious tab. When you return to the impervious tab your choice is shown on the lower right corner of the form.

The area of the impervious fraction is indirectly defined in terms of the total area and the percentage of impervious surface. You can change this from the Catchment tab.

Definition of the Impervious flow length can be done either explicitly or can be indirectly defined by the option choice for Pervious and Impervious flow length in the Catchment tab form.

3.2.6.1 Impervious Flow Length

There are three options to specify the overland flow length on the pervious and impervious areas.

- Use the same length for both
- Make the length proportional to the percentage of the area
- Specify the lengths

Refer to Figure 3-20 for more details on overland flow length options.

3.2.6.2 Impervious Slope

The default value is the same as the average overland slope for the total area. However, you can specify a different value if you wish.

3.2.6.3 Impervious Manning 'n'

Estimates vary widely depending on the surface and depth of flow.

Surface	**‘n’**
Very smooth asphalt	0.013
Very smooth concrete	0.015
Normal concrete	0.02
Rough concrete, paved areas	0.04
Rough paved areas with flow depth < 5 mm or 0.2 inch	0.10

3.2.6.4 Impervious Data for SCS Infiltration

When the SCS Infiltration option is selected you have some choice as to how the required data values are entered. The basic parameter is the SCS Curve number which depends on the type of surface.

As an alternative, you can define a value for the volumetric runoff coefficient. This will cause the equivalent SCS CN value to be computed for the current rainfall event and displayed. Likewise, entry of a SCS CN value causes the equivalent runoff coefficient to be computed and displayed.

Another important parameter is the depth of the initial abstraction. This can be specified explicitly or you can provide a coefficient to define the initial abstraction as a fraction of the storage potential S which is a function of CN. See Pervious Data for SCS infiltration for further details.

3.2.6.5 Impervious SCS Curve number

A totally impervious surface has a CN = 100. However, many apparently impervious surfaces have a small but finite degree of perviousness due to cracking or porosity. Suggested values are shown below.

Surface	**CN**
Well laid asphalt	100
Jointed concrete paving	99
Paved roads & parking lots	98
Well compacted gravel	91-96

3.2.6.6 Impervious Ia/S Ratio

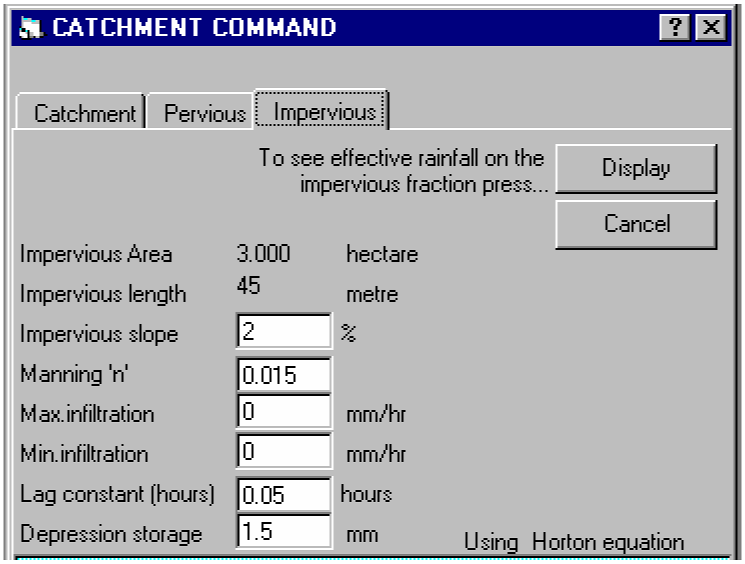
This ratio indirectly defines the Initial Abstraction as a fraction of the potential storage depth S. Since S is very small for an impervious surface it is difficult to define Ia/S with accuracy. For example if CN=98 then S = 5.2 mm (0.2 inch) so that Ia/S might reasonably be between 0.2 and 0.3.

It is often easier to define the initial abstraction directly as a depth. If you do this MIDUSS will compute the corresponding value of Ia/S as a function of S.

3.2.6.7 Impervious Initial Abstraction

For impervious surfaces this value may range from 0.5 - 1.5 mm (0.02 - 0.06 inch) depending on the type and steepness of the surface. It is roughly equivalent to the surface depression storage.

3.2.6.8 Impervious Data for Horton Equation



To see effective rainfall on the impervious fraction press...

Impervious Area 3.000 hectare
Impervious length 45 metre
Impervious slope 2 %
Manning 'n' 0.015
Max.infiltration 0 mm/hr
Min.infiltration 0 mm/hr
Lag constant (hours) 0.05 hours
Depression storage 1.5 mm
Using Horton equation

Figure 3-24 – Data required for impervious fraction (Horton method selected)

The Horton Infiltration option requires four parameters to compute the rainfall losses. These are:

The initial infiltration rate

The final infiltration rate

The exponential decay time constant

and The surface depression depth

For a more complete understanding of the method you should refer to the section in Chapter 7 *Hydrological Theory*; The Horton Equation.

3.2.6.9 Horton fo for Impervious Areas

For impervious surfaces *fo* is usually zero or a very small value such 2 mm/h (0.08 inch/h)

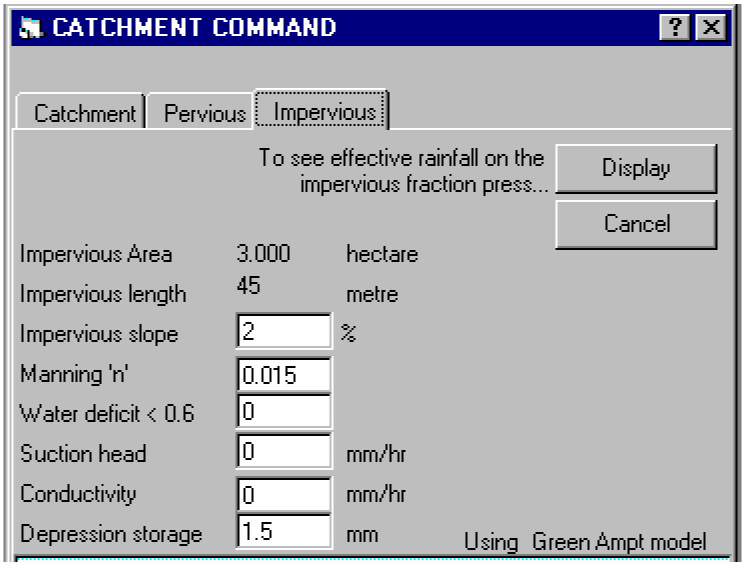
3.2.6.10 Horton fc for Impervious Areas

For impervious surfaces the value of *fc* should be zero.

3.2.6.11 Impervious Surface Depression Storage

See the general topic on Surface Depression Storage.

3.2.6.12 Impervious Data for Green & Ampt Method



To see effective rainfall on the impervious fraction press...

Impervious Area	3.000	hectare
Impervious length	45	metre
Impervious slope	2	%
Manning 'n'	0.015	
Water deficit < 0.6	0	
Suction head	0	mm/hr
Conductivity	0	mm/hr
Depression storage	1.5	mm

Using Green Ampt model

`Figure 3-25 – Data required for the impervious fraction (Green and Ampt method selected)

The Green & Ampt Infiltration option requires four parameters to compute the rainfall losses. These are:

The initial soil moisture deficit

The suction head across the wetting front

The hydraulic conductivity in the soil

and The surface depression storage

For a more complete understanding of the method you should refer to the sections in Chapter 7 *Hydrological Theory* which deal with the Green and Ampt method and also the Green and Ampt parameter Evaluation.

3.2.6.13 Impervious Water Deficit

For impervious surfaces the soil moisture deficit is normally equal to or very close to zero.

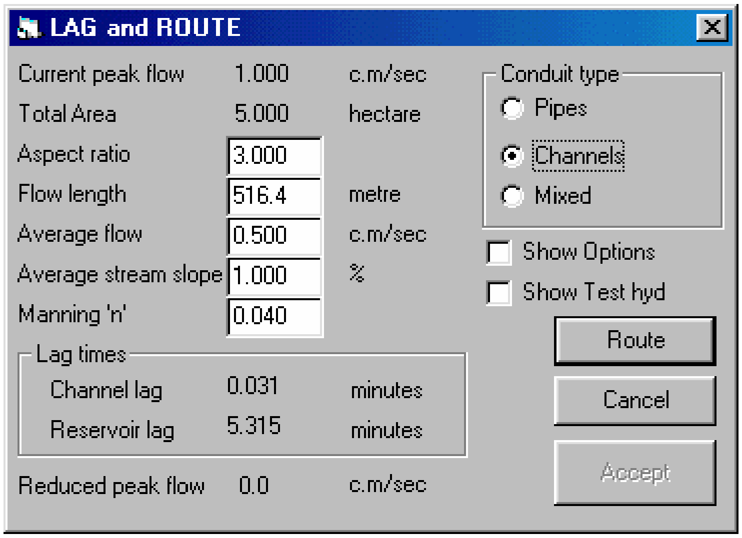
3.2.6.14 Impervious Suction Head

For impervious surfaces the suction head is zero.

3.2.6.15 Impervious Hydraulic Conductivity

For impervious surfaces the hydraulic conductivity is usually negligible.

3.3 Lag and Route Command


LAG and ROUTE

Current peak flow 1.000 c.m/sec
Total Area 5.000 hectare
Aspect ratio 3.000
Flow length 516.4 metre
Average flow 0.500 c.m/sec
Average stream slope 1.000 %
Manning 'n' 0.040

Lag times
Channel lag 0.031 minutes
Reservoir lag 5.315 minutes

Reduced peak flow 0.0 c.m/sec

Conduit type
○ Pipes
● Channels
○ Mixed

☐ Show Options
☐ Show Test hyd

Figure 3-26 – Data entry for the Lag and Route command

The Lag and Route option of the Hydrology menu is enabled only after the Catchment command has been completed and the direct runoff hydrograph has been accepted by pressing the [Accept] button. It is likely that you may want to use this only if the catchment area is quite large - typically over 100 ha or 250 acres.

A typical sequence of steps is as follows:

- Set the Catchment Aspect Ratio.
- Adjust the estimate of longest drainage path if desired.
- Adjust the average flow if desired.
- Adjust the average slope if desired.
- Adjust the conduit roughness if desired.
- Select an option for Conduit Type - Pipes, Channels or Mixed
- Change the Lag and Route Options
- The Lag and Route Operation

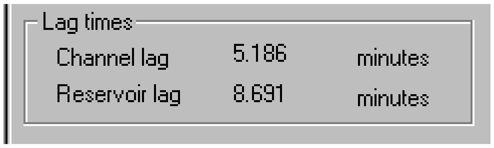
This option lets you simulate a very large, reasonably homogeneous catchment as a single area while still using reasonable values for the overland flow length, slope and roughness. This is achieved by routing the direct runoff hydrograph (generated by the Catchment command) through a hypothetical linear channel and linear reservoir.

The method is largely undocumented and has been developed only for use within the MIDUSS package. You are therefore advised to experiment with the method to satisfy yourself that it is possible to reproduce the results of a highly discretized catchment with reasonable accuracy. This should be done before applying it for the first time to a job of any importance.

The data required is used to estimate the total time of travel of a water particle from the most remote point on the catchment drainage network to the outflow point. This is done by approximating the longest path of pipes or channels as the two edges of an equivalent rectangular area and assigning a slope and roughness to it. More details are provided in the section on Chapter 7 *Hydrological Theory*; Large Catchment Simulation.

These values are clearly rather coarse approximations and are intended as guidelines. A better estimate of length should be available from mapping of the area.

In addition to the data entry boxes, the form also displays the current peak flow and the total area of the last catchment area defined. For the current default values the lag of the linear channel and linear reservoir are displayed in minutes.



Lag times

Channel lag	5.186	minutes
Reservoir lag	8.691	minutes

Figure 3-27 – The lag times are estimated from the approximate data provided.

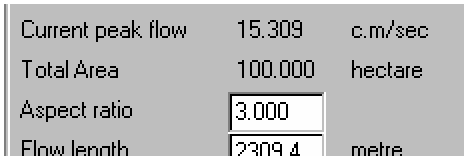
These values are altered indirectly by changing any of the other parameters. However, if you wish to experiment by setting the lag times explicitly, you can select the ‘Show Options’ check box and click on the ‘Select lags explicitly’ check box. (See Figure 3-36). The default value used for the Manning's 'n' roughness depends on whether the drainage network is predominantly pipes or channels.

When the [Route] button is pressed the reduced peak flow is displayed on the form. Also shown is a graphical display of the initial direct runoff hydrograph and the modified runoff hydrograph. The modified runoff is also shown in a tabular display.

You can continue to change any of the parameters and see the effect on the graphical or numeric displays. When satisfied you can press the [Accept] button which causes the summary peak flows table to be updated and the other forms are closed.

Since acceptance of the Lag and Route results changes the Runoff hydrograph permanently, the menu item is disabled until another use of the Catchment command generates another direct runoff hydrograph. However, the Lag and Route affect can be reversed by using the **Hydrograph / Undo** command.

3.3.1 Catchment Aspect Ratio



Current peak flow	15.309	c.m/sec
Total Area	100.000	hectare
Aspect ratio	3.000	
Flow length	2309.4	metre

Fig 3-28 – The catchment aspect ratio is used to estimate the longest flow path

The total catchment area is copied from the last use of the Catchment command. The aspect ratio is used to compute the sum of the dimensions of an equivalent rectangle which is used as an approximate estimate of the longest drainage path in the network draining the catchment. By changing the aspect ratio you can indirectly modify the estimate of the longest flow length.

3.3.2 Longest Flow Length

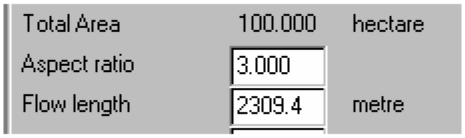

Total Area 100.000 hectare
Aspect ratio 3.000
Flow length 2309.4 metre

Fig 3-29 – The longest flow path can be edited if you have data.

This value is an estimate of the longest drainage path in the network draining the catchment and initially shows a value based on the total area and the aspect ratio of the catchment. You may change this if you can provide a better estimate.

Longer drainage paths result in longer times of concentration which in turn leads to increased lag times and increased attenuation of the flow peak.

Now confirm the time-averaged flow at the outlet.

3.3.3 Average Flow for Lag & Route

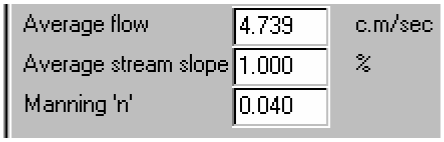

Average flow 4.739 c.m/sec
Average stream slope 1.000 %
Manning 'n' 0.040

Figure 3-30 – Setting the time-averaged flow

The peak flow of the direct runoff hydrograph is known from the Catchment command. This parameter is intended to provide an estimate of the time-averaged flow at the outflow point from the catchment drainage network. The flow is assumed to vary linearly from this value at the outflow point to a negligible flow value at the furthest upstream point in the drainage network.

3.3.4 Average Conduit Slope for Lag & Route

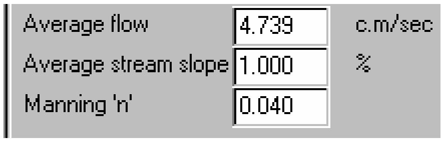
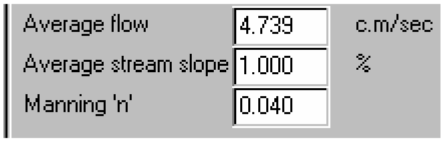

Average flow 4.739 c.m/sec
Average stream slope 1.000 %
Manning 'n' 0.040

Figure 3-31 – Estimate the average conduit slope

You should provide an estimate of the average conduit slope in the longest drainage path. An initial default value of 1% is displayed but this may be too steep for a catchment which is drained predominantly by channels. Flatter slopes result in longer time of concentration which in turn leads to increased lag times and greater reduction of the flow peak.

3.3.5 Conduit Roughness for Lag & Route



Average flow 4.739 c.m/sec

Average stream slope 1.000 %

Manning 'n' 0.040

Figure 3-32 – Setting the conduit roughness

The default values for Manning's "n" are set as follows:

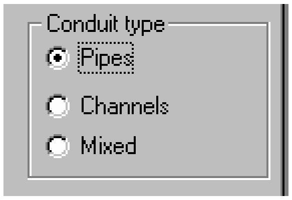
Pipes	n = 0.014
Channels	n = 0.04

The value displayed depends on whether the conduit type option is set as Pipes or Channels. If a mixed type is selected the value is "grayed out". You can change these by selecting either Pipes or Channels as the Conduit Type and then typing in a preferred value in the "Manning's 'n' " data entry box.

For suggested values refer to ‘Manning Roughness for Pipes’, or ‘Channel Parameters’ in Chapter 4 – *Design Options Available*.

The final step before pressing the [Route] button is to select the conduit type which best describes the drainage network.

3.3.6 Lag & Route through Pipes



Conduit type

● Pipes

○ Channels

○ Mixed

Figure 3-33 – Specifying the conduit type

The frame "Conduit Type" contains three option buttons. You can choose to model the catchment assuming the runoff is drained in pipe conduits which have a free surface. MIDUSS assumes the pipes to be 75% full.

If the area is drained mainly through open channels select Channels as the conduit type.

3.3.7 Lag & Route through Channels

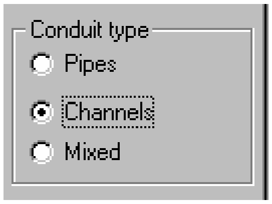


Figure 3-34 – Defining the drainage conduits as channels

The Frame "Conduit Type" contains three option buttons. You can choose to model the catchment assuming the runoff is drained in open channels. MIDUSS assumes that the channel cross-section is triangular with a side-slope of 3H:1V. You can change the assumed side-slope by clicking on the 'Show Options' check box.

If the drainage network contains both pipes and channels select Mixed Conduits.

3.3.8 Lag & Route through Mixed Conduits

For a large catchment area , it is frequently not reasonable to assume that the entire area is drained with only pipes or channels. If you select the "Mixed" option, MIDUSS assumes that the upstream (1-X)% of the drainage length is open channel and the downstream X% is in pipes. The value of " X " is initially set equal to the percentage of impervious area which was defined when the catchment was defined but you can of course change this.

Figure 3-35 – You can modify some of the option values

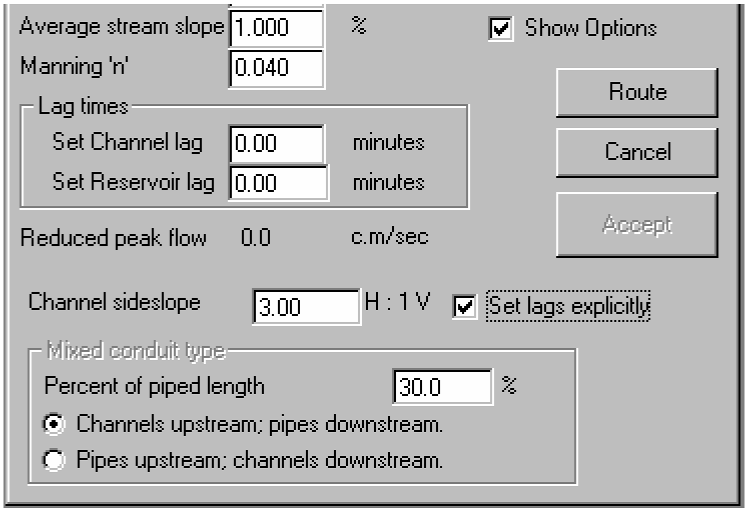
You can change these assumptions by clicking on the 'Show Options' check box.

The percentage of pipe length and whether it is upstream or downstream of the channel section of the flow length can be defined.

If you want to refine the definition of "mixed conduits" adjust the Lag and Route Options

3.3.9 Lag and Route Options

Clicking on the "Show Options" check box opens (or closes) several data entry objects that let you customize the default parameters used in the Lag and Route procedure. These are related mainly to conduit characteristics on which the time of concentration depends.


Average stream slope 1.000 %

Manning 'n' 0.040

Lag times

Set Channel lag 0.00 minutes

Set Reservoir lag 0.00 minutes

Reduced peak flow 0.0 c.m/sec

Channel sideslope 3.00 H : 1 V

☑ Set lags explicitly

☑ Show Options

Mixed conduit type

Percent of piped length 30.0 %

● Channels upstream; pipes downstream.

○ Pipes upstream; channels downstream.

Figure 3-36 – Setting the Lag and Route options

You can change:

- The channel side slope
- The percentage of flow length in pipes (Initially set equal to the percent impervious of the catchment)
- The sequence of pipes and channels - e.g. pipes upstream or downstream

Note that the options related to the Mixed Conduit which are contained within the frame "Mixed Conduit Type" are enabled only when the "Mixed" Conduit Type option is selected.

At any time you can press the [Route] button to see the result of the Lag and Route operation.

3.3.10 The Lag and Route Operation


Reduced peak flow 6.634 c.m/sec

Figure 3-37 – Result of the Lag and Route operation

Pressing the [Route] button causes the reduced peak flow to be displayed as shown, a graphic display of the original Runoff and the modified Runoff is displayed and a tabular display of the modified hydrograph is shown.

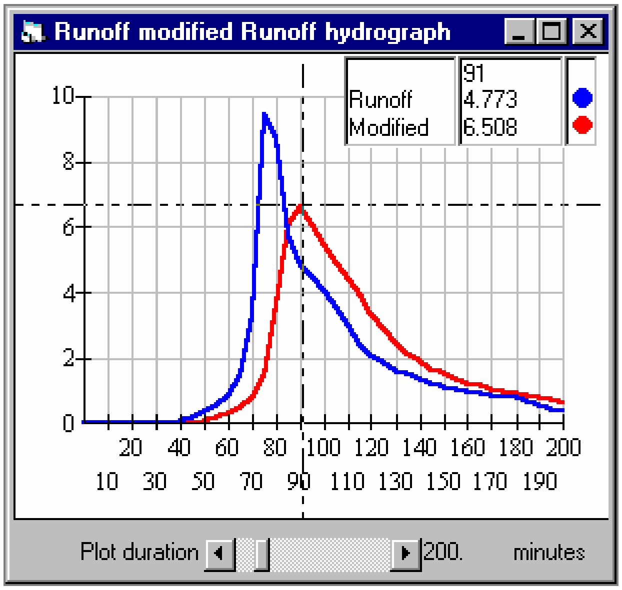
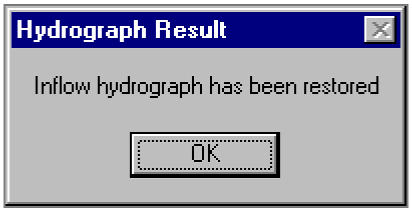


Figure 3-38 – Graphical comparison of the runoff hydrographs

You can experiment by changing some of the parameters to find the sensitivity of the result to the input data.

The Inflow hydrograph is stored in the Backup array to allow the modified Runoff hydrograph to be stored in the Inflow hydrograph array to allow graphical comparison.


Hydrograph Result

Inflow hydrograph has been restored

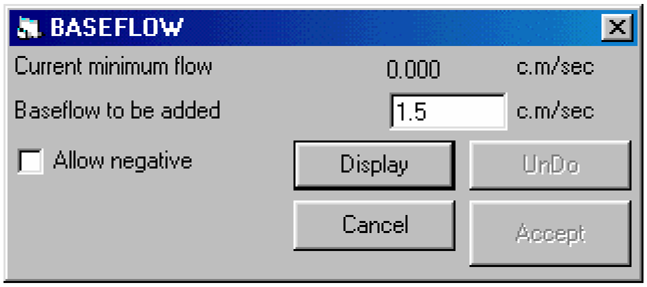
Figure 3-39 – The Inflow is restored when the operation is accepted

Thus when you are satisfied and press the [Accept] button the Runoff hydrograph will be overwritten and you will see the message shown in Figure 3-39 advising that the Inflow hydrograph has been restored.

If you press the [Cancel] button on the main form, both the Runoff and Inflow hydrographs are restored to their original values.

Even after pressing the [Accept] button, the effect of the **Lag and Route** command can be reversed by using the **Hydrograph / Undo** command.

3.4 Baseflow Command



Current minimum flow 0.000 c.m/sec

Baseflow to be added 1.5 c.m/sec

☐ Allow negative

Figure 3-40 – Adding baseflow to the Inflow hydrograph

The direct runoff hydrograph generated by the Catchment command will normally start from zero even if the initial rainfall intensity is finite. This command allows you to add a constant baseflow amount to the inflow hydrograph.

The command is enabled only after the inflow hydrograph has been created by means of the **Hydrograph/Add Runoff** option of the hydrograph menu. The form is shown above and displays only two items of data.

The "Current minimum flow" is the smallest value in the Inflow hydrograph and is reported to indicate if baseflow has been added at a previous stage in the modelling.

You are prompted to enter a constant flow rate which is to be added to every ordinate in the Inflow hydrograph.

This is one of the few cases in MIDUSS in which a negative quantity can be defined. This allows a previously applied baseflow to be removed. Obviously, if a negative value is used it cannot be arithmetically greater than the current minimum flow. If a negative baseflow is used that is numerically greater than the current minimum flow, flows lower than this are reduced to zero. There may be situations such as modeling permeable parking surface in which this may be a useful approximation.

3.5 IUH Hydrograph Command

This command provides a simple way to create an Inflow hydrograph with a user-specified peak flow and time to peak (or duration) with a shape defined by a file containing the coordinates of a pre-defined Instantaneous Unit Hydrograph.

A file containing the SCS IUH curve is included with MIDUSS and you can easily prepare similar files to describe other IUH shapes. The format is a simple text file holding:

(i) the number of coordinate pairs

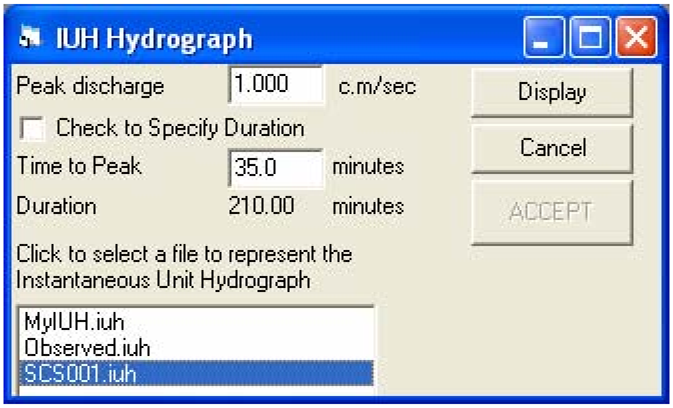
(ii) A series of pairs of values defining the relative time and relative flow values.

Fragments of a typical file are shown in Figure 3-41.

```
32
0,     0
0.1,   0.015
0.2,   0.075
:
0.94,   0.99
1,     1
1.05,   0.99
1.1,   0.98
:
5,     0.004
5.5,   0.002
6,     0
```

Figure 3-41 – Fragments of an IUH File

When the **Hydrology / IUH Hydrograph** menu item is selected the form is displayed with initial default values as shown in Figure 3-42. The times are selected based on the timestep and maximum storm duration and the ratio of Time to Peak and Duration implied by the coordinates in the selected IUH file.



Peak discharge 1.000 c.m/sec

☐ Check to Specify Duration

Time to Peak 35.0 minutes

Duration 210.00 minutes

Click to select a file to represent the Instantaneous Unit Hydrograph

MyIUH.iuh
Observed.iuh
SCS001.iuh

Figure 3-42 – Initial values in the IUH Hydrograph form

You can enter a desired peak flow value and also specify either the time to peak in minutes or the duration in minutes. The former is more usual.

The [Display] command button is disabled until you select a particular IUH file from the list displayed on the form. You may find it useful to select the IUH file and press [Display] to both graph and tabular display of a trial hydrograph. You can then enter trial values for the required data and press [Display] to update the displayed information.

When satisfied, you can press the [Accept] button and the specified hydrograph is entered in the Runoff hydrograph. The peak runoff value in the summary peak flows grid is also updated.

Notes:

Chapter 4 Design Options Available

Figure 4-1 - The Design menu

4.1 Scope of Design

One of the most valuable features of MIDUSS is the ability to design conveyance and storage elements in the drainage network. Design of these elements is facilitated by one of the commands Pipe, Channel, Pond, Trench, Culvert, Cascade or Diversion. In addition, an eighth command in the Design menu called Route carries out a flood routing analysis of the current inflow hydrograph through the most recently designed pipe or channel. Reservoir routing is an integral part of the Pond and other commands.

Pipe	Circular pipes running full (*i.e.* surcharged) or part full under conditions of uniform flow.
Channel	Open channels of simple or complex cross-section under conditions of uniform flow. Simple cross-sections can be trapezoidal, triangular or rectangular. More complex, arbitrary cross-sections can be drawn as a series of straight lines joining points the coordinates of which can be edited during the design process.
Route	The Route command carries out a flood routing operation of the current Inflow hydrograph in the most recently designed pipe or channel.
Pond	Detention ponds with arbitrary characteristics of discharge as a function of depth and storage volume as a function of depth. Outflow control characteristics can be described by a combination of weirs and orifices and stage-storage characteristics can be automatically generated for a number of special shapes such as multi-layer ponds, oversized sewers or "super-pipes" or, pyramidal shapes in the vicinity of catchbasins. Rooftop storage can also be simulated
Diversion	Diversion structures which split the inflow hydrograph into outflow and diverted hydrographs.
Trench	Exfiltration trenches of finite slope and general trapezoidal cross-section with one or more solid or perforated pipes can be designed for different soil and groundwater conditions.
Culvert	A culvert to convey a watercourse through an embankment can be designed for either steady (constant) inflow or for an inflow hydrograph. Different barrel cross-sections are available.

Cascade	An underground storage system comprising two elements at equal or different elevations can be designed to attenuate peak flow. Different conduit cross-sections can be used.

Pipe and channel flood routing is carried out by a kinematic wave method similar to the Muskingum-Cunge technique, involving the automatic calculation of the spatial weighting parameter by a procedure developed by Smith (1980). Reservoir routing is carried out by the Storage Indication method which is discussed in most hydrology texts but with an additional check for numerical stability in the neighbourhood of highly nonlinear storage - discharge relationships.

When the Diversion command is used, the diverted hydrograph is written to a file which can subsequently be read in to the inflow hydrograph array by means of the **File / Load file / Hydrograph** command to allow design of the diverted branch to be continued.

Further details of the design methods are presented in the individual discussions that follow. Some background theory on the hydraulics used in these design options is contained in Chapter 8 - *Theory of Hydraulics*

Before proceeding to the design options, take a moment to review the procedure for updating the Inflow hydrograph

4.2 Updating the Inflow Hydrograph

MIDUSS treats each pipe, channel, pond, diversion structure, culvert, cascade or trench as a link in the drainage system. These links form a tree-shaped network joining a set of nodes (points or junctions) at which overland flow may enter the drainage system.

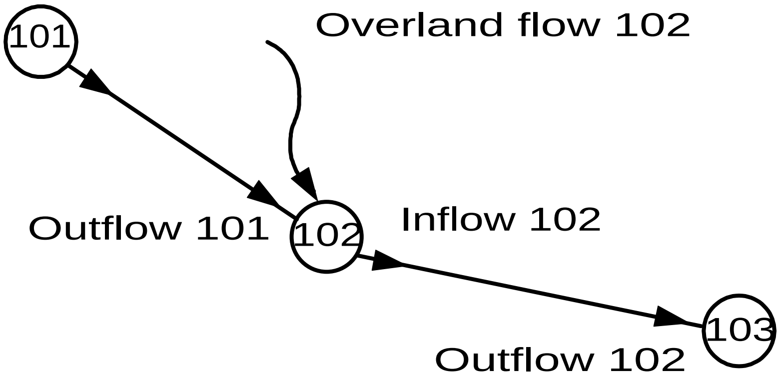


Figure 4-2 – A node and link numbering convention

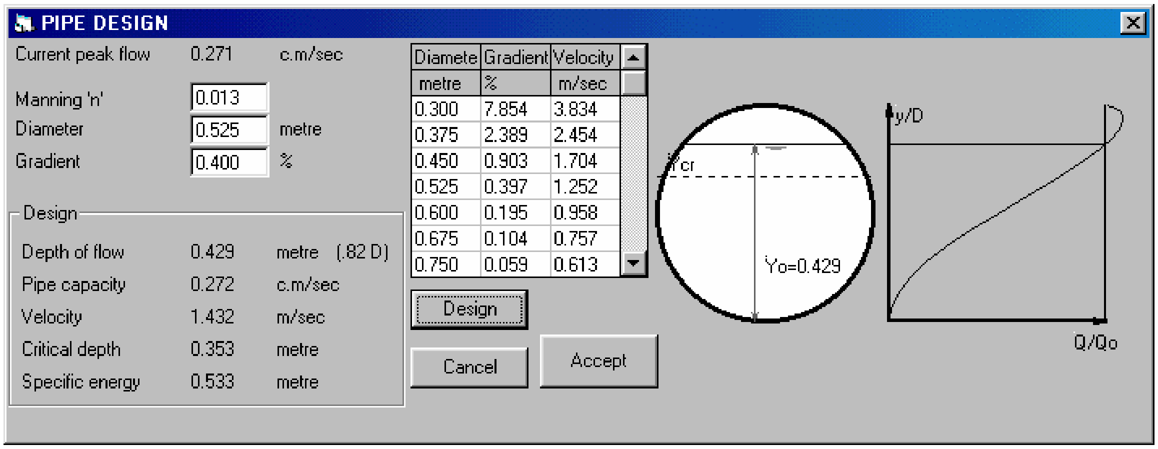
Figure 4.2 illustrates a very simple system comprising 3 nodes and 2 links. MIDUSS uses the convention that links are assigned the same number as the node at its upstream end. Thus, in the figure link #102 joins node 102 to node 103.

Before the link downstream of a node can be designed, any overland flow hydrograph entering at that point must be added to the current inflow. Again with reference to Figure 4.2, the hydrograph labeled 'Overland flow 102' must be added to 'Outflow 101' to make 'Inflow 102' before link #102 can be designed. This is done by means of the **Hydrograph/Add Runoff** command.

After completing the **Hydrology/Catchment** command described in Chapter 3 *Hydrology Used in MIDUSS*, the next logical step is the **Hydrograph/Add Runoff** command. If you use the **Options/Show Next logical menu item**, MIDUSS opens up the **Hydrology** menu and places the mouse pointer at the **Add Runoff** command.

If you try to design a pipe, channel *etc* without using the **Add Runoff** command MIDUSS will warn you that you may be making a mistake. However, this is only a warning and if you persist in trying to use one of the design options described in this chapter, MIDUSS will bow to your superior knowledge and let you have your way.

4.3 Pipe Design



PIPE DESIGN

Current peak flow 0.271 c.m/sec

Manning 'n' 0.013

Diameter 0.525 metre

Gradient 0.400 %

Design

Depth of flow 0.429 metre (.82 D)

Pipe capacity 0.272 c.m/sec

Velocity 1.432 m/sec

Critical depth 0.353 metre

Specific energy 0.533 metre

Diamete	Gradient	Velocity
metre	%	m/sec
0.300	7.854	3.834
0.375	2.389	2.454
0.450	0.903	1.704
0.525	0.397	1.252
0.600	0.195	0.958
0.675	0.104	0.757
0.750	0.059	0.613

Figure 4-3 – The pipe design form

This is one of the seven commands (Pipe, Channel, Pond, Trench, Diversion, Culvert and Cascade) used to design a link in the drainage network. It helps you to select the diameter (metres or feet) and gradient (%) of a circular pipe to carry the peak flow of the current inflow hydrograph. The command involves four steps.

1. Choose a value for Manning's ‘n’ Accept or modify the current default values for Manning's 'n'.

2. Review feasible diameter - gradient values Review a set of diameter-gradient values using a range of commercially available pipe diameters which will carry the peak flow with the pipe flowing full-bore.

3. Specify diameter and gradient Select design values for the proposed pipe diameter and gradient and compute the depth and average velocity of part-full uniform flow in the designed pipe.

4. Accept the design Accept the design or modify any one of the design parameters specified in step (1) or modify the design entered in step (3).

If a surcharged condition is found, MIDUSS reports the slope of the hydraulic grade line instead of the free surface depth. See the item on Surcharged Pipe Design later in this chapter for more details. Refer also to the Pipe Design Log which may serve as a useful reminder if you are interrupted during the MIDUSS session.

You can use the Pipe command to design a pipe even if an Inflow hydrograph has not been created. See Pipe Design for Steady Flow for details.

4.3.1 Manning roughness for Pipes

Current peak flow 0.950 c.m/sec

Manning 'n' 0.013

Figure 4-4 – Specifying the Manning ‘n’ roughness value.

MIDUSS uses the Manning "n" value to define roughness. If you prefer to use a roughness height to define roughness you can use the **Tools / Roughness Height** menu to obtain an equivalent value of Manning’s ‘n’.

Values depend on the type and condition of the pipe material and suggested values can be found in many texts. The table below provides a few typical values.

Description	**"n"**
Metal pipe - spun concrete lining	0.007
Wrought iron	0.008
Smooth pre-cast well jointed concrete	0.009
Uncoated cast iron; well-aligned glazed vitrified clay	0.01
Spun concrete	0.011
Monolithic concrete; rough pre-cast concrete; butt-jointed drain tile; slimed sewers	0.013
Pre-cast pipes with mortar squeeze at joints; well pointed brickwork	0.015
Old brickwork; foul sewers with grease, lime encrusted or sludge	0.019

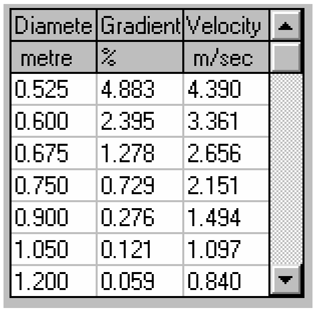
Manning "n" for corrugated pipes is dependent on diameter, especially for smaller diameters. A rough guide is given by:

$n = 0.014 + 0.02$ (D in metres) for $D < 0.4$ m

$n = 0.020 + 0.002$ (D in metres) for $D > 0.4$ m

Once the roughness is set you can review Possible Pipe Designs

4.3.2 Possible Pipe Designs



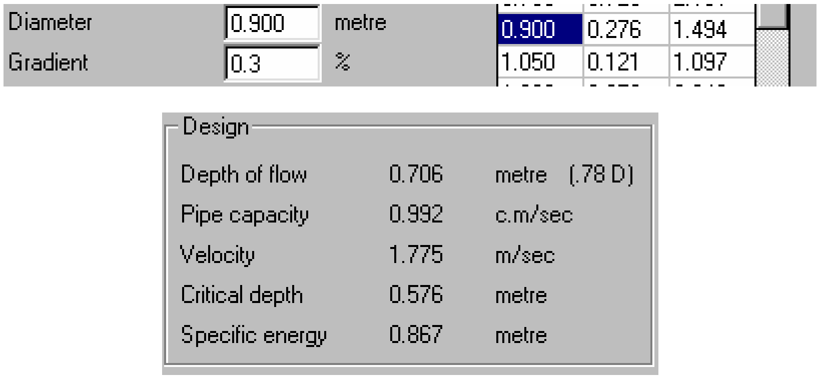
Diamete	Gradient	Velocity
metre	%	m/sec
0.525	4.883	4.390
0.600	2.395	3.361
0.675	1.278	2.656
0.750	0.729	2.151
0.900	0.276	1.494
1.050	0.121	1.097
1.200	0.059	0.840

Figure 4-5 – MIDUSS displays a table of feasible designs.

For the currently defined peak flow and roughness MIDUSS displays a small table showing feasible values of diameter, gradient (%) and average velocity. This provides a guide for a non-surcharged design which will produce an acceptable average velocity.

4.3.3 A Trial Pipe Design

You can enter trial values for diameter and gradient in two ways. You can type desired values in the appropriate text boxes as you did for Manning "n". Alternatively, if you double-click on one of the rows of the table the values will be copied but the gradient will be rounded up to the nearest 0.1%. In Figure 4-6 the gradient has been rounded up from 0.274% to 0.3%.



Diameter 0.900 metre

Gradient 0.3 %

0.900	0.276	1.494
1.050	0.121	1.097

Design

Depth of flow	0.706	metre (.78 D)
Pipe capacity	0.992	c.m/sec
Velocity	1.775	m/sec
Critical depth	0.576	metre
Specific energy	0.867	metre

Figure 4-6 – Copying a trial design (top) and viewing the results

Once the diameter and gradient have been set, press the [Design] button to produce the results as shown in Figure 4-6. MIDUSS reports the depth of uniform flow, the pipe-full capacity, the average velocity, the critical depth and the specific energy. In addition, the uniform flow depth is expressed as a fraction of the diameter. The results in Figure 4-6 (bottom) show that the velocity is rather high. If the normal depth is less than critical the flow will be supercritical with the probable result of a hydraulic jump at some point downstream.

If the pipe is surcharged the design information reports the hydraulic gradient in place of the depth of uniform flow,

You can experiment with different values for diameter, gradient or roughness until you are satisfied with the design.

4.3.4 Accepting the Pipe Design

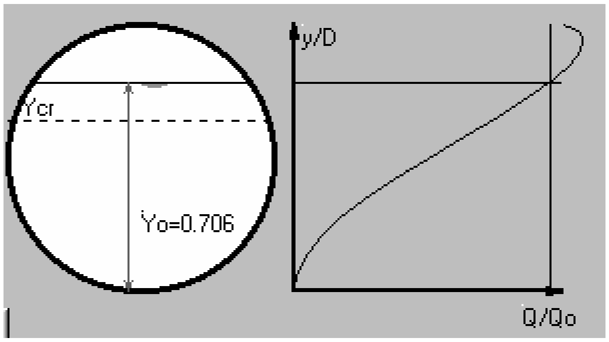


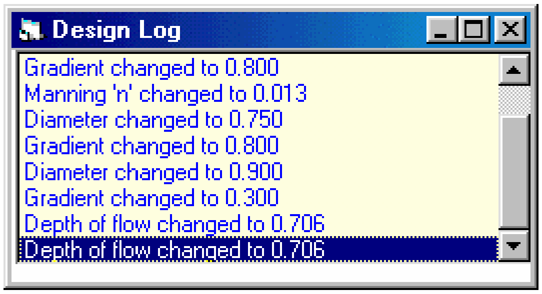
Figure 4-7 – A graphical depiction of the pipe design.

As well as numerical results, MIDUSS displays a simple sketch showing the normal flow depth in the pipe, the critical depth and the corresponding curve of relative discharge (i.e. flow /full-pipe capacity).

If you are satisfied with the design, press the [Accept] button to save the design and close the window.

Your design is saved and also your various trials are shown on the Pipe Design Log.

4.3.5 The Pipe Design Log



```
Gradient changed to 0.800
Manning 'n' changed to 0.013
Diameter changed to 0.750
Gradient changed to 0.800
Diameter changed to 0.900
Gradient changed to 0.300
Depth of flow changed to 0.706
Depth of flow changed to 0.706
```

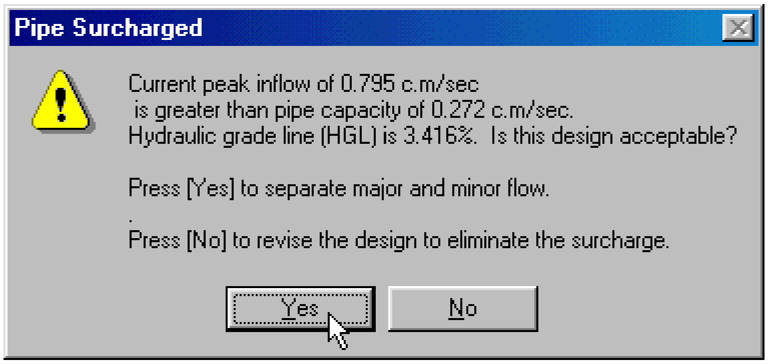
Figure 4-8 - A typical design log for a pipe design

If you experiment with a number of alternative designs you may find it useful to refer to a log which is maintained by MIDUSS. This log is usually located in the top right corner of the screen and appears for each of the design functions. A design log is of more value for facilities (such as the Pond design) in which many parameters are involved. However, it is included for Pipe design for completeness. The contents of the log file can be accumulated and saved or printed out at any time.

4.3.6 Surcharged Pipe Design

If the diameter or gradient is insufficient to convey the peak flow the pipe will be surcharged. In this case you will see the value of the pressurized hydraulic grade line instead of the uniform flow depth. Also the critical depth is not feasible and is shown as zero. The pipe capacity will be seen to be less than the peak inflow reported at the top of the window.

You will also see a warning message as shown in Figure 4-9 below. In some situations you may want to accept the surcharged condition. Such a case might be when the pipe is designed for a 5-year storm but you want to see the impact of a more severe (say 100-year) event. In this case you should press the [OK] button and then accept this design by pressing the [Accept] button. You can then use the Design/Diversion command to separate the minor and major flow. If surcharge is not acceptable press the [OK] button and then change the design until an acceptable design is obtained.



Current peak inflow of 0.795 c.m/sec
is greater than pipe capacity of 0.272 c.m/sec.
Hydraulic grade line (HGL) is 3.416%. Is this design acceptable?

Press [Yes] to separate major and minor flow.
.
Press [No] to revise the design to eliminate the surcharge.

Figure 4-9 – Warning following a surcharged pipe design.

4.3.7 Pipe Design for Steady Flow

The Design/Pipe command is one of three menu items (Pipe, Channel & Culvert) that are enabled as soon as the system of units is defined. This lets you use the Pipe command to design a pipe for any flow that you specify.

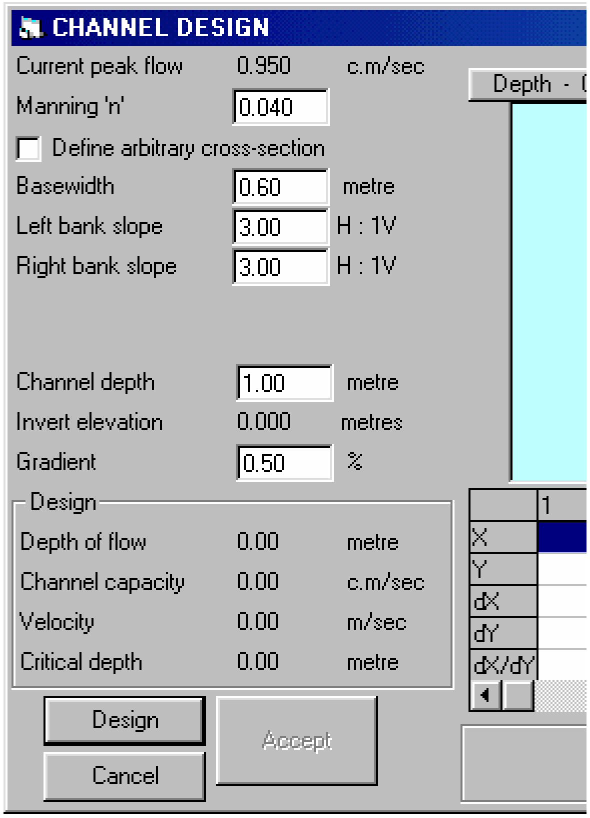
If no Inflow hydrograph has been created the Pipe window opens with a text box against the "Current peak flow" which lets you specify a flow rate instead of simply displaying the current peak inflow. You can then design a pipe without having to do any hydrological simulation.

Current peak flow 0.000 c.m/sec

Figure 4-10 – If no inflow hydrograph exists you can specify the design flow.

The only difference is that a text box is opened against the prompt "Current peak flow". Type in a value and the table of feasible designs will be filled. You can then proceed as usual.

4.4 Channel Design



Current peak flow 0.950 c.m/sec
Manning 'n' 0.040
☐ Define arbitrary cross-section
Basewidth 0.60 metre
Left bank slope 3.00 H : 1V
Right bank slope 3.00 H : 1V
Channel depth 1.00 metre
Invert elevation 0.000 metres
Gradient 0.50 %

Design
Depth of flow 0.00 metre
Channel capacity 0.00 c.m/sec
Velocity 0.00 m/sec
Critical depth 0.00 metre

Figure 4-11 – Data required for a simple channel design

This is one of the seven commands (Pipe, Channel, Pond, Trench, Diversion, Culvert and Cascade) used to design a link in the drainage network. The Channel design can be used to design two types of cross-section - a simple generalized trapezoidal shape, or a more complex, arbitrary cross-section.

Trapezoidal sections. In this mode the command is quite similar to the **Design/Pipe** command and involves four steps.

1. Set values for roughness, base width and side slope.
2. Review a table of Feasible Designs
3. Select values for the overall depth and gradient
4. Accept or Modify the design

Complex sections. The procedure is generally the same but the channel shape is defined by drawing a set of straight lines between points the coordinates of which can be edited to refine the design. Refer to the topic on Switching to a Complex Section for more details.

4.4.1 Set Parameters for the Trapezoidal Channel

CHANNEL DESIGN

Current peak flow 0.950 c.m/sec

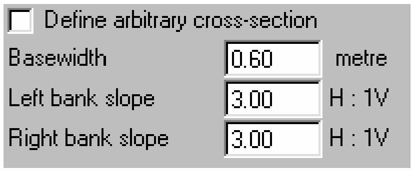
Manning 'n' 0.040

Figure 4-12 – Setting the Manning ‘n’ value for the channel

A value for Manning "n" must be entered in order for the table of feasible designs to be shown. If you prefer to use a roughness height to define roughness you can use the **Tools/Roughness Height** menu to obtain an equivalent value of Manning’s ‘n’.

The initial default value as shown here is 0.04. The table below shows some suggested values for different types and conditions of channel. Notice that the roughness for channel flow is very different from that for overland flow.

Description	**"n"**
Concrete lined, screened and smoothed	0.014
Gunite concrete, not smoothed with sandy deposits	0.018
Irrigation canal in hard-packed smooth sand	0.020
Canal excavated in silty clay	0.024
Channel with cobble stone bottom	0.028
Natural channel with fairly regular cross-section	0.035
Natural channel, irregular section, grass slopes	0.040
Dredged channel, irregular side slopes, grass and weeds	0.050
Irregular channel with dense growth, little foliage	0.080
Irregular channel with dense growth, with much foliage and vegetation	0.110



☐ Define arbitrary cross-section

Basewidth 0.60 metre

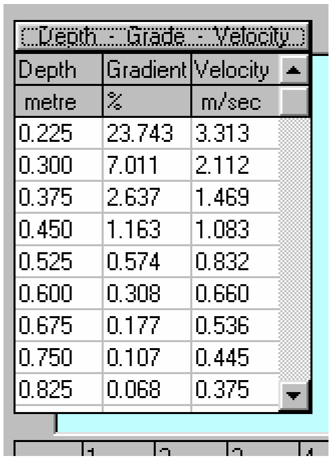
Left bank slope 3.00 H : 1V

Right bank slope 3.00 H : 1V

Figure 4_13 - Defining a trapezoidal cross-section.

The base width and side slopes are self-explanatory. By selecting appropriate values, trapezoidal, rectangular or triangular sections can be defined. You will find that a trapezoidal section can often be a good approximation for a natural channel. Negative (i.e. overhanging) side slopes are not allowed with the trapezoidal cross-section.

4.4.2 Review Feasible Depths and Gradients

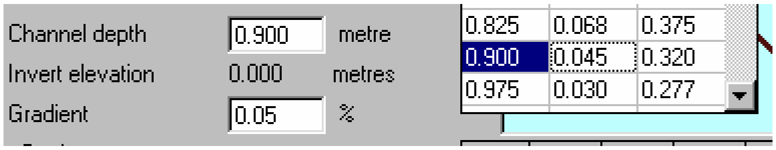


Depth metre	Gradient %	Velocity m/sec
0.225	23.743	3.313
0.300	7.011	2.112
0.375	2.637	1.469
0.450	1.163	1.083
0.525	0.574	0.832
0.600	0.308	0.660
0.675	0.177	0.536
0.750	0.107	0.445
0.825	0.068	0.375

Figure 4-14 – MIDUSS displays a table of feasible designs

Pressing the large "Depth - Grade - Velocity" button causes a table of feasible designs to be displayed or hidden. The table depends on a flow rate and roughness being defined. By double clicking on a row of the table the depth and gradient are copied to the text boxes for channel depth and gradient. However, these designs do not provide much allowance for freeboard and you will usually need to either increase the overall depth and/or steepen the gradient.

4.4.3 Select Channel Depth and Gradient



Channel depth 0.900 metre

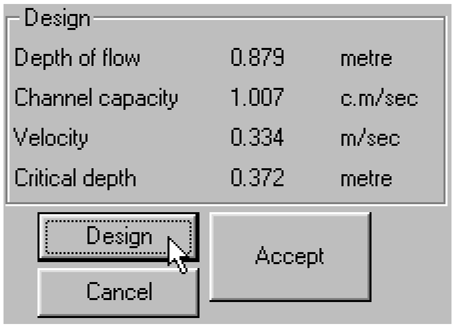
Invert elevation 0.000 metres

Gradient 0.05 %

0.825	0.068	0.375
0.900	0.045	0.320
0.975	0.030	0.277

Figure 4-15 – Entering a trial design

Enter trial values for overall depth and invert gradient in the text boxes shown in Figure 4-15.

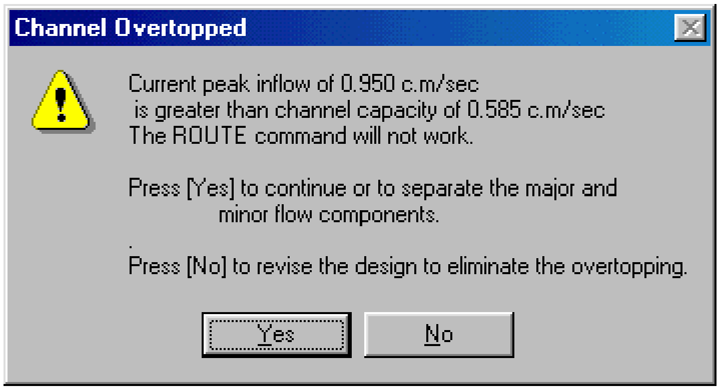

Design

Depth of flow	0.879	metre
Channel capacity	1.007	c.m/sec
Velocity	0.334	m/sec
Critical depth	0.372	metre

Figure 4-16 – Results of a trial design.

Then by clicking on the [Design] button as shown, the results of the uniform flow analysis are displayed. In addition to the normal depth and velocity, the channel capacity and the critical depth are displayed.

If you define a channel with insufficient capacity you will receive a warning as shown below. In such cases MIDUSS completes the design by calculating the depth assuming that the sides are extended at the defined side slope. MIDUSS will not let you "Accept" a design like this as the Route command cannot be used.


Channel Overtopped

Current peak inflow of 0.950 c.m/sec
is greater than channel capacity of 0.585 c.m/sec
The ROUTE command will not work.

Press [Yes] to continue or to separate the major and
minor flow components.

Press [No] to revise the design to eliminate the overtopping.

Figure 4-17 – Warning for an overtopped channel.

When satisfied you can Accept the Channel Design

4.4.4 Accept the Channel Design

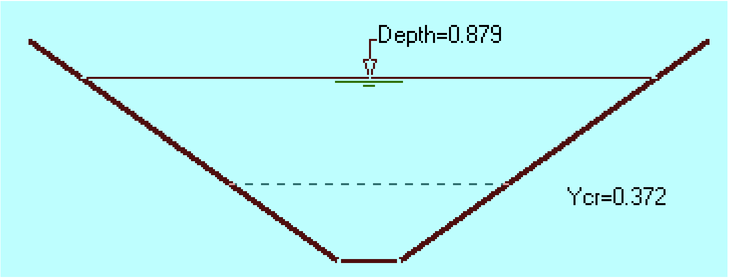
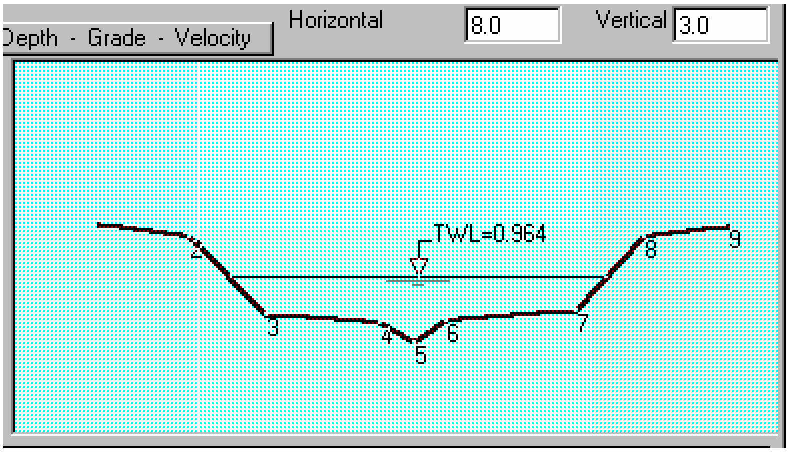


Figure 4-18 – Results of a trapezoidal channel design

In addition to the results displayed in the 'Design' frame', MIDUSS plots the cross-section and computed water surface in the graph window on the right of the form. For a trapezoidal section the water surface elevation is plotted assuming the channel invert to be zero.



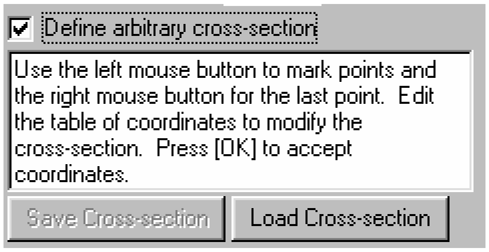
Horizontal 8.0

Vertical 3.0

Figure 4-19 – Results of a complex cross-section channel design

For the complex section the water surface is plotted to the same vertical datum used by the coordinate system when you first sketched the cross-section.

4.4.5 Switching to Complex Section



☑ Define arbitrary cross-section

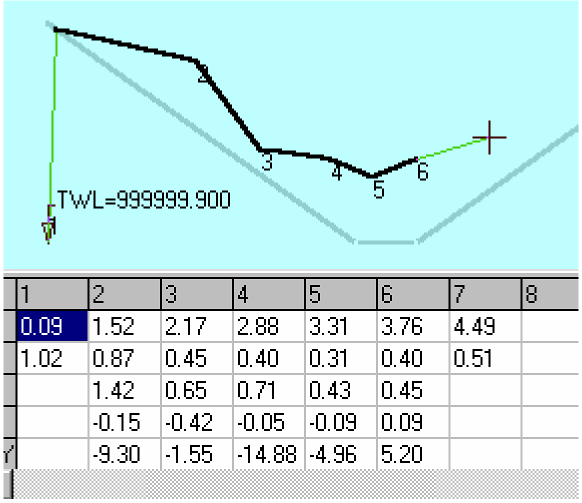
Use the left mouse button to mark points and the right mouse button for the last point. Edit the table of coordinates to modify the cross-section. Press [OK] to accept coordinates.

Figure 4-20 – Selecting the complex cross-section option

To design a complex channel cross-section the first step is to click on the check box labeled "Define arbitrary cross-section". The space previously used for the base width and side slopes of the trapezoidal section is covered by an information window reminding you how to draw the cross-section

Now you can start Drawing a Complex Cross-section.

4.4.6 Drawing a Complex Channel Section



	1	2	3	4	5	6	7	8
	0.09	1.52	2.17	2.88	3.31	3.76	4.49	
	1.02	0.87	0.45	0.40	0.31	0.40	0.51	
		1.42	0.65	0.71	0.43	0.45		
		-0.15	-0.42	-0.05	-0.09	0.09		
Y		-9.30	-1.55	-14.88	-4.96	5.20		

Figure 4-21 – Drawing the channel cross-section

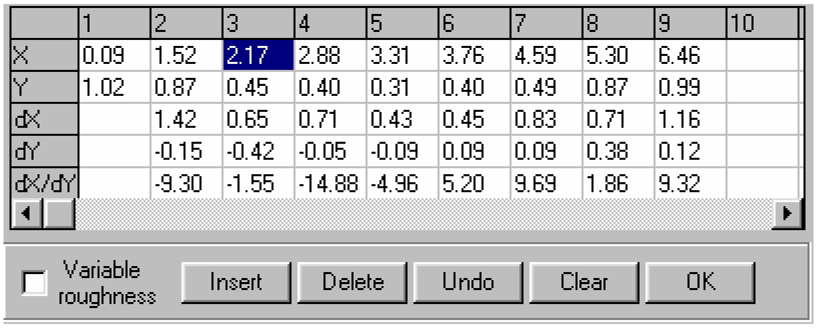
Once you have selected the "arbitrary cross-section" option the graph drawing window is enabled and you will see that the mouse pointer changes to a small cross-hair when it is over the window. The origin of the graph window is near the lower left corner with coordinates (0.0, 0.0). The window coordinates extend from –10% to +90% of the defined scale range. You can select a series of points by clicking with the primary **(left)** mouse button for all but the last point. The figure shows the first 3 points with the mouse pointer about to set point 4. The X and Y coordinates of the mouse pointer are shown in the table.

It is important that **you work from left to right**, although overhanging banks can be defined. With the second and subsequent clicks a straight line is plotted from the previous point. When you reach the last point click on the secondary **(right)** mouse button to signal the end of the section.

If the X- and Y-scales are not correct for the size of channel you want to draw you can change these at any time (except while drawing the cross-section) by typing in the width and height you want the graph window to represent.

You may notice an initial line drawn from the origin to point 1. This will be removed in subsequent displays.

4.4.7 Coordinates of the Complex Section



	1	2	3	4	5	6	7	8	9	10
X	0.09	1.52	2.17	2.88	3.31	3.76	4.59	5.30	6.46	
Y	1.02	0.87	0.45	0.40	0.31	0.40	0.49	0.87	0.99	
dX		1.42	0.65	0.71	0.43	0.45	0.83	0.71	1.16	
dY		-0.15	-0.42	-0.05	-0.09	0.09	0.09	0.38	0.12	
dX/dY		-9.30	-1.55	-14.88	-4.96	5.20	9.69	1.86	9.32	

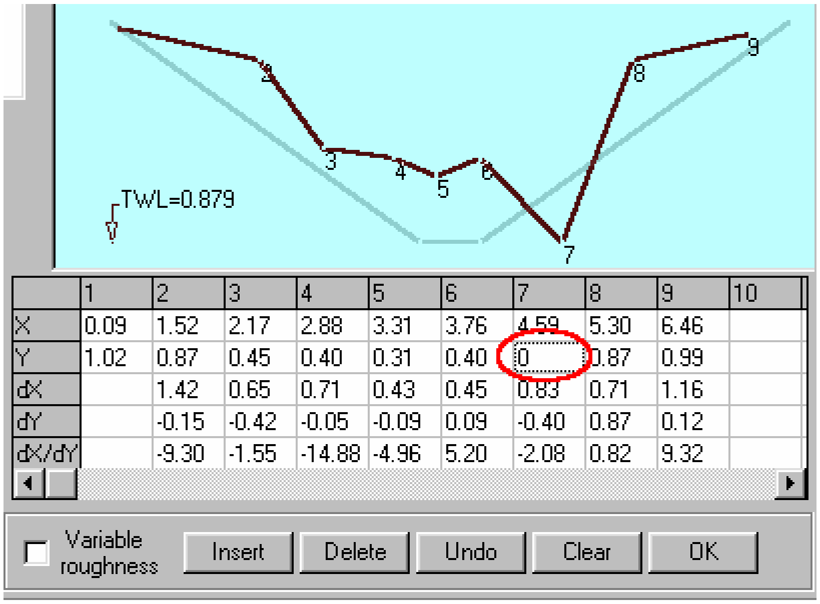
Figure 4-22 – The coordinates of a complex cross-section can be edited

As each point is defined the coordinates are displayed in the table below the graph window. As you move the mouse pointer the X and Y coordinates change to indicate the current position and are fixed when you press the mouse button.

For the second and subsequent points the X and Y coordinates are supplemented by the incremental change dX and dY for the preceding line segment and also the dX/dY slope of the line (i.e. dX/dY=0.0 means horizontal and dX/dY=999 means vertical). These values can be used to adjust the shape of the section without having to work out specific X and Y values.

Unless you have been very careful, it is likely that the cross-section is not exactly what you want, or you may need to modify it for the purpose of your design. You therefore need to Edit the Coordinates.

4.4.8 Editing the Coordinates

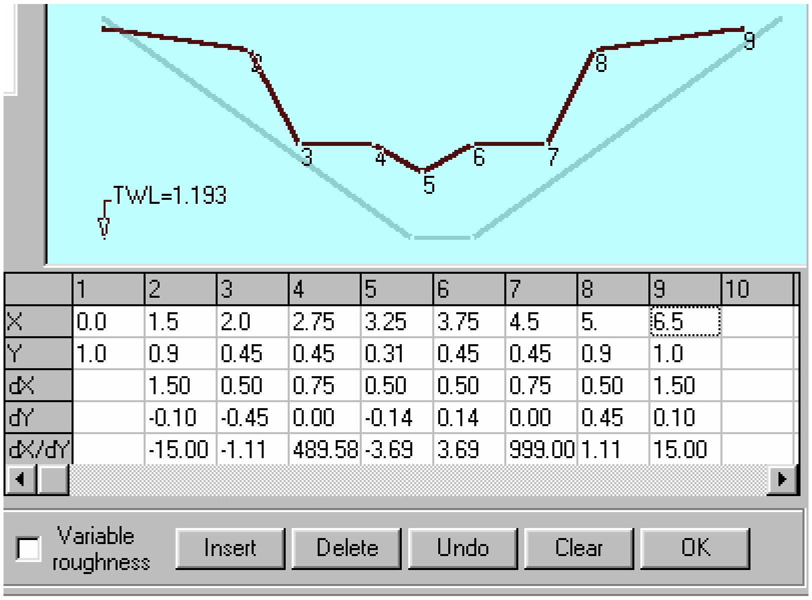


	1	2	3	4	5	6	7	8	9	10
X	0.09	1.52	2.17	2.88	3.31	3.76	4.59	5.30	6.46	
Y	1.02	0.87	0.45	0.40	0.31	0.40	0	0.87	0.99	
dX		1.42	0.65	0.71	0.43	0.45	0.83	0.71	1.16	
dY		-0.15	-0.42	-0.05	-0.09	0.09	-0.40	0.87	0.12	
dX/dY		-9.30	-1.55	-14.88	-4.96	5.20	-2.08	0.82	9.32	

Figure 4-23 – Editing the elevation of point #7

Figure 4-23 shows the effect of editing the coordinates. Clicking on the cell for Y7 makes it the active cell. Alternatively, you can move around the grid by pressing the arrow keys. To change the value in the cell you can either re-type the value or position the cursor and use the Back Space key to delete and replace a character. In the figure, the user has deleted the '49' from the Y7 value making it zero. This is reflected in the plot of the section. Now by typing '45' the value of Y7 is made the same as that for Y3 and the plot of the section is restored..

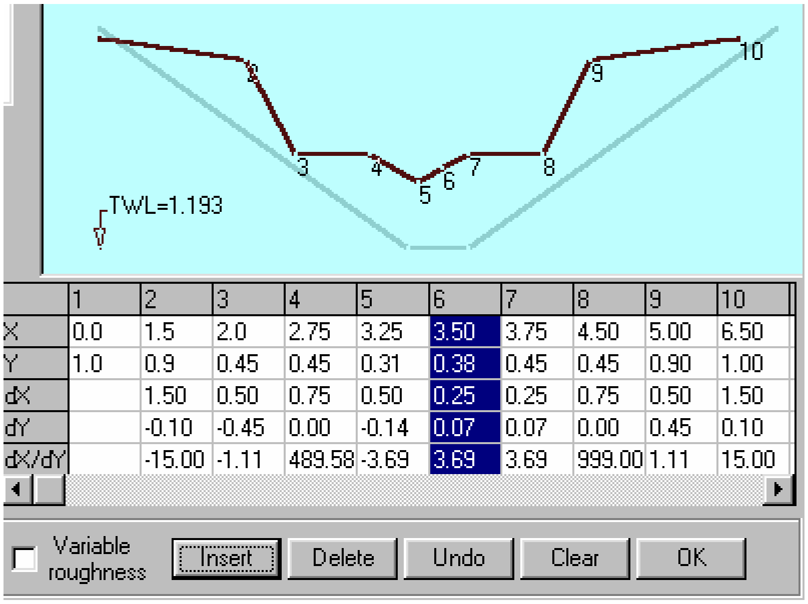
Fig 4-23a below shows the coordinates after tidying up.



	1	2	3	4	5	6	7	8	9	10
X	0.0	1.5	2.0	2.75	3.25	3.75	4.5	5.	6.5	
Y	1.0	0.9	0.45	0.45	0.31	0.45	0.45	0.9	1.0	
dX		1.50	0.50	0.75	0.50	0.50	0.75	0.50	1.50	
dY		-0.10	-0.45	0.00	-0.14	0.14	0.00	0.45	0.10	
dX/dY		-15.00	-1.11	489.58	-3.69	3.69	999.00	1.11	15.00	

Figure 4-23a – Tidying up the cross-section.

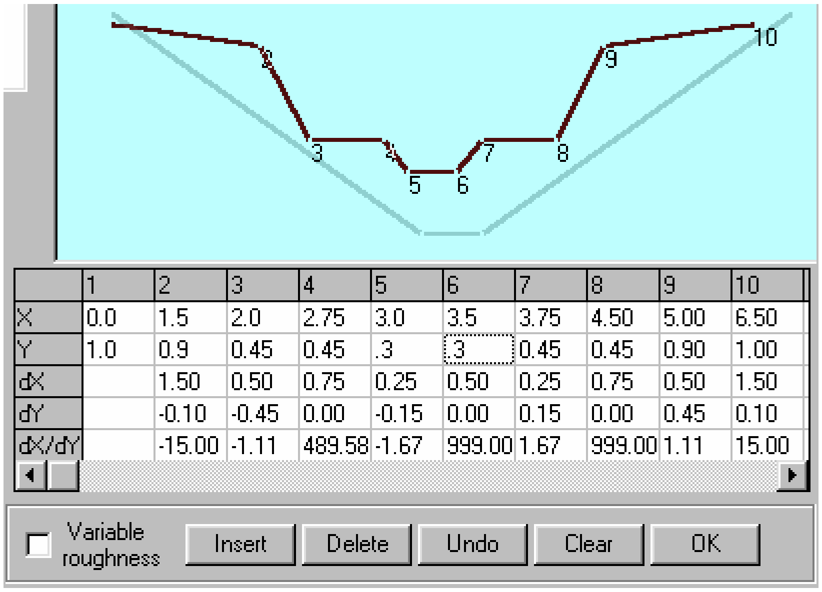
You can also insert or delete points on the cross section. The [Insert] button will add a column (i.e. new point) after the cell where the editing is focused. Fig 4-23b shows an example of the insertion of point 6. All the original points are moved to the right and renumbered. The new point is given data that interpolates the point before and after the insertion. In this example point 6 (the new point) is an average of the data of point 5 and point 7.



	1	2	3	4	5	6	7	8	9	10
X	0.0	1.5	2.0	2.75	3.25	3.50	3.75	4.50	5.00	6.50
Y	1.0	0.9	0.45	0.45	0.31	0.38	0.45	0.45	0.90	1.00
dX		1.50	0.50	0.75	0.50	0.25	0.25	0.75	0.50	1.50
dY		-0.10	-0.45	0.00	-0.14	0.07	0.07	0.00	0.45	0.10
dX/dY		-15.00	-1.11	489.58	-3.69	3.69	3.69	999.00	1.11	15.00

Figure 4-23b – Inserting a data point.

Fig 4.23c shows the new point 6 after editing the coordinates table.



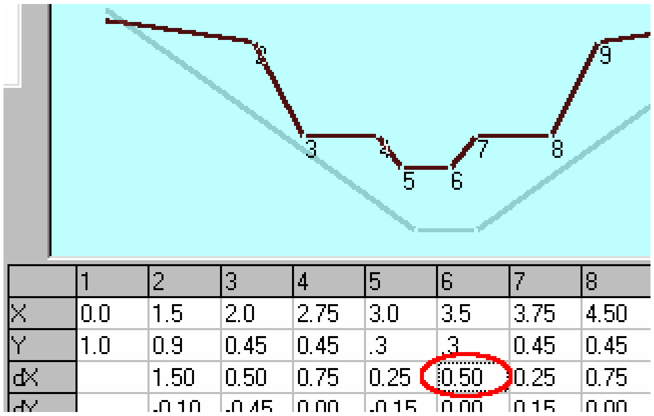
	1	2	3	4	5	6	7	8	9	10
X	0.0	1.5	2.0	2.75	3.0	3.5	3.75	4.50	5.00	6.50
Y	1.0	0.9	0.45	0.45	.3	.3	0.45	0.45	0.90	1.00
dX		1.50	0.50	0.75	0.25	0.50	0.25	0.75	0.50	1.50
dY		-0.10	-0.45	0.00	-0.15	0.00	0.15	0.00	0.45	0.10
dX/dY		-15.00	-1.11	489.58	-1.67	999.00	1.67	999.00	1.11	15.00

Figure 4-23c – Adjusting the new data point

You may need to increase the width of the channel and you can do this by changing the dX value for a point that is simply the width of the preceding line segment. If this makes the section too wide for the graph window you can fix this by clicking [OK] to automatically adjust the scale factors.

This editing ability is particularly useful if you need to Widen a Channel cross-section.

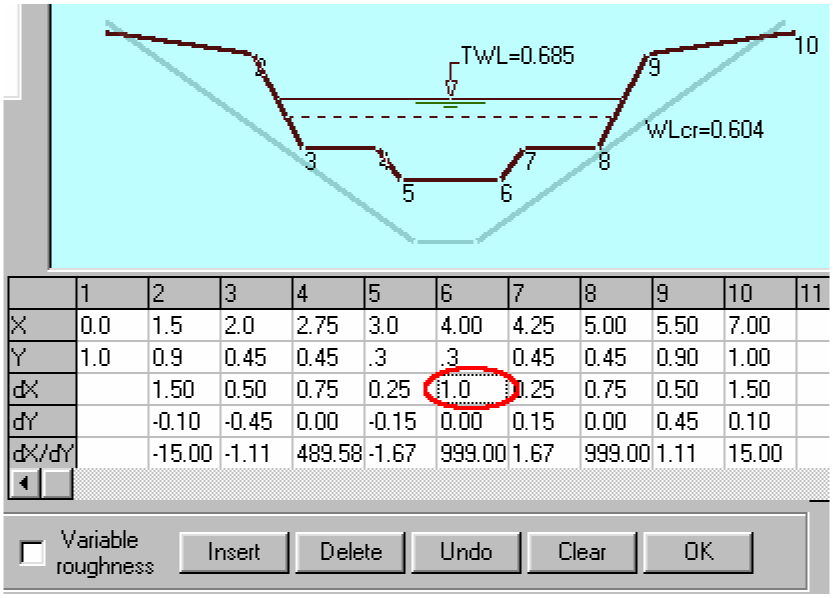
4.4.9 Widening the Channel



	1	2	3	4	5	6	7	8
X	0.0	1.5	2.0	2.75	3.0	3.5	3.75	4.50
Y	1.0	0.9	0.45	0.45	.3	.3	0.45	0.45
dX		1.50	0.50	0.75	0.25	0.50	0.25	0.75
dY		-0.10	-0.45	0.00	-0.15	0.00	0.15	0.00

Figure 4-24 – Preparing to widen the low-flow channel

After defining a cross-section you may find you need to widen the low flow channel. You can do this easily by editing the 'dX' value for the appropriate line segment. The two figures show an increase in width of the line segment (5)-(6) from 0.5 to 1.0 as shown in the cell outlined in red. The 'X' values of all points to the right of point 5 are increased and the section is re-plotted.



	1	2	3	4	5	6	7	8	9	10	11
X	0.0	1.5	2.0	2.75	3.0	4.00	4.25	5.00	5.50	7.00	
Y	1.0	0.9	0.45	0.45	.3	.3	0.45	0.45	0.90	1.00	
dX		1.50	0.50	0.75	0.25	1.0	0.25	0.75	0.50	1.50	
dY		-0.10	-0.45	0.00	-0.15	0.00	0.15	0.00	0.45	0.10	
dX/dY		-15.00	-1.11	489.58	-1.67	999.00	1.67	999.00	1.11	15.00	

Figure 4-25 – Changing dX between points #5 and #6 widens the entire channel

If you widen the channel you may find the plot is wider than the current window. This can be fixed easily by clicking [OK] which will cause the graphic scales to be adjusted automatically.

4.4.10 Adjusting the Graphic Scales

Horizontal 8

Vertical 1.8

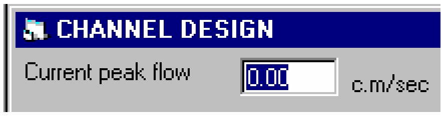
Figure 4-26 – The scale factors of the drawing window can be changed

The initial values of the scaling factors are X=10 m (30 ft) and Y=3.5 m.(10 ft) depending on the system of units selected. You can change these at any time, **except** when you are in the middle of drawing a cross-section. When you click on either the [OK] button or the [Design] button, the Horizontal and Vertical scales will be automatically adjusted to suit the dimensions of the cross-section.

Once you are satisfied with the design you can accept the Channel design.

4.4.11 Channel Design for Steady Flow

The **Design / Channel** command is one of the few menu items which are enabled as soon as the system of units is defined. This lets you use the Channel command to design a channel for a user specified flow.

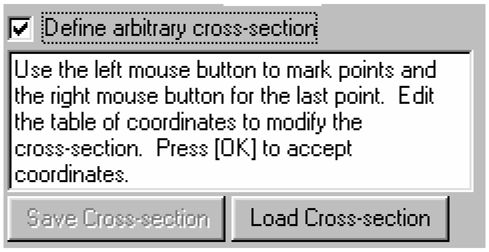


Current peak flow 0.00 c.m/sec

Figure 4-27 – The design flow can be entered if no Inflow hydrograph exists

If no Inflow hydrograph has been created the Channel window opens with a text box against the "Current peak flow" instead of simply displaying the current peak inflow. This lets you specify a flow rate and you can then design a channel without having to do any hydrological simulation. Type in a value and the table of feasible designs will be displayed. You can then proceed as usual.

4.4.12 Saving and Loading Cross-section Files



☑ Define arbitrary cross-section

Use the left mouse button to mark points and the right mouse button for the last point. Edit the table of coordinates to modify the cross-section. Press [OK] to accept coordinates.

Figure 4-27a – The channel dialog for saving or loading a cross section.

Once a complex cross-section has been defined by clicking on the [OK] button it is possible to save the data in a file which is stored by default in the current Output directory. You may navigate to another folder if desired. The default file extension is *.XSEC.

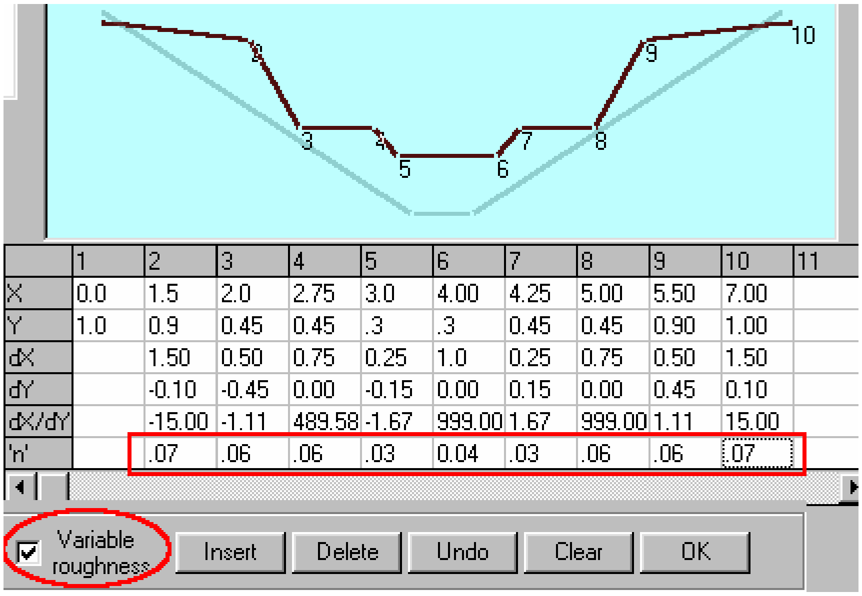
Subsequently the file can be imported into the Channel command and can be edited in the usual way by modifying the data displayed in the grid.

4.4.13 Using Variable Roughness

Complex cross-sections will often include both a main channel part and adjacent flood plain areas over which the flow depth is shallower and the roughness elements are much larger. To analyse the flow on this type of section it is necessary to do two things.

1. Use an array of values of Manning's 'n' for each segment in the cross-section.
2. Calculate the conveyance of the channel by subdividing the channel into a series of small channel segments and then estimate the total conveyance as the sum of the conveyance values for each segment.

Both of these requirements are provided by checking the check-box labeled "Variable Roughness".



	1	2	3	4	5	6	7	8	9	10	11
X	0.0	1.5	2.0	2.75	3.0	4.00	4.25	5.00	5.50	7.00	
Y	1.0	0.9	0.45	0.45	.3	.3	0.45	0.45	0.90	1.00	
dX		1.50	0.50	0.75	0.25	1.0	0.25	0.75	0.50	1.50	
dY		-0.10	-0.45	0.00	-0.15	0.00	0.15	0.00	0.45	0.10	
dX/dY		-15.00	-1.11	489.58	-1.67	999.00	1.67	999.00	1.11	15.00	
'n'		.07	.06	.06	.03	0.04	.03	.06	.06	.07	

☑ Variable roughness

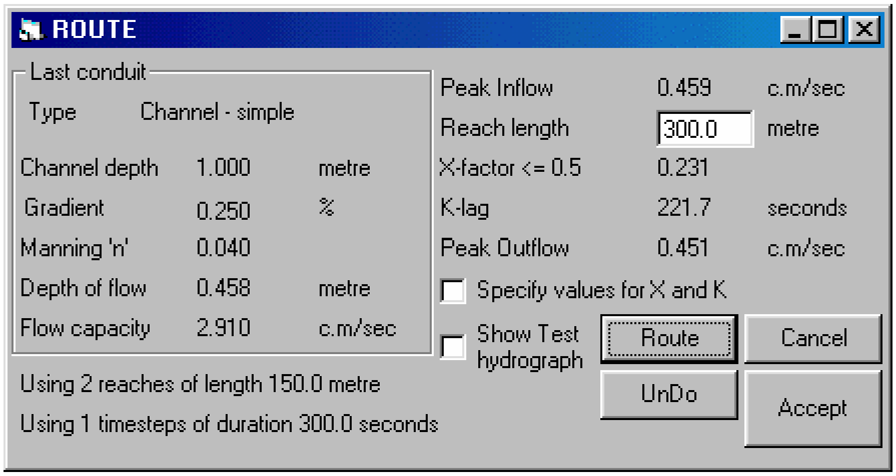
Figure 4_27b – Using variable roughness in a complex cross section

When this check box is selected an extra row in the data table is displayed. Initially this is filled with the default value of Manning 'n'. You can edit the values in the usual way to describe high values of roughness due to flood plane vegetation or other roughness elements.

The 2-dimensional method of calculation is also used automatically when Variable Roughness is invoked. The conveyance of each small segment of channel is affected not only by the roughness but also by the hydraulic radius which depends on the depth of flow. You will notice that if Variable Roughness is invoked but all of the Manning 'n' values are the same, the computed normal depth for a particular flow is slightly lower than that obtained when 1-dimensional analysis is used – i.e. when Variable Roughness is un-checked.

The variable roughness feature is disabled in Demonstration mode.

4.5 Routing the Inflow Hydrograph


ROUTE

Last conduit
Type Channel - simple
Channel depth 1.000 metre
Gradient 0.250 %
Manning 'n' 0.040
Depth of flow 0.458 metre
Flow capacity 2.910 c.m/sec

Peak Inflow 0.459 c.m/sec
Reach length 300.0 metre
X-factor <= 0.5 0.231
K-lag 221.7 seconds
Peak Outflow 0.451 c.m/sec
☐ Specify values for X and K
☐ Show Test hydrograph

Using 2 reaches of length 150.0 metre
Using 1 timesteps of duration 300.0 seconds

Figure 4-28 – Data required for the Route command

The **Route** command carries out a flood routing analysis of the current Inflow hydrograph through the most recently designed pipe or channel. The technique used is the Muskingum method with the modification that the weighting parameter X that defines the significance of the wedge-storage effect is computed by a method proposed separately by Cunge (1969) and Smith (1980). The background theory of the Modified Muskingum method is described in Chapter 8 *Theory of Hydraulics* – Theory of Kinematic Flood Routing.

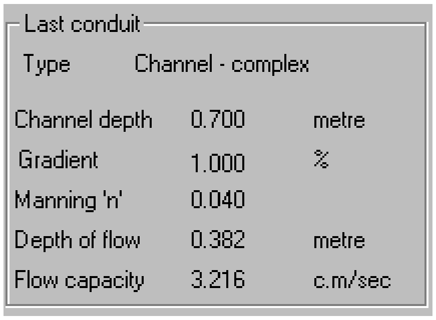
MIDUSS remembers whether the last link designed was a pipe or a channel. If neither a pipe nor a channel has been designed the Design/Route menu item is disabled.

The command involves the following four steps.

1. Check Parameters of Last Conduit - The properties of the last pipe or channel are displayed along with the peak inflow and the flow capacity of the conduit.
2. Specify a Conduit Length - You are prompted to supply the length of the reach (metres or feet).
3. The Muskingum Routing parameters K and X are estimated and displayed for you to either accept or modify.
4. Review and Accept the Results - The inflow hydrograph is routed through the reach and the peak outflow is displayed. You can review the results and accept them when satisfied.

If the reach length is zero the inflow hydrograph is copied to the outflow hydrograph. Very short but non-zero reach lengths may result in rather long execution times. For reaches of zero or negligible length you can use the **Hydrograph/Copy Inflow to Outflow** command.

4.5.1 Conduit Parameters for Route


Last conduit

Type Channel - complex

Channel depth 0.700 metre
Gradient 1.000 %
Manning 'n' 0.040
Depth of flow 0.382 metre
Flow capacity 3.216 c.m/sec

Figure 4-29 – Data for the last designed conduit is displayed.

The information displayed at the start comprises the pipe or channel geometry, roughness, gradient and uniform flow depth. The peak value of the inflow hydrograph and the pipe or channel capacity are also shown. If a pipe is surcharged, the inflow hydrograph is copied to the outflow hydrograph with no attenuation or lag. If a channel is overtopped, a warning message and the outflow hydrograph is left equal to the inflow hydrograph.

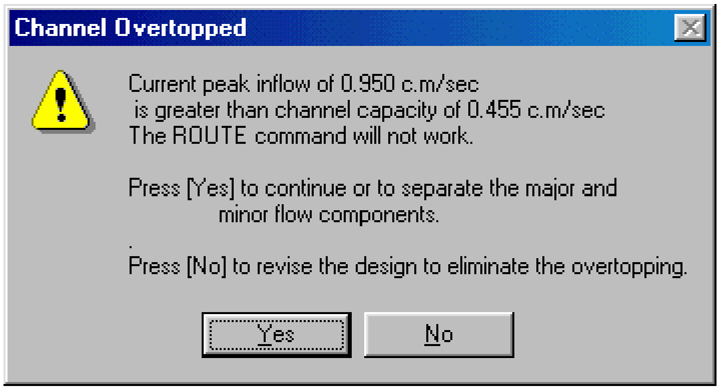
Sometimes you may define a pipe or channel cross-section which is to remain unchanged for several links. Over this length the discharge may increase quite significantly due to runoff from one or more sub-areas. If you cause the inflow to be increased by means of the **Hydrograph/Add Runoff** command and then proceed directly to the **Route** command MIDUSS will continue to use the last-defined pipe or channel but will update the uniform flow analysis to obtain the correct depth and velocity. If this happens you will see the informative message shown in Figure 4-30 below and the new depth of flow will be shown in the 'Last Conduit' frame.


MIDUSS

Channel depth of flow has been updated to 0.382 metre

Figure 4-30 – Information message provided by the Route command.

However, if you increase the flow to a value in excess of the pipe or channel capacity the computed depth of uniform flow will exceed the specified overall depth and routing will be impossible. You will see a message advising you to re-design the pipe or channel with an increased flow capacity.


Channel Overtopped

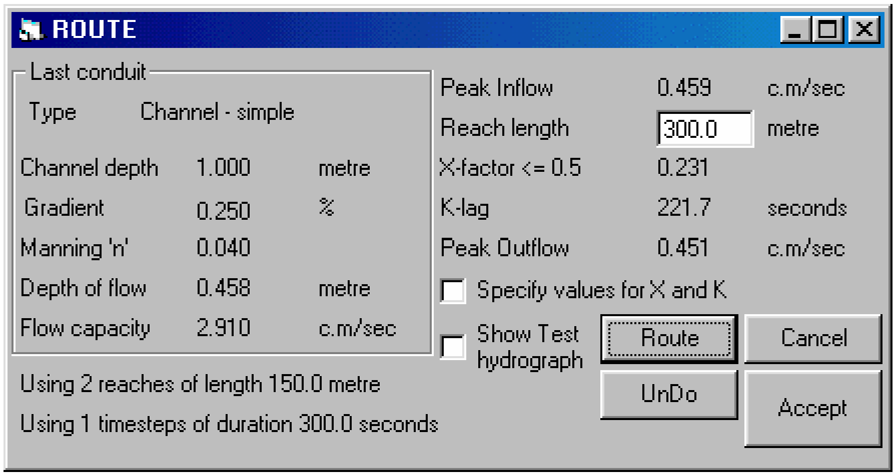
Current peak inflow of 0.950 c.m/sec
is greater than channel capacity of 0.455 c.m/sec
The ROUTE command will not work.

Press [Yes] to continue or to separate the major and
minor flow components.

.
Press [No] to revise the design to eliminate the overtopping.

Figure 4-31 – A warning is displayed if the channel is overtopped.

4.5.2 Conduit Length for Route



Type Channel - simple

Channel depth 1.000 metre

Gradient 0.250 %

Manning 'n' 0.040

Depth of flow 0.458 metre

Flow capacity 2.910 c.m/sec

Peak Inflow 0.459 c.m/sec

Reach length 300.0 metre

X-factor <= 0.5 0.231

K-lag 221.7 seconds

Peak Outflow 0.451 c.m/sec

☐ Specify values for X and K

☐ Show Test hydrograph

Using 2 reaches of length 150.0 metre

Using 1 timesteps of duration 300.0 seconds

Figure 4-32 – Data used by the Route command

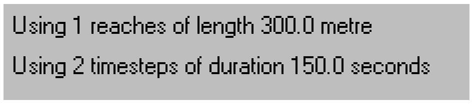
The reach length is specified simply by typing the value in the text box. When you do so the Muskingum routing parameters are computed and displayed as the values for 'X factor' and 'K-lag'. If you wish to specify your own values for these parameters you can check the box labeled 'Specify values for X and K' and the labels will be changed to text boxes so that you can modify them if you wish. However, it is not likely that you will be able to improve on the estimate made by MIDUSS.

If you enter a zero value for the reach length MIDUSS ignores the rest of the calculation and simply copies the Inflow hydrograph to the Outflow. A message will be displayed and the [Accept] button will be enabled.

Before pressing the [Route] button it is probably with looking briefly at the Muskingum Routing Parameters which have been estimated.

4.5.3 The Muskingum Routing Parameters

For the specified reach length the lag K is computed using a celerity which is the average of the flow velocity and the kinematic wave velocity for the peak discharge.



Using 1 reaches of length 300.0 metre

Using 2 timesteps of duration 150.0 seconds

Figure 4-33 – Changes required for numerical stability are reported

When a finite reach length is specified MIDUSS also checks on the numerical stability of the calculation and, if necessary, uses sub-multiples of the time step or the reach length to ensure stability. The result of this adjustment is displayed below the 'Last conduit' frame as illustrated in Figure 4-33.

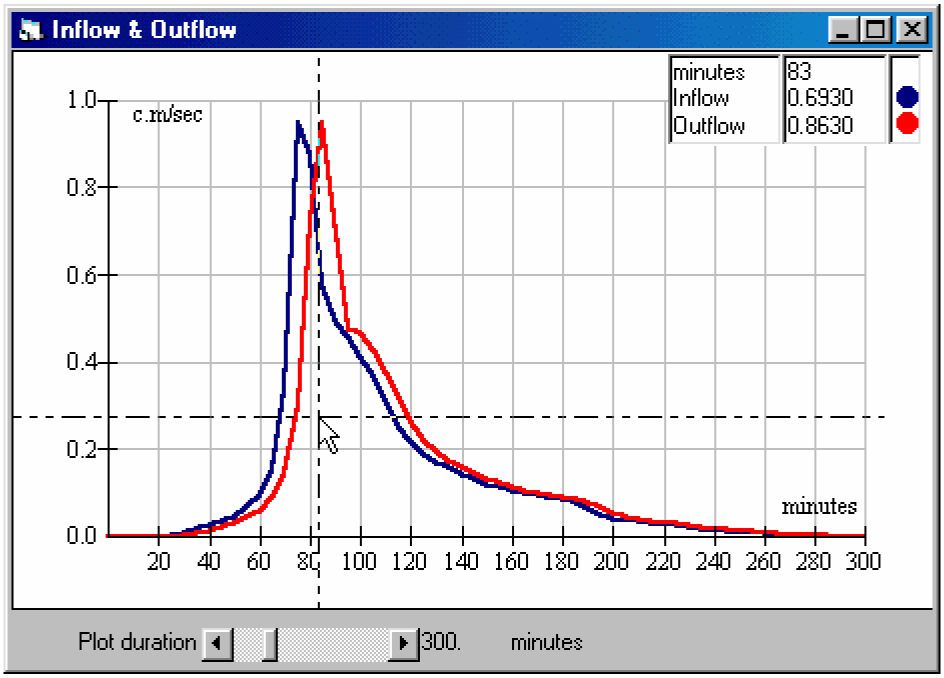
It can be shown (see eq. [8.41] in Chapter 8) that numerical stability of the Muskingum method requires that the spatial weighting coefficient X must satisfy the condition $X <= \delta t/2K <= (1-X)$

where δt is the routing time-step. For very short reaches this criterion requires a routing time step δt which is a sub-multiple of the hydrograph time step Δt (*i.e.* δt = Δt/n). Similarly, very long reaches are divided into sub-multiple reaches and multiple Muskingum routing is used. If multiple reaches are used you are informed and the lag K is displayed with an appropriate multiplying factor. The routing time step δt is also displayed. These checks for numerical stability are performed automatically by MIDUSS and you do not need to take any special action. Chapter 8 *Theory of Hydraulics* contains a full description of the method of calculating these parameters.

For the cases encountered in storm water drainage the values of X will usually be in the range 0.4 to 0.5. Lower values of X result in greater attenuation of the peak discharge.

The estimated values of X and K are displayed and you have the opportunity to either accept these or modify one or both values. Once the values are accepted the routing operation is performed and the peak outflow is displayed. At this point you can either accept the results obtained or return to the previous step at which the Muskingum routing parameters can be further modified. Assuming you wish to accept the values suggested by MIDUSS all you need to do is press the [Route] button.

4.5.4 Review and Accept Routing Results



minutes	83	
Inflow	0.6930	●
Outflow	0.8630	●

Figure 4-34 – Inflow and Outflow hydrographs following the Route command.

The best way to judge the result of the routing operation is from the graphical display as shown in Figure 4-34. The peak outflow is shown in the Route window and the modified Outflow hydrograph is displayed in a tabular form. However, the picture is what will provide a feeling of comfort or concern.

You can, of course, experiment with different values of the controllable variables and press [Route] again to see what difference this causes in the result. Once satisfied you can press the [Accept] key to save the design. The summary table of peak flows is updated to reflect the result.

4.6 Pond Design

In order to reduce the impact of urbanization it is common practice to provide storm water management in the form of a detention facility which reduces or 'shaves' the peak of the runoff hydrograph to an acceptable value. The actual facility may take a variety of forms such as:-

- an 'in-line' pond comprising a storage area with an outflow control device through which drainage occurs for all storm events.
- an 'off-line' storage chamber to which storm water is diverted when the water surface in a channel or the hydraulic grade line in a pipe rises above some elevation.
- roof-top storage on flat roof tops of industrial or commercial developments using flow control devices at the top of rainwater down pipes.
- parking lot storage on peripheral areas of large parking surfaces around shopping malls using inlet control devices in catchbasins.

The method is particularly effective in reducing the peak flow values of the 'peaky' hydrographs which result from large impervious areas.

In all of these facilities the principal of design is to provide a storage area which is normally dry and which fills rapidly when the runoff rate increases above a certain value. At any instant in time the rate of change of storage represents the difference between the inflow rate and the attenuated outflow.

Figure 4-35 illustrates a typical detention pond with an outflow control device comprising an open-topped box with an orifice in the upstream face. Floods of modest size are passed through the orifice but more extreme events cause overtopping of the weir around the upper edge of the box.

The inflow and outflow hydrographs for a typical event are also illustrated. The maximum storage volume is indicated by the shaded area between the two curves. Note also that the peak of the outflow hydrograph lies on the falling limb of the inflow hydrograph. At the instant when storage (and water level) reaches a maximum the inflow and outflow must be equal.

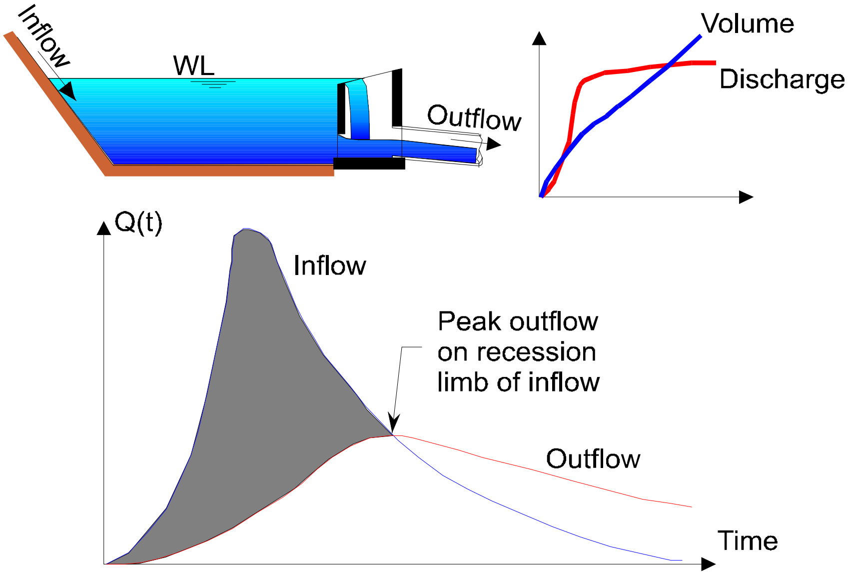


Figure 4-35 - An illustration of reservoir routing.

4.6.1 Pond Design - Main Steps

The POND command helps you to determine the proportions and discharge characteristics of a detention pond with the object of attenuating the peak flow in a link of a drainage network to an acceptable value. As with the other design commands in this chapter, the design comprises four steps.

1. Specify the desired peak outflow and determine an approximate storage volume which might achieve this.
2. Set up a table to define the Depth - Discharge and Depth -Storage volume relationships for the proposed pond. The discharge and storage functions must start from zero and increase monotonically. MIDUSS contains a number of special commands to help you to define the Depth - Discharge - Storage values for a variety of standard outflow control devices and storage units of different geometry.
3. Route the inflow hydrograph through the proposed pond and get peak values of outflow discharge, pond depth and storage volume.
4. Accept or reject the design. If rejected, edit the Depth Discharge and Storage table or the associated control devices and repeat step (3) with the revised configuration until an acceptable design is achieved.

Many of the options available in the Pond command are accessed through a special menu which is enabled when the **Design/Pond** command is selected. Refer to the topic **The Pond Menu** later in this chapter to see a summary of the main features of this menu.

A logical series of steps is summarized in the following list of topics. You may use these in an order different than that shown but the sequence illustrated here is typical.

- Specify the target outflow
- Set the Number of Stages
- Specify Minimum and Maximum Water Levels

- The Outflow Control Device
- Define an Orifice or Outflow Control
- Define a Weir Control
- Plotting the H-Q Curve
- Define Storage Devices
- A Single Stage Pond
- Describing a Pond with Multiple Stages
- Plotting Pond Storage
- Routing the Hydrograph
- Accepting the Pond Design

In addition to the general type of facility described in the above topics, the Pond command includes options for special types of stormwater management facility. You can review the main steps for these by referring to the topics below.

- Oversized Sewers (Super-Pipes)
- Parking Lot On-site Control
- Rooftop On-site Control

4.6.2 Oversized Sewers or Super-Pipes

Storage can sometimes be provided in the form of a reach of large diameter pipe, usually within a road allowance. The following sections describe the steps in designing this type of detention storage pond.

- Using a Super-Pipe for Storage
- Check the Storage Curve for the Super-Pipes
- View the Results of Super-Pipe Design

Some theoretical background is given in Chapter 8, *Theory of Hydraulics*; Super-pipes for Pond Storage

4.6.3 Parking Lot On-site Control

Commercial developments usually have a high percentage of impervious area (>75%) a large fraction of which is used for parking. Significant reduction in peak flows can be achieved by grading the parking area to form shallow storage areas around the catch-basins which drain the lot. The following steps describe how the pond command can be used to prepare a preliminary design for on-site control on parking areas.

- Using Parking Lot Storage
- Set limits on the Parking lot grading
- Defining the Wedge Storage
- Define the Parking lot Catchbasin Capacity
- View the Results of parking lot Storage

Background theory on Wedges (or Inverted Cones) for Pond Storage is contained in Chapter 8 *Theory of Hydraulics*

4.6.4 Rooftop On-Site Control

The following steps describe how the Pond command can be used to design rooftop storage on large commercial buildings such as shopping malls and "big box stores".

- Using Rooftop Storage
- Generating the Rooftop Inflow
- Target Outflow from the Roof
- Desired Depth Range on the Roof
- Parameters for the Rooftop System
- Rooftop Discharge and Storage Characteristics
- Rooftop Flow Routing
- Graphing Rooftop Runoff
- Design Tips for Rooftop Storage
- Rooftop Error Messages

More detailed information can be fund in Chapter 8 *Theory of Hydraulics*; Rooftop Flow Control for Pond Storage

4.6.5 The Pond Menu

Figure 4-36 – A special menu is provided for the Pond design

When the Pond window is displayed (and has the focus as indicated by the coloured title bar) a special menu replaces the main MIDUSS menu. It includes the following items most of which are discussed in more detail in the items which follow.

Main Menu	Lets you return to the main MIDUSS menu. Also enables the Clipboard controls.
Storage Geometry	Displays items to compute storage volumes as a function of the stage H for various storage facilities such as a single or multi-stage pond, an oversized storm sewer (sometimes called a 'super-pipe') and wedge or pyramid shaped storage (such as a parking lot).
Outflow Control	Contains items to let you compute the stage-discharge values for one or more (up to 10) of the following types of control device. Orifice controls, horizontal orifices, weir controls and outflow pipes with positive or negative gradient.
Rooftops	Lets you define parameters relevant to rooftop storage
Plot	Lets you display various types of plot involving the depth (or stage), discharge and storage volume.

Split Outflow If 2 or more outflow control devices are used, this menu item is available and lets you separate the outflow hydrographs for each devices.

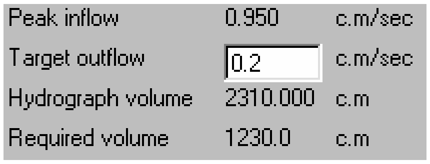
You can, of course, enter values directly into the grid of H, Q and V values in the Pond window. However, the tools provided will often provide a good estimate which you can refine if required. You can also transfer data from and to the grid using the Clipboard.

Sometimes when the Pond window loses the focus to another window (such as a graph display or a data entry form) the main MIDUSS menu is displayed in the menu bar.

4.6.5.1 Specify the target outflow

MIDUSS arbitrarily sets the target outflow to half the peak inflow. You will usually want to change this but if you do not, MIDUSS will remind you and ask to confirm the default target flow.

Click on the text box at “Target outflow” to highlight the current value and type in the desired peak outflow that you would like to achieve for this hydrograph.



Peak inflow	0.950	c.m/sec
Target outflow	0.2	c.m/sec
Hydrograph volume	2310.000	c.m
Required volume	1230.0	c.m

Figure 4-37 – Specify the target outflow to see an estimate of the required storage.

As shown in Figure 4-37, an approximate estimate of the required storage volume is displayed for information along with the total volume of the inflow hydrograph. The storage required will increase as the desired peak outflow is reduced.

If you do not explicitly accept or define the target outflow you will be reminded to do so when you try to use the Geometry Storage commands.

4.6.5.2 Set the Number of Stages

The default number of stages or water levels used to describe the Depth - Discharge - Storage functions is set at 21. If you have changed this the most recent value will be displayed.

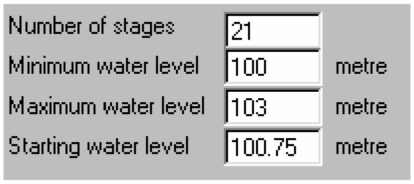
Number of stages 21

Figure 4-38 – Up to 50 stages can be defined.

Click on the text box to highlight the current value. Type in a new value. The number of stages must be at least 3 and not more than 50. The default value of 20 depth increments (21 stages) will often be sufficient.

4.6.5.3 Minimum, Maximum and Starting Water Levels

The minimum and maximum water levels should be defined next.



Number of stages 21

Minimum water level 100 metre

Maximum water level 103 metre

Starting water level 100.75 metre

Figure 4-39 – The range of depth is defined in terms of actual elevation.

Click on each of these data entry boxes and type in the desired value. The maximum level must be greater than the minimum water level. As you change these values the column of levels in the H-Q-V Table changes as long as the values are feasible (i.e. Maximum > Minimum).

If the pond is to contain some permanent storage (sometimes called a wet pond) you may also wish to specify the invert level of the lowest outflow control as the starting water level. Note that if the outflow is controlled by an outflow pipe with a negative slope, the starting water level will be the downstream invert of the pipe.

4.6.6 The Outflow Control Device

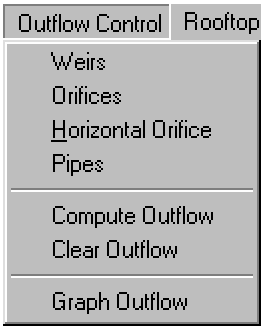


Figure 4-40 – The menu for the Outflow control

The outflow control device usually comprises a number of devices designed to control a wide range of outflow. By clicking on either of the first two items you can include as many as 10 orifices and 10 weirs. Usually only one or two of each is required.

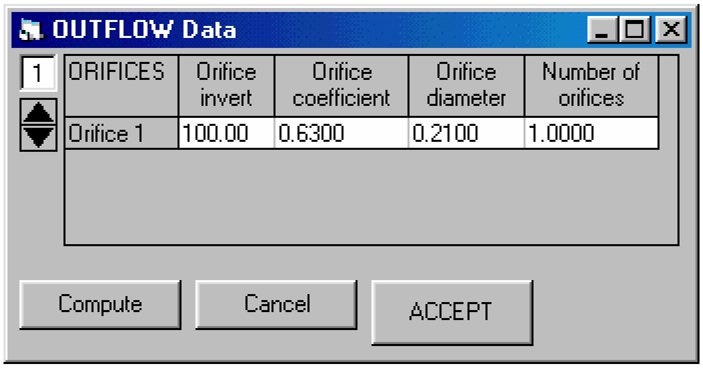
In addition, you can define one or more horizontal orifices. The term ‘horizontal’ applies to the plate in which the orifice is formed. The flow will therefore be vertically downward.

Finally, you can define an outflow pipe. If the pond is intended to provide some permanent storage to improve outflow quality, it is common to install a pipe with a negative slope that draws water from below the permanent surface.

It is normal to define the first orifice at the lowest level to control the low flows and then add another orifice or a weir at a higher level to handle the larger flows. Orifices require a much larger head to pass a significant discharge than with a weir. The result of combining an orifice and a weir is a very nonlinear stage-discharge curve.

4.6.6.1 Define an Orifice Control

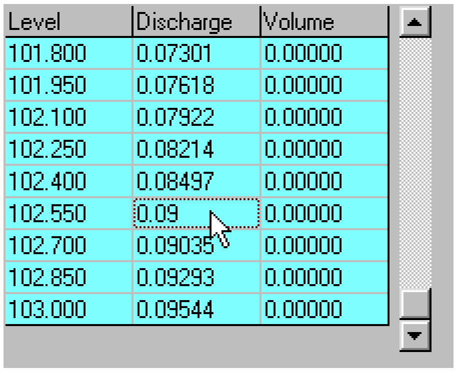
When you select the menu item ‘**Outflow Control / Orifice**’ a grid table is displayed with four columns for Invert level, discharge coefficient, orifice diameter and the number of identical orifices. Initially no empty rows are displayed. Click on the 'Up-Arrow' of the Spin-Button to open up a row for the first orifice.



ORIFICES	Orifice invert	Orifice coefficient	Orifice diameter	Number of orifices
Orifice 1	100.00	0.6300	0.2100	1.0000

Figure 4-41 – Data required to define an orifice.

MIDUSS displays suggested values showing the invert at the lowest water level, a coefficient of 0.63 and a diameter estimated to pass a reasonable fraction of the peak inflow. You can edit these values by selecting a cell by clicking the mouse pointer in it and then typing the desired value.



Level	Discharge	Volume
101.800	0.07301	0.00000
101.950	0.07618	0.00000
102.100	0.07922	0.00000
102.250	0.08214	0.00000
102.400	0.08497	0.00000
102.550	0.09	0.00000
102.700	0.09035	0.00000
102.850	0.09293	0.00000
103.000	0.09544	0.00000

Figure 4-42 – Results after defining an orifice control.

When you are ready, press the [Compute] button. You will see the column of Discharge values fill up in the H-Q-V table. The maximum discharge corresponding to the maximum head or water level is highlighted as shown. You can edit the values in the Orifice data entry table and re-compute the discharge column until you are satisfied.

The [Accept] button is enabled when you press [Compute] and is disabled if you change the data. When ready, press the [Accept] button to accept this orifice design. This causes the highlighted cell to be restored to normal.

You can define other orifices at this stage but it is recommended that you start with one and then add other devices after you have seen the result of an initial Route operation.

You can see a plot of the stage discharge curve for the orifice you have just designed by using the Plot Command.

Normally to complete the outflow control device you need to define a Weir Control to handle extreme events.

4.6.7 Plot the Orifice H-Q Curve

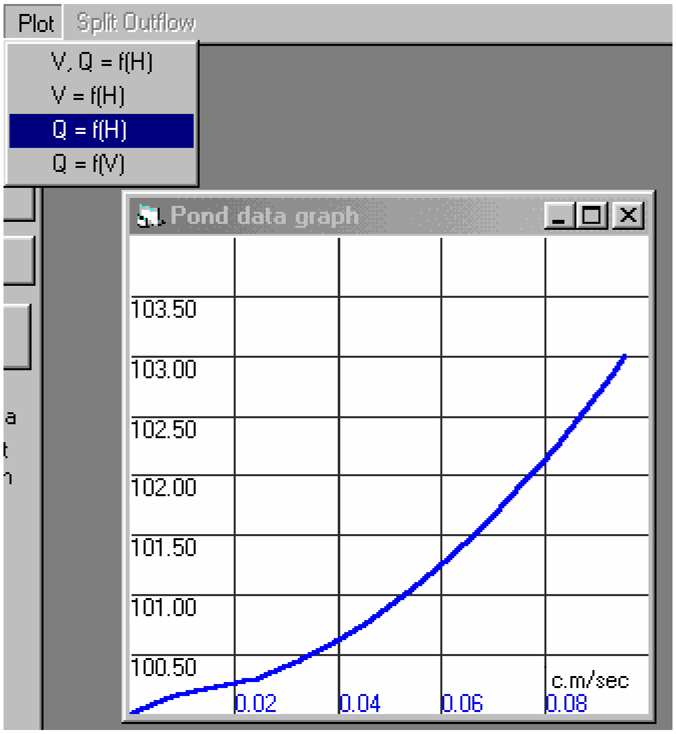


Figure 4-43 – Plotting the stage discharge curve for the orifice.

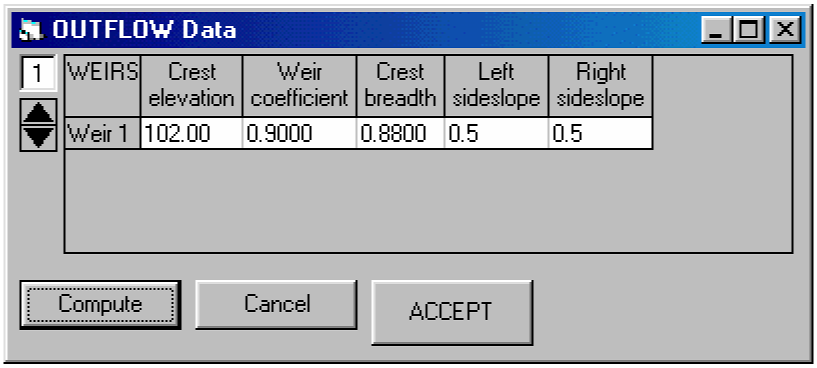
The plot menu offers several choices as shown. All but one of these plot one or more variables against elevation H. To see the stage-discharge curve for the current outflow control select the plot you require. If only an orifice has been defined so far, select the 'Q = f(H)' option.

A small graph is displayed as shown here for a single orifice located with its invert at the lowest water level. The slight change in curvature at the foot of the curve represents discharge when the water level is lower than the obvert or top of the orifice. This graph window can be re-sized as usual by dragging an edge or corner of the window.

This orifice will control low flows. To provide a control for more extreme events you should specify a weir control at a higher level. The next step is to define a Weir Control.

4.6.8 Define a Weir Control

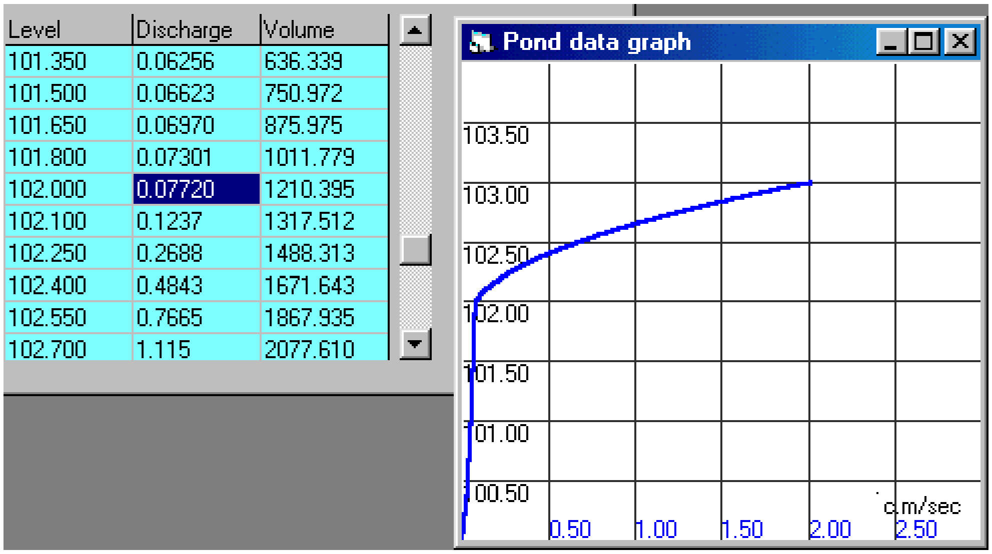
When you select the menu item ‘Outflow Control/Weir’ the grid table displayed has five columns to let you specify the crest level, a discharge coefficient, the breadth of the horizontal crest and the left and right side slopes. Initially no empty rows are displayed. Click on the ‘Up-Arrow’ of the Spin-Button to open up a row for the first weir.



WEIRS	Crest elevation	Weir coefficient	Crest breadth	Left sideslope	Right sideslope
Weir 1	102.00	0.9000	0.8800	0.5	0.5

Figure 4-44 – Data required for a weir specification

MIDUSS displays suggested values showing the crest elevation at approximately 70% of the total pond depth and a discharge coefficient of 0.9. The breadth is estimated to pass the peak discharge with a reasonable ratio of depth to breadth and the side slopes are assumed to be vertical (i.e. 0.0). In Figure 4-44, the crest elevation has been changed from 102.1 to 102.0 m and the sides have been given a slope of 0.5 H:1V (or about 27 degrees from the vertical). These values can be edited by clicking the mouse pointer on the appropriate cell and then typing the desired value. You can move between cells using the left and right arrow keys.



Level	Discharge	Volume
101.350	0.06256	636.339
101.500	0.06623	750.972
101.650	0.06970	875.975
101.800	0.07301	1011.779
102.000	0.07720	1210.395
102.100	0.1237	1317.512
102.250	0.2688	1488.313
102.400	0.4843	1671.643
102.550	0.7665	1867.935
102.700	1.115	2077.610

Figure 4-45 – Graphical and tabular results of adding a weir.

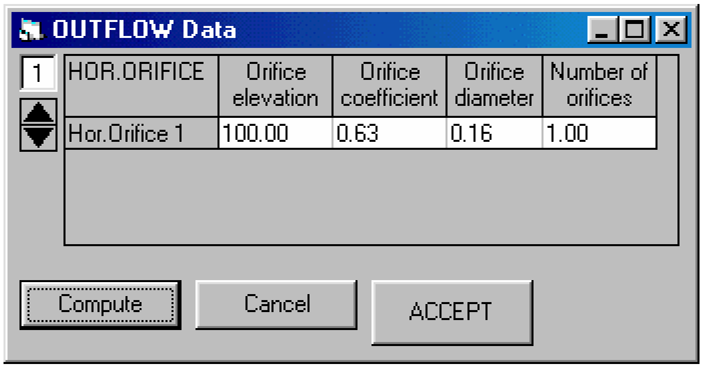
You can experiment by changing the parameters and pressing the [Compute] button with each change. With each trial you will note that in the HQV grid, the Discharge cell with a level above or equal to the crest elevation is highlighted and all discharges for levels above the crest will show an increased flow. Check to make sure that the maximum discharge (at the bottom of the grid) is close to or greater than the target outflow and not much less than the peak inflow otherwise you may get an error message when you attempt to Route the inflow through the pond.

If you have left the Plot window open, the graph of discharge is automatically updated with each change. The plot of Figure 4-45 shows the highly nonlinear relationship which results from the combination of orifice and weir.

Now that you have defined an outflow control device, the next step is to specify the storage characteristics of the pond. Click Defining a Storage Device to go to the next step or click here to return to the Main Steps of Pond Design

4.6.9 Define a Horizontal Orifice

When you select the menu item '**Outflow Control / Orifice**' a grid table is displayed (Fig 4-45a) with four columns for Invert level, discharge coefficient, orifice diameter and the number of identical orifices. Initially no empty rows are displayed. Click on the 'Up-Arrow' of the Spin-Button to open up a row for the first orifice.



HOR.ORIFICE	Orifice elevation	Orifice coefficient	Orifice diameter	Number of orifices
Hor.Orifice 1	100.00	0.63	0.16	1.00

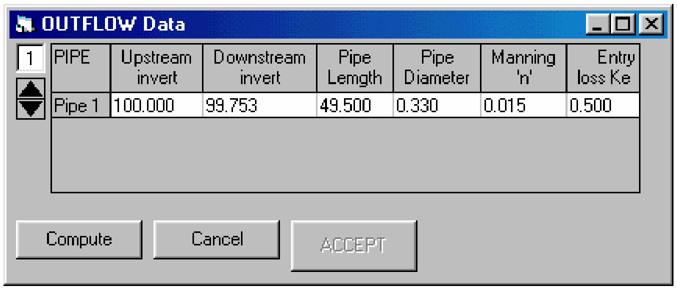
Figure 4-45a – Setting up data for a horizontal orifice.

The command is very similar to the normal (vertical) orifice plate but the discharge through the horizontal orifice plate is slightly higher, other factors being the same.

4.6.10 Define an Outflow Pipe Control

When you select the **Outflow Control/Pipes** menu a grid is displayed for the entry of six parameters for each outflow pipe. From left to right, these hold:

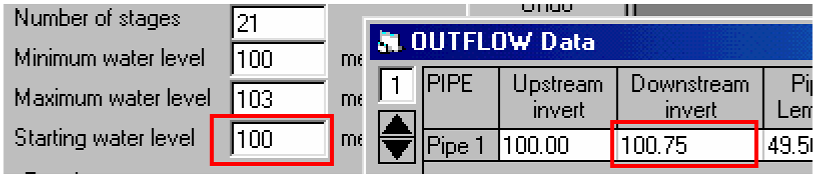
1. Upstream invert level of the pipe.
2. Downstream invert level of the pipe. (Note: The figure below shows a pipe with a negative slope.)
3. Pipe length
4. Pipe diameter estimated by MIDUSS
5. Pipe roughness (as Manning's 'n')
6. An entrance energy loss coefficient Ke



PIPE	Upstream invert	Downstream invert	Pipe Length	Pipe Diameter	Manning 'n'	Entry loss Ke
Pipe 1	100.000	99.753	49.500	0.330	0.015	0.500

Figure 4-45b Data for a pipe control

Note that the ‘Starting water level’ is initially set equal to the Minimum water level and this must be changed if the lowest outflow control is intended to retain some permanent water level. In the figure below, discharge will not commence until the water level in the pond is above 100.75 m (the downstream invert of the pipe) and the starting water level must match this.



Number of stages 21
Minimum water level 100
Maximum water level 103
Starting water level 100

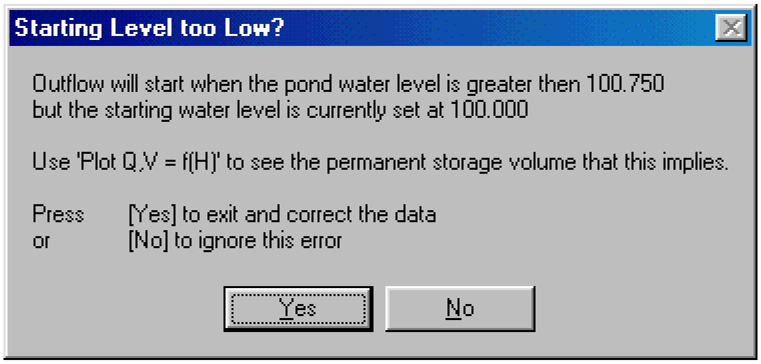
OUTFLOW Data

1	PIPE	Upstream invert	Downstream invert	Pi Len
	Pipe 1	100.00	100.75	49.5

Figure 4-45c – Mismatched D/S invert and Starting water level will produce a warning.

If you forget to do this, MIDUSS will detect the potential error when you press the [Route] button and will display a warning message similar to that shown below.

There may be circumstances when you want to start flood routing when the permanent water level has been drawn down for some reason but these cases are not usual.



Starting Level too Low?

Outflow will start when the pond water level is greater then 100.750
but the starting water level is currently set at 100.000

Use 'Plot Q,V = f(H)' to see the permanent storage volume that this implies.

Press [Yes] to exit and correct the data
or [No] to ignore this error

Figure 4-45d – The MIDUSS Starting Level warning message.

4.6.11 Defining Storage Devices

Figure 4-46 – The menu for the Storage Geometry command

Three types of storage device can be designed. These comprise:

1. an idealized rectangular pond with any length to width aspect ratio and one or more stages or layers;

2. storage in the form of an inverted cone which may form around a catch basin of a parking area, and

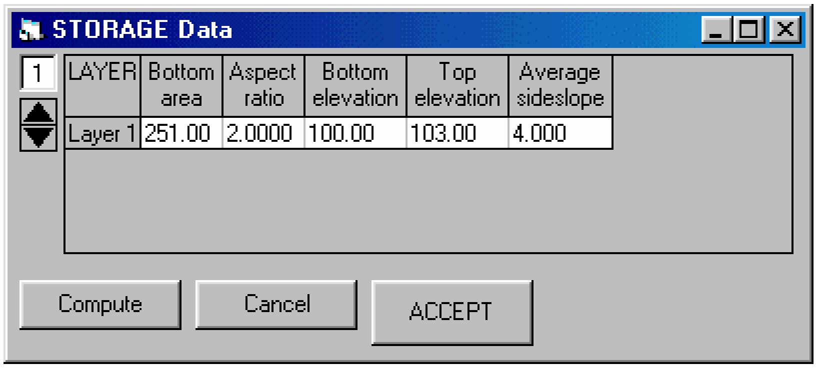
3. storage in oversized storm sewers - sometimes called 'super-pipes'.

You can see further discussion and illustration of these various storage facilities in Chapter 8 *Theory of Hydraulics*; Typical Storage Components for Detention Ponds.

The most commonly used is a surface detention storage pond.

4.6.12 A Single Stage Pond

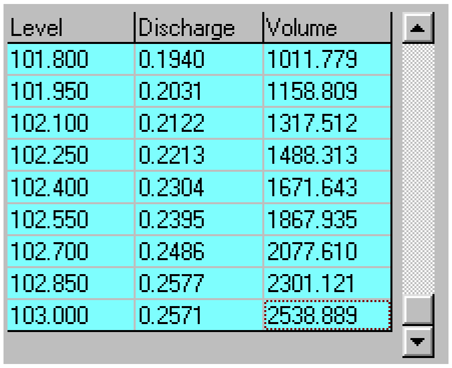
When you select the menu item **Storage Geometry/Rectangular pond** the grid table displayed has five columns. These let you specify the base area, the length/width ratio at the base, the bottom and top elevations of the layer and the side slope which is assumed constant around the layer being defined. Initially no rows are displayed and you must click on the 'Up-Arrow' of the Spin-Button to open up a row for the first layer.



LAYER	Bottom area	Aspect ratio	Bottom elevation	Top elevation	Average sideslope
Layer 1	251.00	2.0000	100.00	103.00	4.000

Figure 4-47 – Data required for a single layer rectangular pond.

MIDUSS displays suggested values showing the rectangular area at the bottom of the pond. This is estimated from the target outflow to be achieved and is only approximate. The aspect ratio is set to 2.0. This will change for any upper layers depending on the side slope adopted. MIDUSS assumes a single layer covering the total depth which you have specified and assumes also a bank side slope of 4H:1V for ease of maintenance or landscaping. All of these values can be edited by clicking the mouse pointer on the appropriate cell and then typing the desired value. You can move between cells using the left and right arrow keys.



Level	Discharge	Volume
101.800	0.1940	1011.779
101.950	0.2031	1158.809
102.100	0.2122	1317.512
102.250	0.2213	1488.313
102.400	0.2304	1671.643
102.550	0.2395	1867.935
102.700	0.2486	2077.610
102.850	0.2577	2301.121
103.000	0.2571	2538.889

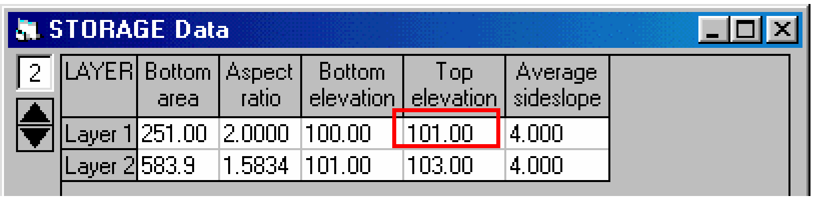
Figure 4-48 – Result of defining a rectangular pond.

Press the [Compute] button to see the range of volumes which this design will produce. You can experiment by changing the parameters pressing the [Compute] button with each change.

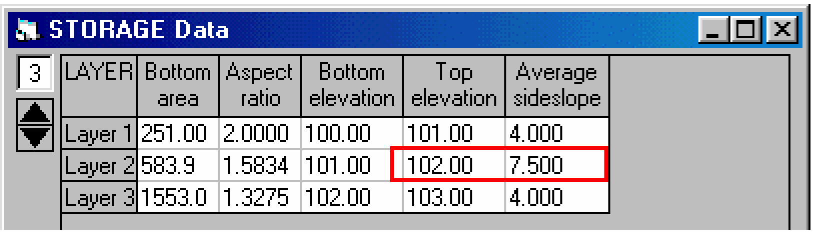
Frequently you may want to define a multi-layer pond which - for reasons of safety - has a much flatter bank slope for several metres above and below the normal water level. This is especially true if the pond is to have some permanent storage to provide some quality control. You can easily Define a Pond with Multiple Layers or Stages.

You can see a plot of storage as a function of depth by using the Plot Storage option in the **Pond/Plot** menu.

4.6.13 Describing a Pond with Multiple Stages



LAYER	Bottom area	Aspect ratio	Bottom elevation	Top elevation	Average sideslope
Layer 1	251.00	2.0000	100.00	101.00	4.000
Layer 2	583.9	1.5834	101.00	103.00	4.000



LAYER	Bottom area	Aspect ratio	Bottom elevation	Top elevation	Average sideslope
Layer 1	251.00	2.0000	100.00	101.00	4.000
Layer 2	583.9	1.5834	101.00	102.00	7.500
Layer 3	1553.0	1.3275	102.00	103.00	4.000

Figure 4-49 – Adding extra layers to a rectangular pond.

To increase the number of layers defining a pond, click once more on the up arrow to reveal a second row in the table. As shown in Figure 4-49, the top of the first layer (in red) has been changed from 103.0 to 101.0. The bottom of the second layer is automatically adjusted to match the top of the layer below. In addition, the bottom area and the aspect ratio have been automatically re-calculated to match the corresponding values at the top of the first layer.

A third layer can be added in the same way. In the lower figure, the cells boxed in red have been edited to set the depth of each of the three layers to 1.0 metre, and the bank slope of the second (middle) layer has been flattened to 7.5H:1V.

You can define up to 10 layers in this way. Often 3 or 4 are sufficient.

4.6.14 Plotting Pond Storage

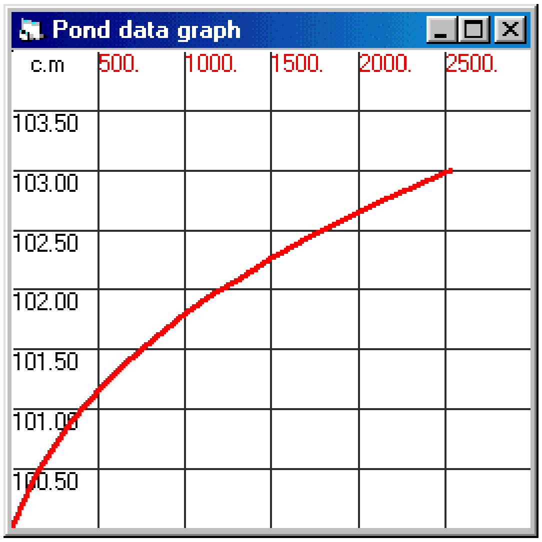


Figure 4-50 – A typical stage discharge curve for a rectangular pond

The **Pond/Plot** menu item lets you plot storage volume as a function of depth or - as illustrated in Figure 4-51 - you can plot both storage volume and discharge against depth or elevation on the same graph. If you are making changes to either the outflow control device or the storage facility the changes will be shown on the plot automatically as soon as you press the [Compute] button. Comparing the discharge and storage volume curves may be of interest if you plan to employ a flatter bank slope at or near the crest of the weir to reduce head over the weir for more extreme events.

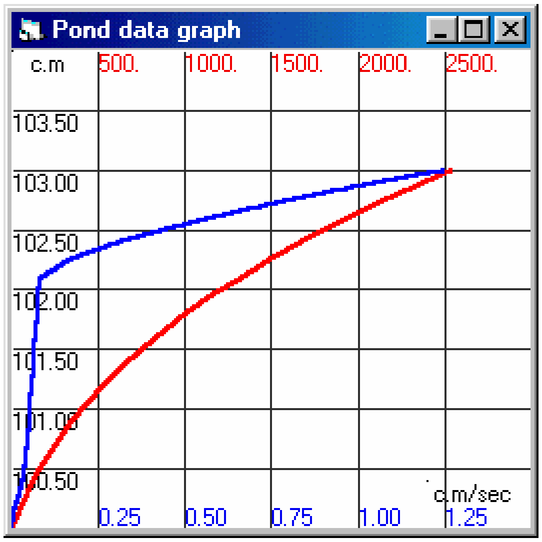


Figure 4-51 – Displaying a plot of discharge and storage

Although not shown here, you may also plot discharge Q as a function of the storage volume V. This will indicate the extent to which your design deviates from a linear reservoir. The estimates made in MIDUSS for the required storage are calculated assuming a linear reservoir which accounts for differences between the initial approximate estimate and the results obtained from routing through the nonlinear reservoir.

4.6.15 Using a Super-Pipe for Storage

Underground storage is relatively expensive but may be justified in cases such as commercial development where land values are high. Underground storage can be useful in conjunction with parking lot storage in order to reduce the frequency with which ponding around the catchbasins occurs to the annoyance of the parking public and the distress of commercial lease-holders. A typical arrangement is shown in Chapter 8 *Theory of Hydraulics*; Super-Pipes for Pond Storage.

As shown below, the **Storage Geometry** menu contains an item for **Super pipes** that expands to show the five cross-sectional shapes that are available in MIDUSS.

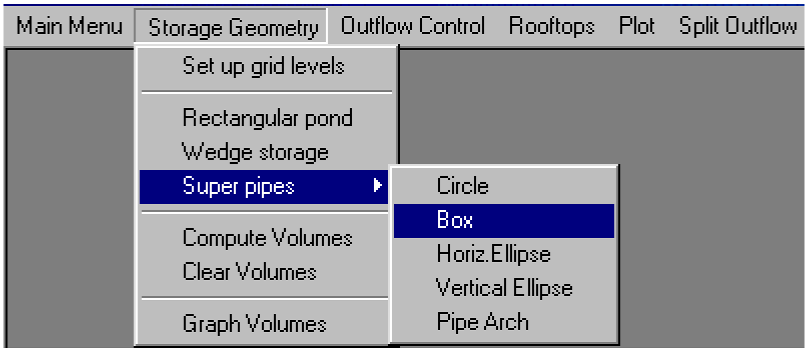


Figure 4-52a – Selecting the Super pipe storage geometry

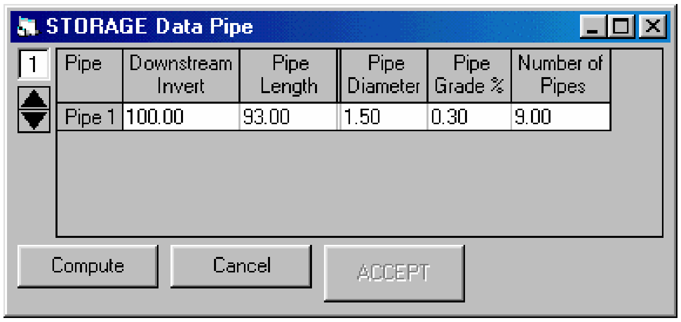
The data requirements differ slightly for the various cross-sectional shapes:

4.6.15.1 Circle (Pipe)

For all types, the downstream invert is assumed to be at the specified minimum water level.

For a pipe, diameter and grade are required.

Length is assumed to have a maximum value of around 100 m or 300 ft and multiple lengths are estimated to provide the estimated required volume.



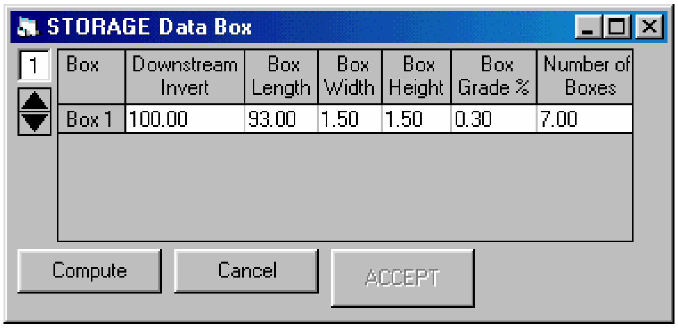
Pipe	Downstream Invert	Pipe Length	Pipe Diameter	Pipe Grade %	Number of Pipes
Pipe 1	100.00	93.00	1.50	0.30	9.00

Fig 4-52b – Data for a circular (pipe) Super-pipe

4.6.15.2 Box

Rectangular sections are initially assumed to be square but width and height can be altered.

Multiple lengths be arranged in various ways to suit the site configuration.



Box	Downstream Invert	Box Length	Box Width	Box Height	Box Grade %	Number of Boxes
Box 1	100.00	93.00	1.50	1.50	0.30	7.00

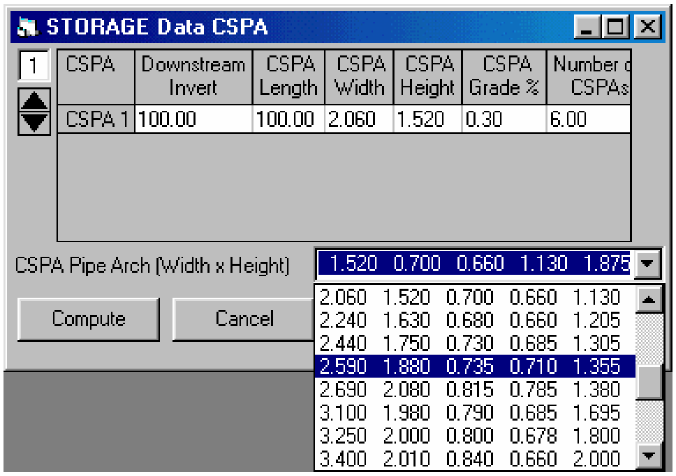
Fig 4-52c - Data for a box type Super-pipe

4.6.15.3 Elliptical and Pipe Arch

For the other cross-sections, a drop-down list of commercially available sizes is available.

Clicking on a selected size causes the corresponding width and height to be copied to the grid.

For pipe arch sections, other dimensions are copied for springing height and various radii. These are not displayed but are available within the program for calculation of cross-section properties.



CSPA	Downstream Invert	CSPA Length	CSPA Width	CSPA Height	CSPA Grade %	Number o CSPAs
CSPA 1	100.00	100.00	2.060	1.520	0.30	6.00

CSPA Pipe Arch (Width x Height): 1.520 0.700 0.660 1.130 1.875

2.060	1.520	0.700	0.660	1.130
2.240	1.630	0.680	0.660	1.205
2.440	1.750	0.730	0.685	1.305
2.590	1.880	0.735	0.710	1.355
2.690	2.080	0.815	0.785	1.380
3.100	1.980	0.790	0.685	1.695
3.250	2.000	0.800	0.678	1.800
3.400	2.010	0.840	0.660	2.000

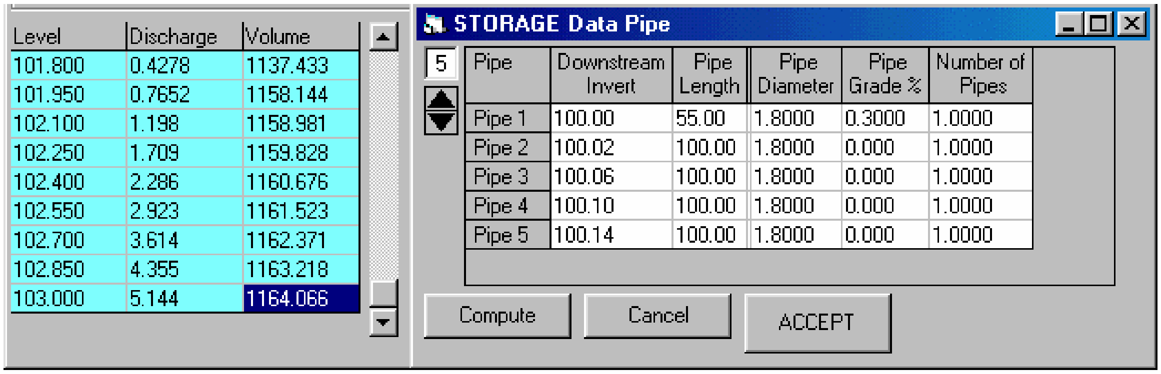
Fig 4-52d - Data for a Pipe Arch Super-pipe

4.6.16 Super-Pipe Data entry

A typical arrangement is shown in Chapter 8 *Theory of Hydraulics*; Super-Pipes for Pond Storage.

The data entry table shown in Figures 4-52b to 4-52d above is similar to others used in the Pond command and contains columns for:

- downstream invert level
- length of the super-pipe
- diameter of the pipe
- gradient
- number of pipes



Level	Discharge	Volume
101.800	0.4278	1137.433
101.950	0.7652	1158.144
102.100	1.198	1158.981
102.250	1.709	1159.828
102.400	2.286	1160.676
102.550	2.923	1161.523
102.700	3.614	1162.371
102.850	4.355	1163.218
103.000	5.144	1164.066

STORAGE Data Pipe

5

Pipe	Downstream Invert	Pipe Length	Pipe Diameter	Pipe Grade %	Number of Pipes
Pipe 1	100.00	55.00	1.8000	0.3000	1.0000
Pipe 2	100.02	100.00	1.8000	0.000	1.0000
Pipe 3	100.06	100.00	1.8000	0.000	1.0000
Pipe 4	100.10	100.00	1.8000	0.000	1.0000
Pipe 5	100.14	100.00	1.8000	0.000	1.0000

Figure 4-52e – Data required to define two super-pipes and the resulting storage

Figure 4-52f shows four 100 m pipes meeting at a fifth 55 m gallery all with diameters of 1.8 metres. It is important to install the pipes at a reasonably flat gradient in order to get maximum benefit from the potential storage. If the downstream inverts were all the same (i.e. 100.0m) then only 2 data lines would be needed on the form. The second line in Figure 4-52e would have ‘4’ as the Number of Pipes (because they would all be identical).

Figure 4-52f shows a diagram of the super-pipe design. The data in the Storage Data pipe window above (figure 4-52e) reflects the information on this diagram.

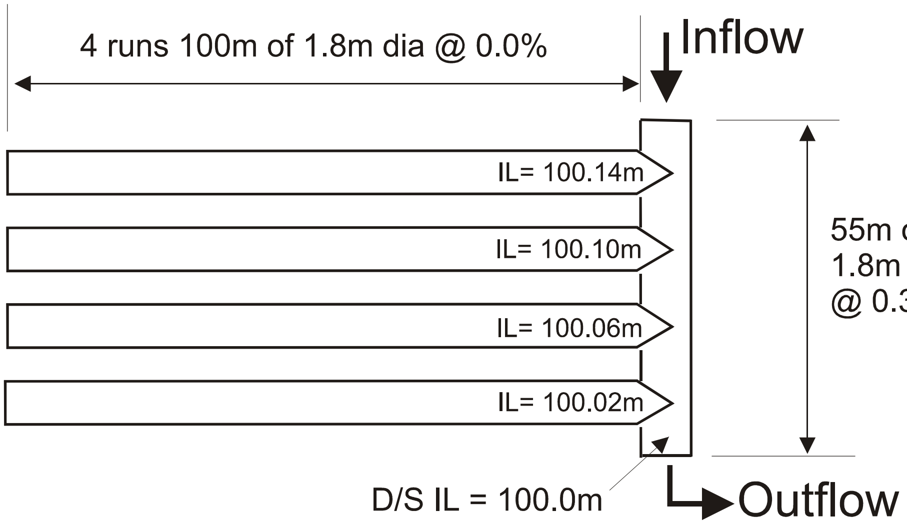


Figure 4-52f - A super-pipe design

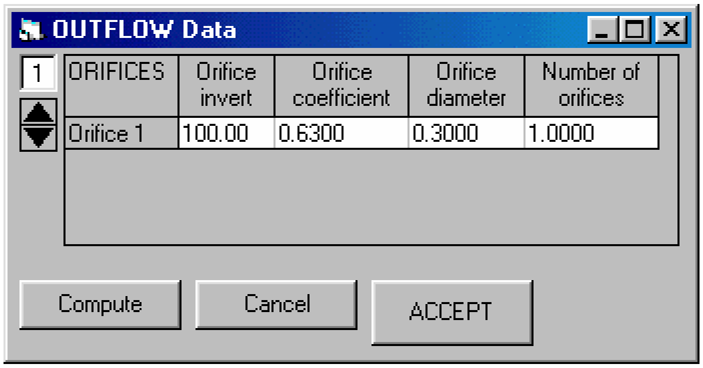
On pressing the [Compute] button, the maximum potential storage is seen to be 1164 cub.m.

The use of Super-Pipes is described in the following topics.

- Outflow Control for the Super-Pipes
- Results of Super-Pipe Design

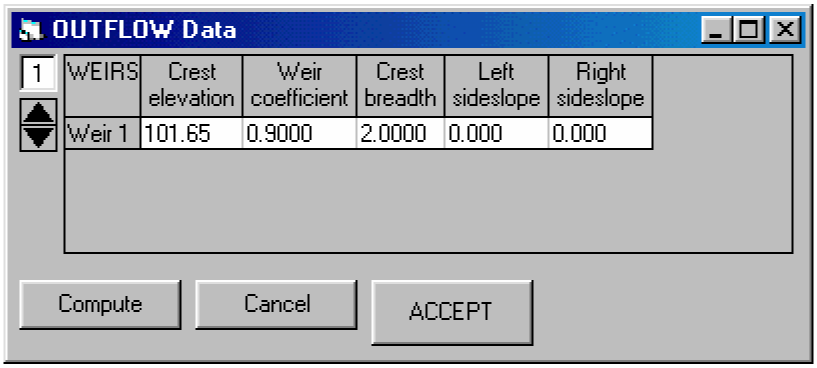
The first step is to define the outflow control device and then check a graphical display of the stage - discharge - storage characteristics of the system.

4.6.17 Outflow Control for the Super-Pipes



ORIFICES	Orifice invert	Orifice coefficient	Orifice diameter	Number of orifices
Orifice 1	100.00	0.6300	0.3000	1.0000

Figure 4-53a – Define an Outflow Orifice



WEIRS	Crest elevation	Weir coefficient	Crest breadth	Left sideslope	Right sideslope
Weir 1	101.65	0.9000	2.0000	0.000	0.000

Figure 4-53b – Defining the outflow control for the Super pipe

For simplicity, the outflow control device is assumed to comprise a single orifice and weir installed in the downstream manhole chamber thus controlling the flow in all of the super-pipes. Note that the weir crest is above the obvert at the downstream end but below the obverts of the other pipes at their upstream limits. If the weir is to be formed by a baffle at the downstream end of each pipe it will be necessary to consider the inflow in each branch of the system and design each super-pipe separately.

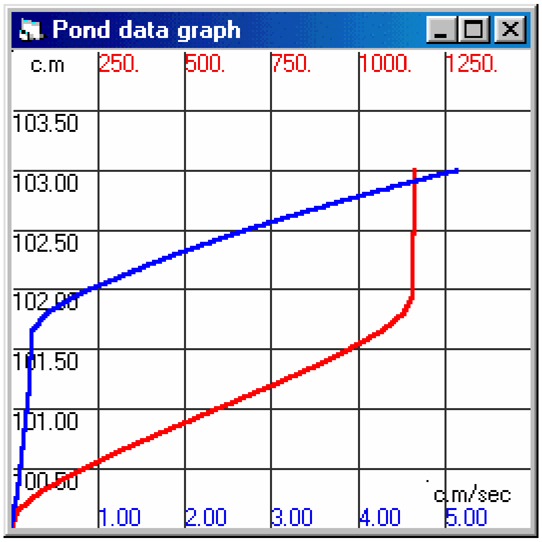


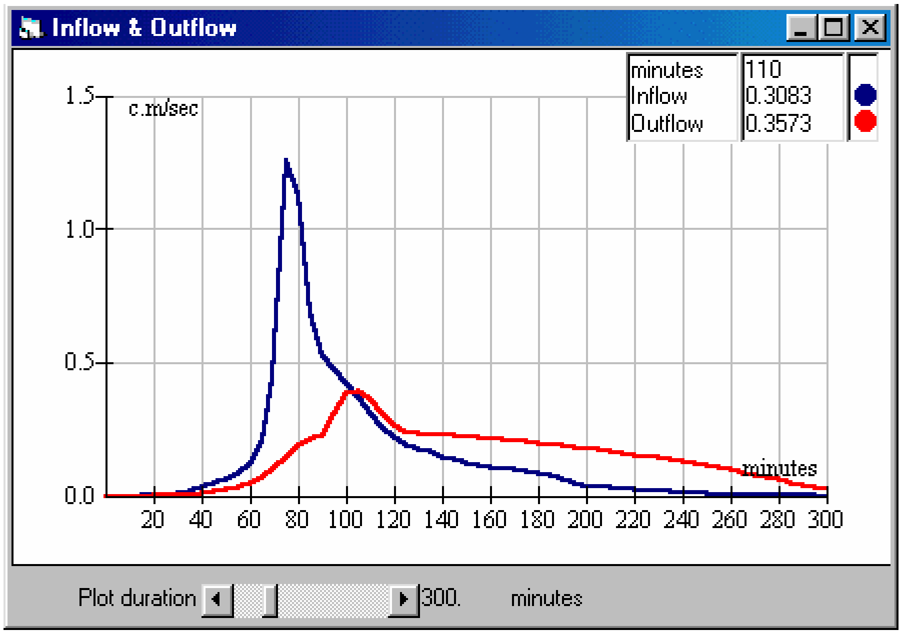
Figure 4-54 – Discharge and storage characteristics of the Super pipe system

Using the **Plot/V,Q = f(H)** option the storage and discharge curves are displayed as shown in Figure 4-54. Notice the reversal of the storage curve due to the closed top of the pipes close to an elevation of 102.0.

If the design appears reasonable, the final step is to press the [Route] button and see the results of the Super-Pipe design.

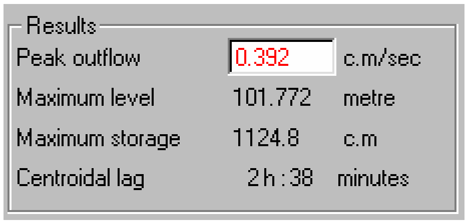
4.6.18 Results of Super-Pipe Design

Pressing the [Route] button produces the results shown below in Figure 4-55. The "Results" frame in the Pond window shows the peak outflow, maximum depth and storage in the facility. The estimate of required storage volume was in fact about 10% low but adequate for a preliminary design.



minutes	110	
Inflow	0.3083	●
Outflow	0.3573	●

Plot duration 300. minutes



Results

Peak outflow	0.392	c.m/sec
Maximum level	101.772	metre
Maximum storage	1124.8	c.m
Centroidal lag	2 h : 38	minutes

Figure 4-55 – Results of routing through the Super pipe system

The graph of the inflow and outflow hydrographs is shown. The "blip" on the outflow hydrograph indicates the point at which the weir is overtopped. In addition to the Results frame and the graph display, the outflow hydrograph is displayed in tabular form. If the super-pipe becomes almost full or surcharged there is no further volume available for storage and you will see that the outflow hydrograph will be identical to the inflow hydrograph along a part of the recession limb.

Thus a super-pipe will operate properly for the designed inflow but will pass higher peaks for more extreme events which cause surcharge upstream of the weir control. The solution in such cases may be to restrict the amount of flow which can be captured by the minor (piped) system. This can be done by limiting the number of catchbasins or installing flow restrictors (sometimes referred to as Inflow Control Devices or ICDs) in the tailpipes of the catchbasins thus forcing excess flow resulting from extreme storm events to be diverted to the major system.

4.6.19 Using Parking Lot Storage

Stormwater management on commercial developments often involves some measure of on-site control. A possible exception is when the development is immediately adjacent to a receiving watercourse with a large catchment area and with a time of concentration very much longer than that for the proposed development. In this case, the local runoff from a storm will be superimposed on the rising limb of the mainstream hydrograph resulting in zero or negligible increase in the mainstream peak flow.

When on-site control is required, storage on graded parking areas is commonly employed for this purpose. MIDUSS provides the **Wedge storage** option to provide assistance in estimating the stage-storage values for the volumes around catchbasins draining the parking area. The procedure is described in the following topics.

- Parking lot grading
- Defining Wedge Storage
- Parking lot Catchbasin Capacity
- Results of parking lot Storage

Typically the grading of a parking area uses two grades draining towards the catchbasin that produces a ponded volume shaped like an inverted pyramid or cone. MIDUSS assumes that the shape of the storage volume takes the form of an inverted cone the top surface of which is elliptical. The major radius and minor radius will equal the depth at the catchbasin multiplied by the grade in the relevant direction. A schematic of the arrangement is shown in Figure 4.56.

Since the maximum depth of storage is likely to be quite small - much smaller than for a conventional detention pond - the first step is to set minimum and maximum elevations for the Parking Lot grading.

4.6.20 Parking lot grading

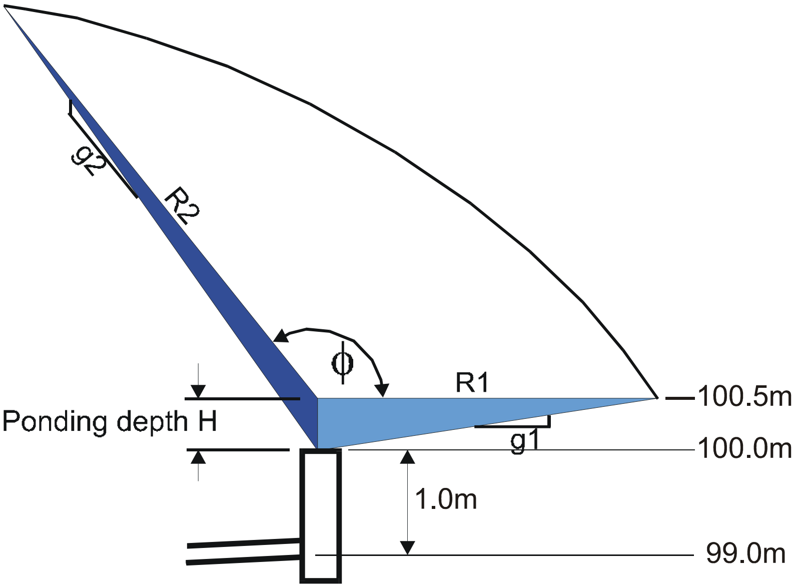
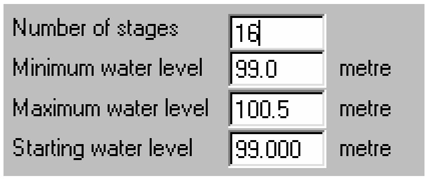


Figure 4-56 – A schematic for a catchbasin with a rim elevation of 100m

Most authorities require parking lot storage to be not deeper than a specified maximum to prevent vehicles from being floated and to allow access for emergency vehicles. In the example of Figure 4-56, assume that the maximum depth above the catchbasin rim elevation will be about 0.5 m (or 1.5 ft). The values entered for water levels will usually be actual site elevations. Notice that the invert of the catchbasin tailpipe will be 1.0 m below rim elevation, which in the illustration is 100.00 m. Using 15 depth increments (i.e. 16 depth values) provides values of discharge and volume at vertical intervals of 100 mm (or 4"). The storage volume will be zero for water levels below the rim elevation. For smaller depth increments you can use up to 50 water levels. Figure 4-56a shows you a portion of the Pond Design window where the water elevations and depth increments have been entered.



Number of stages	16	
Minimum water level	99.0	metre
Maximum water level	100.5	metre
Starting water level	99.000	metre

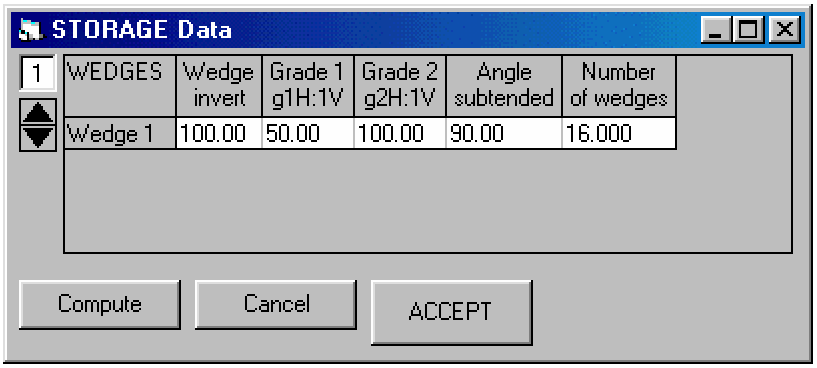
Figure 4-56a Setting water levels and number of stages.

The next step is to describe the geometry of the inverted storage cones around the catchbasins by Defining the Wedge Storage.

4.6.21 Defining Wedge Storage

The wedge has the general shape of a wedge of pie or a segment of a circle or ellipse. The parameters required are:

- The invert level (or rim level) of the catchbasin
- The grade g1 expressed as g1 H:1 V along one edge of the segment
- The grade g2 expressed as g2 H:1 V along the other edge of the segment
- The angle in degrees subtended at the centre of the segment (i.e. the catchbasin) by the two edges.
- The total number of such wedges.



WEDGES	Wedge invert	Grade 1 g1H:1V	Grade 2 g2H:1V	Angle subtended	Number of wedges
Wedge 1	100.00	50.00	100.00	90.00	16.000

Figure 4-57a – Data required to define wedge storage for 4 catch basins

The example shown describes four complete elliptical cones, each of which can be defined in terms of four 90 degree segments. Thus there are a total of 16 quarter segments. The grades used are 2% or 50H:1V on the steepest edge and 1% or 100H:1V on the flattest edge. The catchbasin rim elevation is set as 100.00 m and the angle subtended by each quarter segment is 90 degrees.

Figure 4-57b show a plan view of the parking lot.

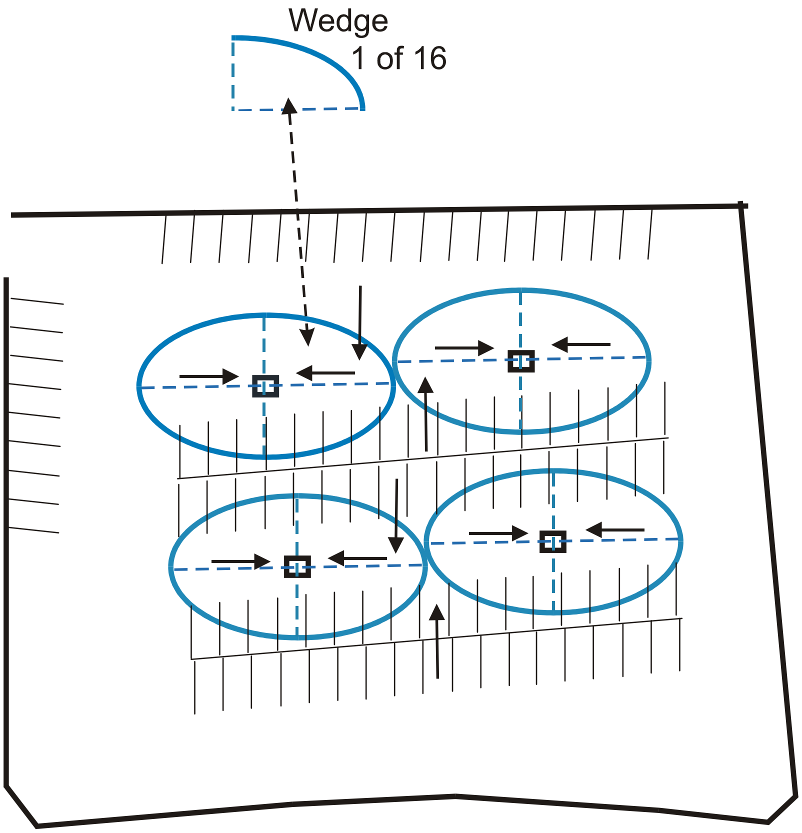
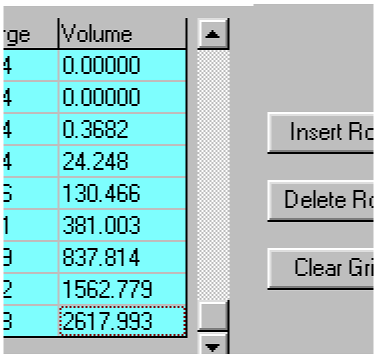


Figure 4-57b – Parking lot showing 16 wedges.



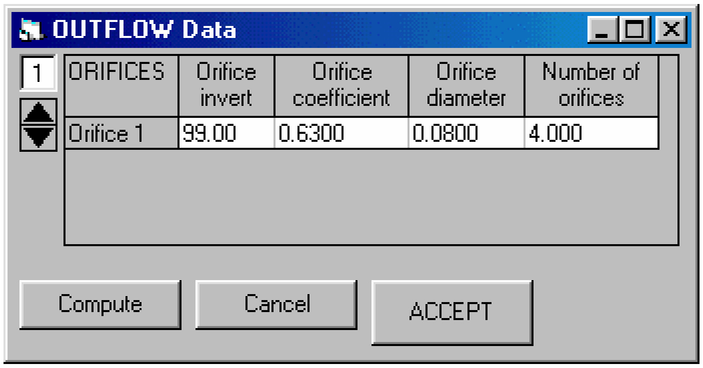
rge	Volume
4	0.00000
4	0.00000
4	0.3682
4	24.248
6	130.466
1	381.003
9	837.814
2	1562.779
3	2617.993

Figure 4-58 – Volume generated by the wedge storage

Pressing the [Compute] button causes the volume vector of the H-Q-V grid to be computed as shown in Figure 4-58. If the maximum depth is 0.5 m (which is probably greater than is desirable) the total volume available would be 2618 cub.m

The next step is to define the discharge capacity of the four catchbasins draining these four elliptical areas.

4.6.22 Parking lot Catchbasin Capacity

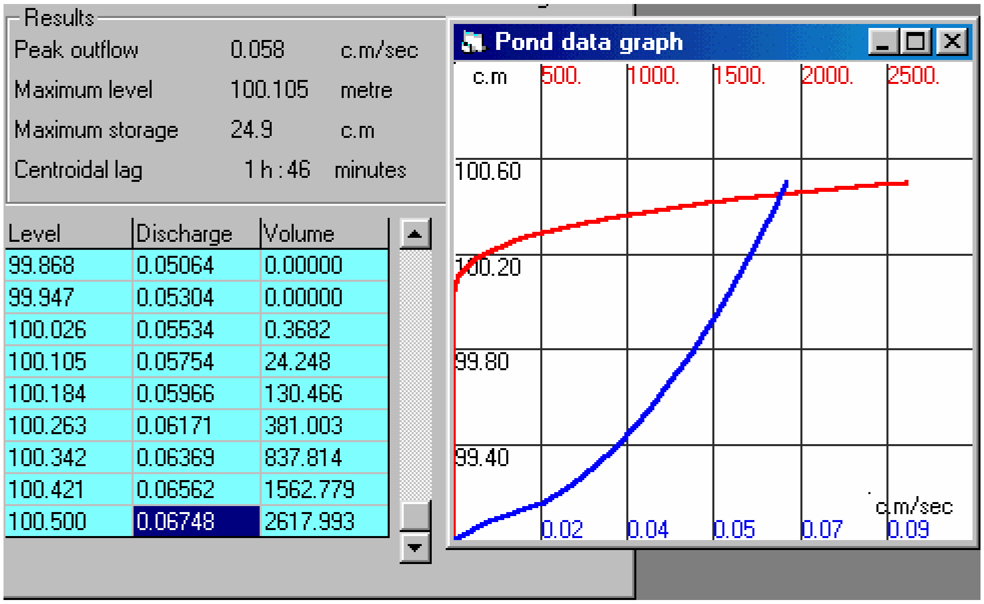


1	ORIFICES	Orifice invert	Orifice coefficient	Orifice diameter	Number of orifices
	Orifice 1	99.00	0.6300	0.0800	4.000

Figure 4-59a – An outflow control for four catchbasins

Four identical orifice controls are defined using the **Orifice** option of the **Outflow Control** menu. The invert of the tailpipe from the catchbasin is assumed to be 1.0 m (or 3.3 ft) below the rim level, that is 100.0 - 1.0 or 99.0 m elevation. A diameter of 80mm (3.25") is assumed here and a coefficient of discharge = 0.63 is used.

Pressing the [Compute] button generates a maximum discharge for the four catchbasins of 0.068 c.m/sec (or 2.4 cfs) when the depth over the catchbasin rim is 0.5 m. For extreme events you may wish to simulate the spill level from the parking area by defining one or more very flat, triangular weirs. This has not been done in this example.



Results

Peak outflow	0.058	c.m/sec
Maximum level	100.105	metre
Maximum storage	24.9	c.m
Centroidal lag	1 h : 46	minutes

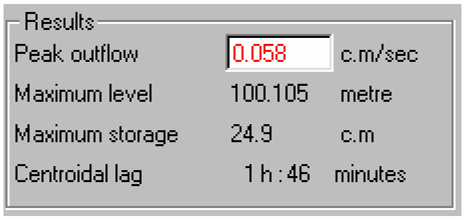
Level	Discharge	Volume
99.868	0.05064	0.00000
99.947	0.05304	0.00000
100.026	0.05534	0.3682
100.105	0.05754	24.248
100.184	0.05966	130.466
100.263	0.06171	381.003
100.342	0.06369	837.814
100.421	0.06562	1562.779
100.500	0.06748	2617.993

Figure 4-59b – Discharge and storage characteristics of the parking lot storage.

A plot of discharge and volume against the head H (or inspection of the H-Q-V table) shows that the volume is zero for values of H less than 100 m but discharge is finite over the full depth range. This special case can result in instability in the normal storage routing procedure and a special algorithm is used in MIDUSS to ensure both stability and conservation of mass.

You can now test this trial design and see the results of the Parking lot storage by pressing the [Route] button.

4.6.23 Results of parking lot Storage



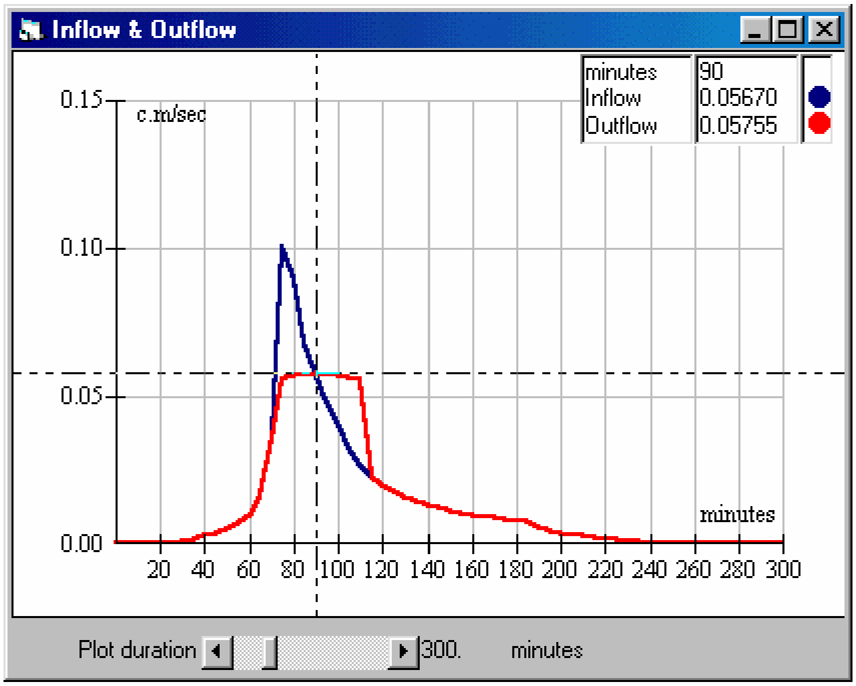
Results

Peak outflow	0.058	c.m/sec
Maximum level	100.105	metre
Maximum storage	24.9	c.m
Centroidal lag	1 h : 46	minutes

Figure 4-60 – Results of parking lot storage

The main results following the Route operation are shown in the "Results" frame in the Pond window. The maximum depth at the catchbasin is 0.015 m (6"). Peak flow has been reduced from 0.098 c.m/sec (3.5 cfs) to 0.059 c.m/sec (or 2.1 cfs) which is reasonable for four catchbasins under this modest level of surcharge.

From the depth and the defined grades the plan extent of the flooded elliptical areas will be close to 7.5m x 15m (24.5 ft x 49 ft) around each of the catchbasins. You can experiment by changing the grades or any of the discharge parameters until a satisfactory design is obtained.



minutes	90	
Inflow	0.05670	●
Outflow	0.05755	●

Plot duration 300. minutes

Figure 4-61 – Outflow hydrograph from the parking lot

The results are also indicated by a graph of the inflow and outflow hydrographs shown above and a table of the outflow hydrograph. The graph gives an indication of the duration of ponding on the parking surface - in this case less than four hours from start of rainfall.

4.6.24 Using Rooftop Storage

Large commercial buildings with flat roofs can provide an opportunity for on-site storage on the roof which can substantially reduce increased peak flow resulting from development. Flow from the roof is controlled by special devices which limit the outflow per roof drain to roughly 6 US gallons/minute for each inch of head (or 24 litres/minute for each 25mm of head).

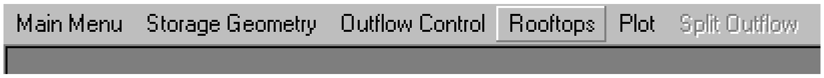


Figure 4_62 – Selecting Rooftop storage from the Pond command menu

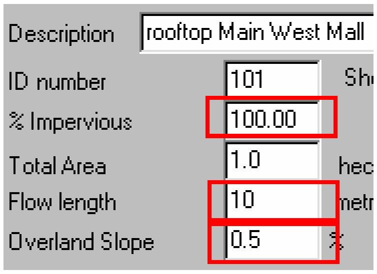
MIDUSS offers a Rooftop Storage option within the Pond command. The procedure is described in the following steps or topics.

- Generating the Rooftop Inflow
- Target Outflow from the Roof
- Desired Depth Range on the Roof
- Parameters for the Rooftop System
- Rooftop Discharge and Storage Characteristics
- Rooftop Flow Routing
- Graphing Rooftop Runoff
- Design Tips for Rooftop Storage
- Rooftop Error Messages

The first step is to generate the flow on the rooftop. This must be done using the **Hydrology / Catchment** and **Hydrograph / Add Runoff** commands to generate the Rooftop Inflow before you invoke the Pond command.

4.6.25 Generating the Rooftop Inflow

To create the inflow hydrograph the total building roof area must be modelled using the **Hydrology/Catchment** command



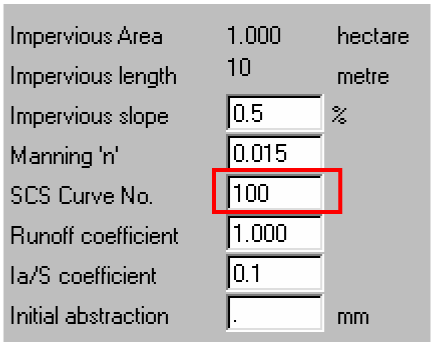
Description	rooftop Main West Mall	
ID number	101	Sh
% Impervious	100.00	
Total Area	1.0	hec
Flow length	10	metr
Overland Slope	0.5	%

Figure 4-63 – Defining catchment data for rooftop runoff

The total area contributing to runoff on the roof will normally be equal to the building footprint. The example shows an area of 10,000 sq.m or about 108,000 sq.ft which is typical for a modest "big box" store or mall building.

Three parameters which you must set carefully are outlined. These are:

- The percent of impervious area which will normally be 100% (i.e. no roof gardens).
- The flow length which will be quite short and roughly equal to half the square root of the area tributary to each roof drain (e.g. 10 m for a 400 sq.m area).
- The gradient which typically should be less than 1%.



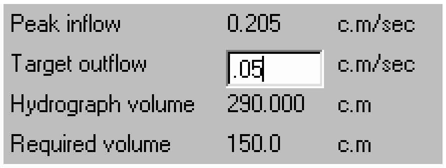
Impervious Area	1.000	hectare
Impervious length	10	metre
Impervious slope	0.5	%
Manning 'n'	0.015	
SCS Curve No.	100	
Runoff coefficient	1.000	
Ia/S coefficient	0.1	
Initial abstraction	.	mm

Figure 4-64 – Defining rainfall loss parameters for rooftop runoff

In addition, it is important to set the infiltration parameters for the impervious fraction to generate a runoff coefficient of 1.0. In the case illustrated in Figure 4-64, setting the SCS Curve Number to 100 automatically sets the initial abstraction to zero. For the Horton or Green and Ampt methods the surface depression storage can be explicitly set to zero or to some very small value.

After accepting the catchment results you must use the **Hydrograph/Add Runoff** command to move the runoff into the Inflow hydrograph. If you haven't already done so you should clear the Inflow hydrograph by means of the **Hydrograph/Start/New Tributary** command.

4.6.26 Target Outflow from the Roof



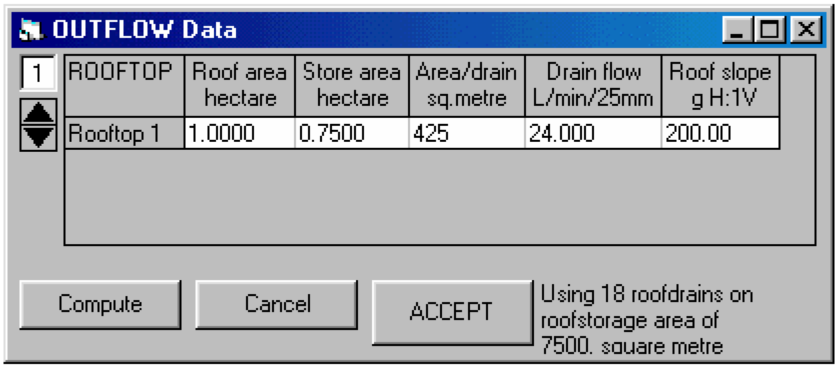
Peak inflow	0.205	c.m/sec
Target outflow	.05	c.m/sec
Hydrograph volume	290.000	c.m
Required volume	150.0	c.m

Figure 4-65 – If desired, you can set the target outflow from the roof.

You may specify a desired peak outflow from the roof and MIDUSS will display an approximate estimate of the required volume. However, since both the area available for storage and the maximum depth is limited you will probably need to refine the design by trial and error.

Before generating the table of depth, discharge and storage you must next set the desired range of depths to be used.

4.6.27 Parameters for the Rooftop System



ROOFTOP	Roof area hectare	Store area hectare	Area/drain sq.metre	Drain flow L/min/25mm	Roof slope g H:1V
Rooftop 1	1.0000	0.7500	425	24.000	200.00

Using 18 roofdrains on roofstorage area of 7500. square metre

Figure 4-66 – Data required for rooftop storage (metric units)

As with the other Pond data entry forms, there is initially no row opened on the grid. You must press the 'Up' arrow on the spin button to display the default data. You can display more than one row but normally only one rooftop can be designed at a time since the inflow to each roof must be generated by use of the Catchment command. However, if the roof drainage is complex with different sections having different densities of roof drain or roof slope you may want to use multiple roof sections.

Five values are displayed one of which cannot be varied.

1. **Roof area** is the total building footprint and is taken from the previous Catchment command. You cannot change this at this point.

2. **Storage area** is the roof area available for storage and is typically about 75% of the total area and this is shown as the default. However you can change this if you wish. In Figure 4-66 the value has been left at 75% of 1.0 ha.

3. **Area per Drain** is usually recommended by the manufacturer of the roof drain. A typical value is around 425 sq.m. (4500 sq. ft.).

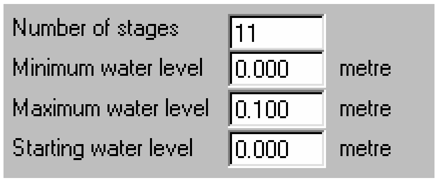
4. **Drain flow rate**. This is the flow capacity of the drain as a function of the head or depth at the drain inlet. This is also specified by the manufacturer and is often very close to a linear function of the depth. You should provide a value in litres per minute for each 25mm of depth. When using U.S. Customary units this will be measured in U.S. gallons per minute for each inch of depth.

5. **Roof slope**. Initially this is set to the gradient used in the Catchment command but expressed as a ratio xH:1V (e.g. 0.67% is shown as 150H:1V). You can change this value to see the sensitivity of the results to using a flatter or dead flat roof.

The line informing you of the number of roof drains required is added only after you press the [Compute] button.

With this data you can now generate the discharge and storage characteristics of the rooftop.

4.6.28 Desired Depth Range on the Roof



Number of stages 11
Minimum water level 0.000 metre
Maximum water level 0.100 metre
Starting water level 0.000 metre

Figure 4-68 – MIDUSS estimates default values for the depth range.

MIDUSS automatically calculates a maximum depth which would provide storage equal to the total volume of the inflow hydrograph. The Rooftop storage data window is the only pond data window that will trigger an automatic calculation of the minimum and maximum depths over in the Pond Design window.

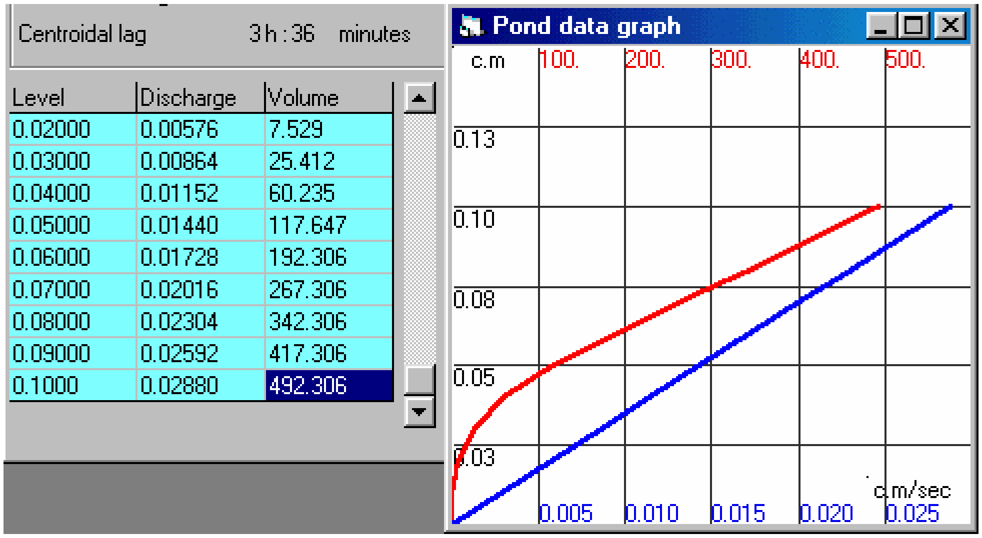
The automatically calculated depths are rounded up to a multiple of a convenient depth increment such as 25mm or 1 inch. In Figure 4-68, the minimum depth is set at zero and 11 depths are defined to give 10 depth increments of 0.01m.

You can change these values if you wish. The computed defaults are provided as a guide.

Note that the suggested value of maximum depth takes into account the flow length and gradient used in the Catchment command.

You can now invoke the Rooftop option to set the necessary parameters for the Rooftop Storage system.

4.6.29 Rooftop Discharge and Storage Characteristics


Centroidal lag 3h:36 minutes

Level	Discharge	Volume
0.02000	0.00576	7.529
0.03000	0.00864	25.412
0.04000	0.01152	60.235
0.05000	0.01440	117.647
0.06000	0.01728	192.306
0.07000	0.02016	267.306
0.08000	0.02304	342.306
0.09000	0.02592	417.306
0.1000	0.02880	492.306

Figure 4-69 – Discharge and storage characteristics for rooftop storage.

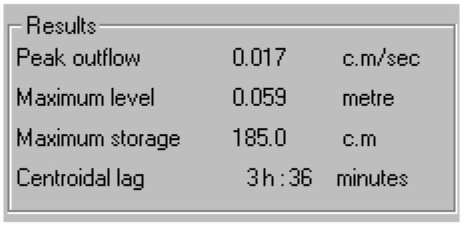
Pressing the [Compute] button on the Rooftop data table causes the usual table of depth, discharge and storage volume values to be displayed as shown on the left of Figure 4-69. The method used by MIDUSS to estimate the necessary maximum depth will usually provide an adequate margin of safety

The Plot option in the Pond menu will show that the discharge (in blue) is assumed to be a linear function of depth. The storage (in red) is nonlinear because the area tributary to each roof drain will normally be shaped like an inverted pyramid since a finite drainage slope must normally be provided.

You can experiment by changing the rooftop parameters. When you press the [Accept] button the highlight on the maximum volume is removed.

You can now press the [Route] button to complete the Rooftop flow routing operation.

4.6.30 Rooftop Flow Routing


Results

Peak outflow	0.017	c.m/sec
Maximum level	0.059	metre
Maximum storage	185.0	c.m
Centroidal lag	3h:36	minutes

Figure 4-70 – Results of routing the rooftop runoff

The [Route] button is enabled after you accept the discharge and storage data generated in the previous step. The result of the flow routing is displayed:

- In the "Results" frame of the Pond window,
- As a graph of inflow and outflow, and

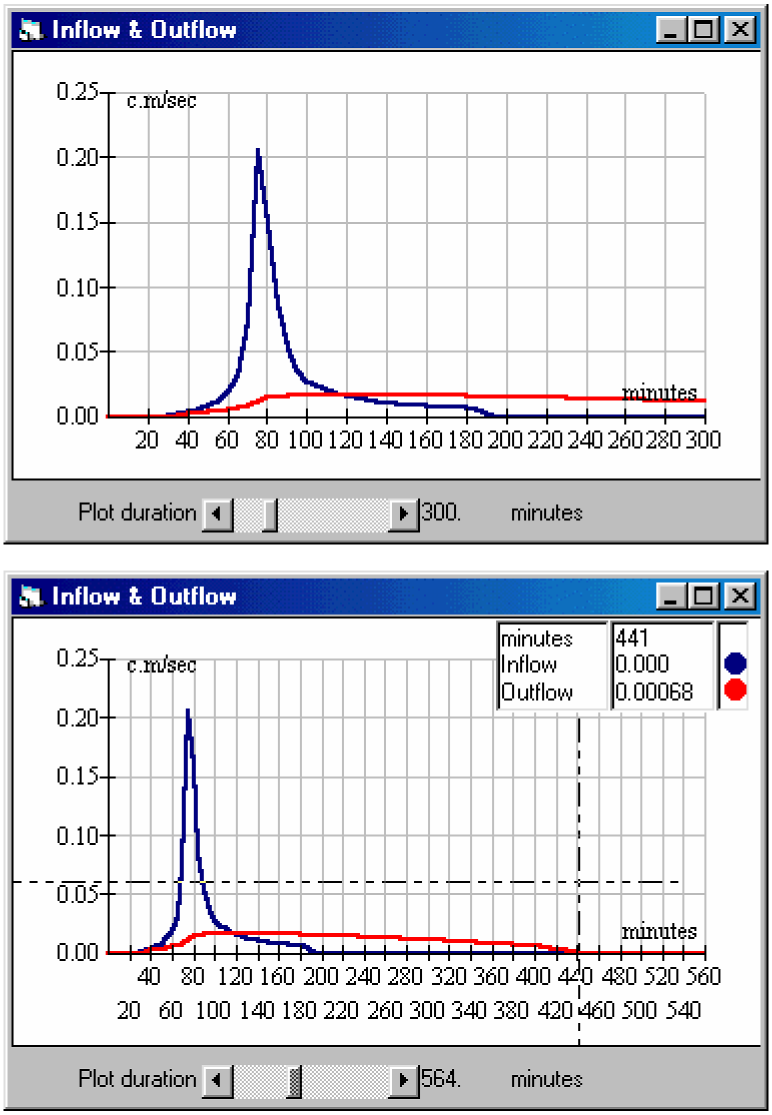
- As a tabular display of the outflow hydrograph.

The results shown in Figure 4-70 give the peak outflow, the maximum depth, the maximum storage volume and the lag to the centroid of the outflow from the start of rainfall. Note that in Figure 4-70 the peak outflow value of 0.22 is highlighted and is not a text box which you can edit.

You can also see how quickly (or slowly) the roof storage drains by experimenting with the Graph of Inflow and Outflow.

4.6.31 Graphing Rooftop Runoff

The graph of inflow and outflow is displayed in the normal way and you can explore this with the mouse pointer as illustrated here. If you want to check the duration of ponding you can extend the time base of the graph by clicking on the slider control as shown below.



Plot duration 300. minutes

minutes	441	
Inflow	0.000	●
Outflow	0.00068	●

Plot duration 564. minutes

Figure 4-71 – You can experiment with the hydrograph plot to see the drainage time

Before concluding this section you may find it useful to review some design tips for using rooftop storage.

4.6.32 Design Tips for Rooftop Storage

The structural integrity of the roof must be considered. In areas subject to potentially severe snowfall, the code requirements for carrying live load or snow load will provide load bearing capacity for up to 4 inches (100mm) of water or more. Some codes which limit maximum depth to this amount may allow an extra 25% to compensate for the grading of the roof around the roof drains.

Building codes often require a drainage slope of 2% but this is excessive and severely limits the available storage. If rooftop storage is to be used it is often possible to relax this requirement to slopes not less than 0.5%.

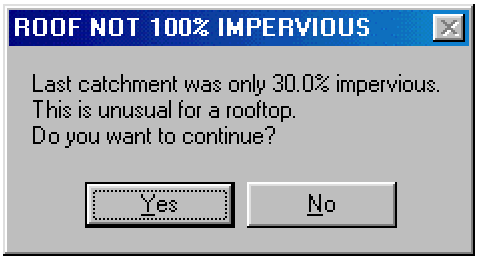
If the building is very wide it may be necessary to have roof drains remote from the perimeter of the building. This in turn requires provision of storm drains suspended below the roof (and above the suspended ceiling) which pick up one or more drains. It is very important that these drains be properly designed to avoid surcharge in the pipe which could drown out the control at the drain inlet.

Down-pipes from the roof drains will usually terminate in a 90 degree vertical bend at the bottom to connect with external services. Sometimes a maintenance access is provided inside the building to allow cleaning (e.g. rodding) of the drains. In such cases it is essential to secure the access cover to resist the high pressure which can develop in the down-pipe. Failure to do so can result in quite spectacular fountain displays to the severe displeasure of the tenant.

MIDUSS does some basic error checking and the error messages you may see are described in the next topic.

4.6.33 Rooftop Error Messages

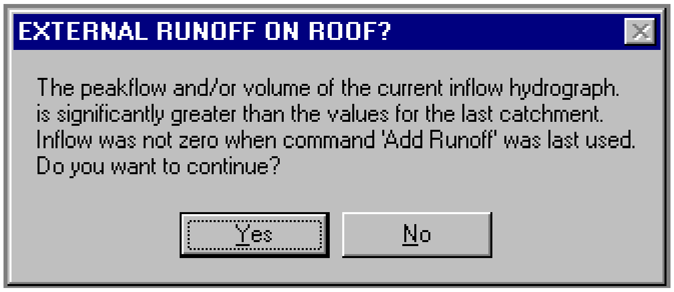
Design of a rooftop control system requires proper use of three commands - the Catchment command, the Add Runoff command and finally the Pond command. MIDUSS performs some basic checks to warn you if there appears to be some inconsistency in the data. Three cases are described below.


ROOF NOT 100% IMPERVIOUS

Last catchment was only 30.0% impervious.
This is unusual for a rooftop.
Do you want to continue?

Figure 4-72 – Warning message for bad % impervious value

The percent of impervious area in the Catchment command will normally be 100%. If you have forgotten to set this properly the warning lets you cancel out of the Pond command and re-define the catchment. It is possible - although unlikely - that the roof may have a significant area of landscaping or roof-garden in which case you can bypass this warning and proceed with the design.

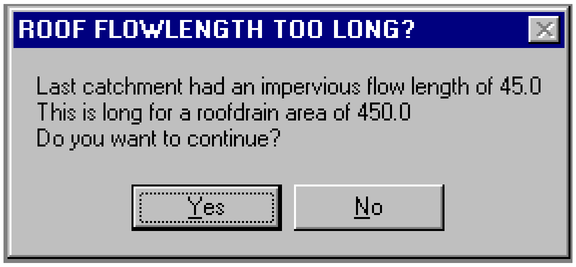


EXTERNAL RUNOFF ON ROOF?

The peakflow and/or volume of the current inflow hydrograph.
is significantly greater than the values for the last catchment.
Inflow was not zero when command 'Add Runoff' was last used.
Do you want to continue?

Figure 4-73 – Warning if Inflow has not been set to zero

The Catchment command must be used for each individual roof area and the Inflow hydrograph must be set to zero before using the Add Runoff command. MIDUSS checks to ensure that this has been done and shows this warning if either the peak flow or volume is inconsistent. However, in unusual circumstances, it is possible that external runoff may be intentionally directed on to the roof in which case the warning can be ignored.



ROOF FLOWLENGTH TOO LONG?

Last catchment had an impervious flow length of 45.0
This is long for a roofdrain area of 450.0
Do you want to continue?

Figure 4-74 – Warning for bad flow length value.

The flow length on the impervious area should be relatively short and will normally be less than the square root of the area contributing to each roof drain. For example if the area per drain is 5000 sq.ft the length would normally be between 40 and 70 ft. The illustration shows an equivalent case in metric units.

4.6.34 Accepting the Pond Design

When you are satisfied with the design click on the [Accept] button on the Pond window. All currently open windows will be closed and the results will be copied to the current Output file.

You may find it useful to assemble a number of windows showing principle data, plots of storage and discharge, etc. and print a hardcopy of the results by use of the **Files/Print** command.

4.7 Diversion Structure Design

The purpose of this command is to split the inflow hydrograph into two components. The diversion structure is assumed to be similar to a side discharging weir or a catchbasin in a road.

In a **side-weir** there is a threshold flow below which all of the inflow is transmitted downstream as outflow. Once the inflow is greater than the specified threshold flow the excess flow (i.e. inflow - threshold) is divided to two parts. One fraction *F* of the excess flow is diverted to a different watercourse and the balance **(***1-F***)** is transmitted downstream as part of the outflow.

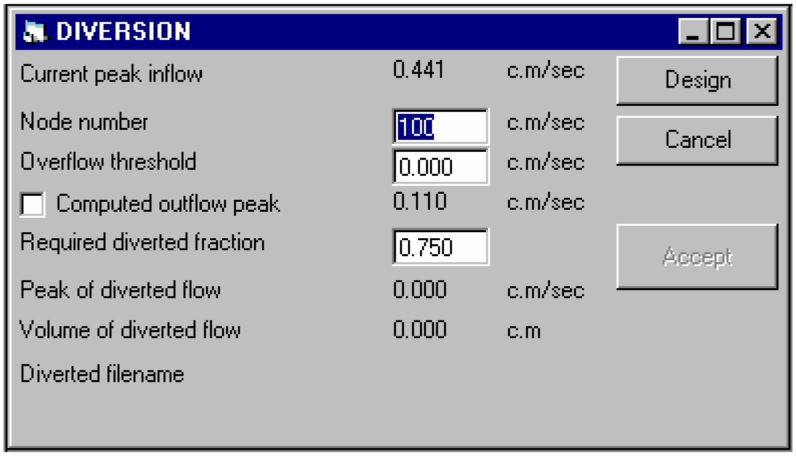
In the case of a **catchbasin** the inflow is the runoff in the gutter. At low inflow rates all of the inflow is captured by the catchbasin and is conveyed as outflow to the minor system. When the inflow reaches a certain threshold value some fraction *F* of the excess flow bypasses the catchbasin and is 'diverted' to the major system flow. The balance **(***1-F***)** is added to the outflow captured by the minor system.

The design process is described in the following steps.

- The Diversion Window
- Defining the Diversion Node Number
- Defining the Threshold Flow
- Designing for a specified Maximum Outflow from the Diversion
- The Diverted Fraction
- Results of the Diversion Design
- Graphing the Diversion Flows
- Accepting the Diversion Design

The first step is to open the Diversion window and review the current default data.

4.7.1 The Diversion Window



Current peak inflow	0.441	c.m/sec
Node number	100	c.m/sec
Overflow threshold	0.000	c.m/sec
☐ Computed outflow peak	0.110	c.m/sec
Required diverted fraction	0.750	
Peak of diverted flow	0.000	c.m/sec
Volume of diverted flow	0.000	c.m
Diverted filename		

Figure 4-75 – Data required to define a diversion structure

When the Diversion window is displayed for the first time the initial display shows the current peak inflow, an initial threshold flow of zero and a diversion fraction of 0.75. In the illustration the peak inflow is 0.441. If the overflow threshold is zero then the diverted fraction *F* is applied to the total inflow so that the outflow is **(***1-F***)** x 0.441 which is approximately 0.110.

Frequently the Diversion command is used to split the inflow hydrograph into a minor component that is carried in the drainage conduit and a major component that is conveyed on the surface such as the roadway. If you use the Diversion command to separate the inflow before a pipe or channel design is attempted, MIDUSS will remind you that you can cancel the Diversion command and design a conduit if you wish.

You have not yet designed a Pipe or Channel with finite capacity. If you want to design a conduit first press the [Cancel] command in the Diversion window.

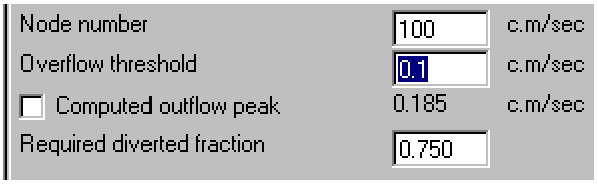
Figure 4-75a – Diversion message if no conduit has been designed.

4.7.2 Defining the Diversion Node Number

The node number serves to define the location of the diversion and is also embedded into the name of the file which will hold the diverted hydrograph. You can use any 5 digit integer "nnnnn" from which a filename of the form **DIVnnnnn.hyd** is formed.

The default number is the nearest upstream Catchment ID number.

4.7.3 Defining the Threshold Flow



Node number 100 c.m/sec
Overflow threshold 0.1 c.m/sec
☐ Computed outflow peak 0.185 c.m/sec
Required diverted fraction 0.750

Figure 4-76 – The diverted fraction can be defined explicitly

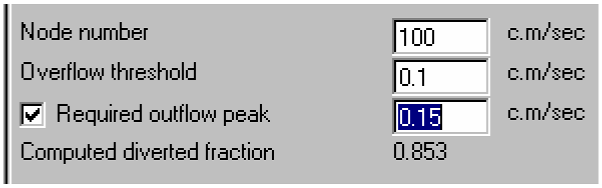
When you enter a finite value for the threshold flow the Diverted Fraction is applied to the new excess flow and the peak outflow is recalculated and updated. In the sample shown here, a threshold flow of 0.1 reduces the excess flow to 0.341 of which 75% is diverted and 25% x 0.341 or 0.085 is added to the overflow of 0.1 to give a total outflow peak of 0.185.

You can of course change the value of the Diverted Fraction to any fraction in the interval 0.0 to 1.0.

At this point you should also define the node number where the diversion structure is located. This is used to construct a file name which will contain a record of the diverted hydrograph for future use or reference.

An alternative approach is to design the diversion structure for a desired maximum outflow which is described in the next topic.

4.7.4 Designing for a Maximum Outflow from the Diversion



Node number 100 c.m/sec
Overflow threshold 0.1 c.m/sec
☑ Required outflow peak 0.15 c.m/sec
Computed diverted fraction 0.853

Figure 4-77 – You can specify the outflow to implicitly define the diverted fraction

If you click on the check box to the left of the label "Computed Outflow" several things happen. The label changes to "Required Outflow peak" and the value is displayed in a data entry box instead of a simple value. The label "Required diverted fraction" changes to "Computed diverted fraction", the value can no longer be edited in a data box and the computed value is displayed for information.

This lets you specify the maximum outflow that you would like to result from this design using the current inflow, threshold flow and diverted fraction value. When you enter the required outflow the value of the diverted fraction is adjusted to produce the required result with the current threshold flow. You can alter the threshold flow to see the effect of this on the computed diverted fraction.

When you are satisfied with this trial design you can press the [Design] button to see the results of the diversion design.

4.7.5 The Diverted Fraction

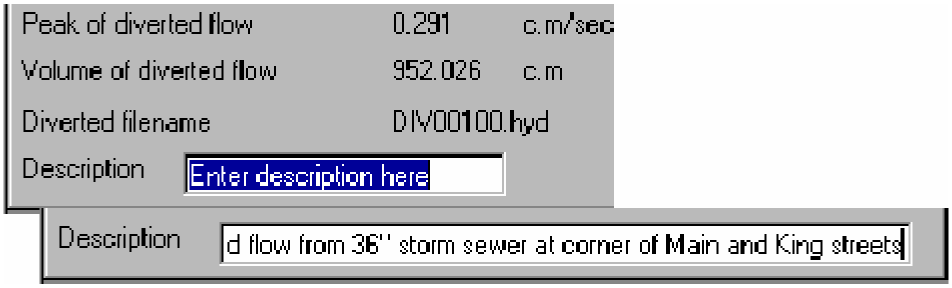
If the checkbox against "Computed outflow peak" is unchecked, the text-box for this data item is displayed and you can enter a fractional value in the range 0.0 to 1.0. This represents the fraction of the excess flow which is diverted where excess flow is (Inflow - Threshold flow).

The value is very dependent on the type of diversion. MIDUSS uses an initial default of 1.00.

Note that if the checkbox is ticked, you will be prompted to define the peak outflow and the diverted fraction will be computed and displayed for information.

4.7.6 Results of the Diversion Design

The calculation applies a simple continuity rule to the design and computes the peak flow and volume of the diverted hydrograph. Also displayed is the name of the hydrograph file to which the diverted hydrograph will be written. However, this is not done until you press the [Accept] button and until then you can refine the design if desired.

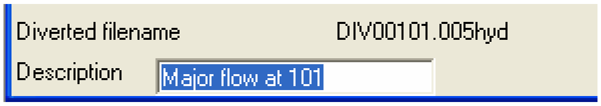


Peak of diverted flow 0.291 c.m/sec
Volume of diverted flow 952.026 c.m
Diverted filename DIV00100.hyd
Description Enter description here
Description d flow from 36'' storm sewer at corner of Main and King streets

Figure 4-78 – Results of the Diversion command

Also shown in Figure 4-78 is a text entry box which prompts you to type in a brief description of the diverted hydrograph which will be incorporated in the header of the file containing the diverted hydrograph. As shown above, the text box expands and the text scrolls left to allow any reasonable length of description.

A short default description is displayed as shown below but you can easily delete or replace this.

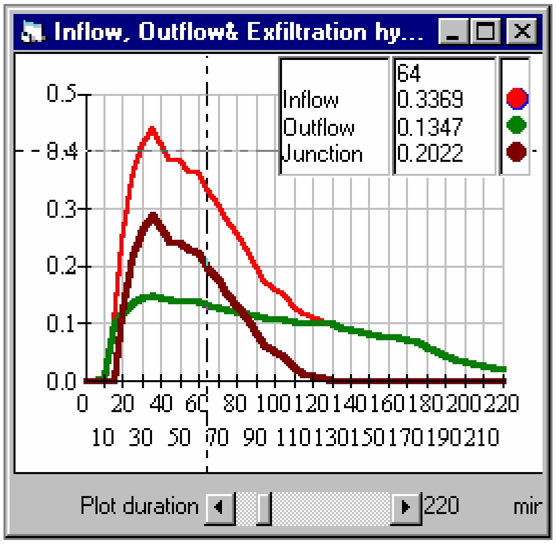


Diverted filename DIV00101.005hyd
Description Major flow at 101

Figure 4-78a – Description of the Diversion file.

In addition to the results shown in the Diversion window, the diverted hydrograph is shown in tabular form and the Graph display shows the three hydrographs of interest as described in the next topic.

4.7.7 Graphing the Diversion Flows



64
Inflow 0.3369
Outflow 0.1347
Junction 0.2022

Figure 4-79 – Graphical result of the Diversion command.

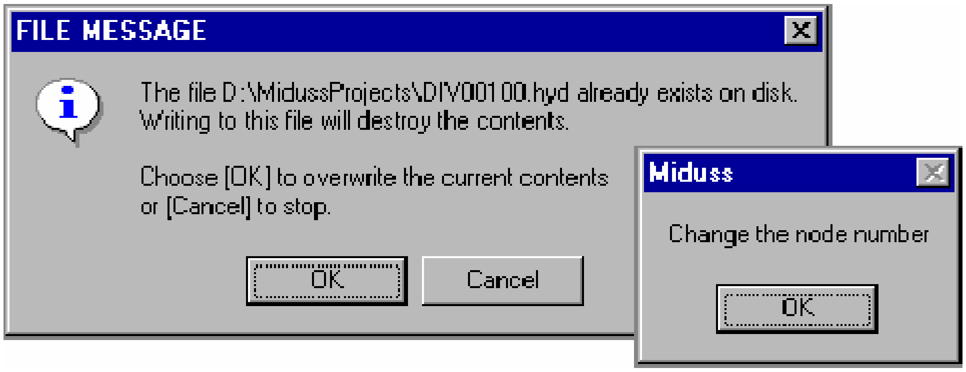
The diverted hydrograph for the current trial design is written to the Temporary or junction hydrograph and this is displayed along with the inflow and outflow hydrographs. The illustration shows the cursor over the graph with the three numerical values displayed in the top right corner of the graph. The inflow can be seen to be equal to the sum of the outflow and the diverted flow within the accuracy of the display.

If you are satisfied with the design the final step is to accept the Diversion design by pressing the [Accept] button.

4.7.8 Accepting the Diversion Design

Before pressing the [Accept] button you should type in a brief description in the text box labeled "Description".. The text box will expand to hold a reasonable length of description which will be included in the header of the file holding the diverted hydrograph.

The diverted hydrograph is written to the file only after the [Accept] button is pressed, however, MIDUSS checks to see if a file of the same name already exists in the current job directory. If it does you will see the warning shown below.



The file D:\MidussProjects\DIV00100.hyd already exists on disk. Writing to this file will destroy the contents.

Choose [OK] to overwrite the current contents or [Cancel] to stop.

Change the node number

Figure 4-80 – MIDUSS protects you from overwriting a previous diversion hydrograph

If you don't want to overwrite the existing file you can press the [Cancel] button and the [Accept] command is aborted.

You will be prompted to change the node number which changes the file name and you can try again.

4.8 Exfiltration Trench Design

An exfiltration trench is a device which promotes the distribution and return of stormwater runoff into the soil and thus eventually to replenish the groundwater. It is common to see the term infiltration applied to facilities of this type. However, since inflow and infiltration (I & I) generally describes the increase of hydraulic load in a sanitary sewer system resulting from infiltration of water, the term ***exfiltration*** will be used in this section to describe the type of trench used for underground disposal of stormwater runoff.

In general, the trench causes an inflow hydrograph to be split into two components:

(1) an outflow hydrograph which continues down the storm sewer (if one exists), and

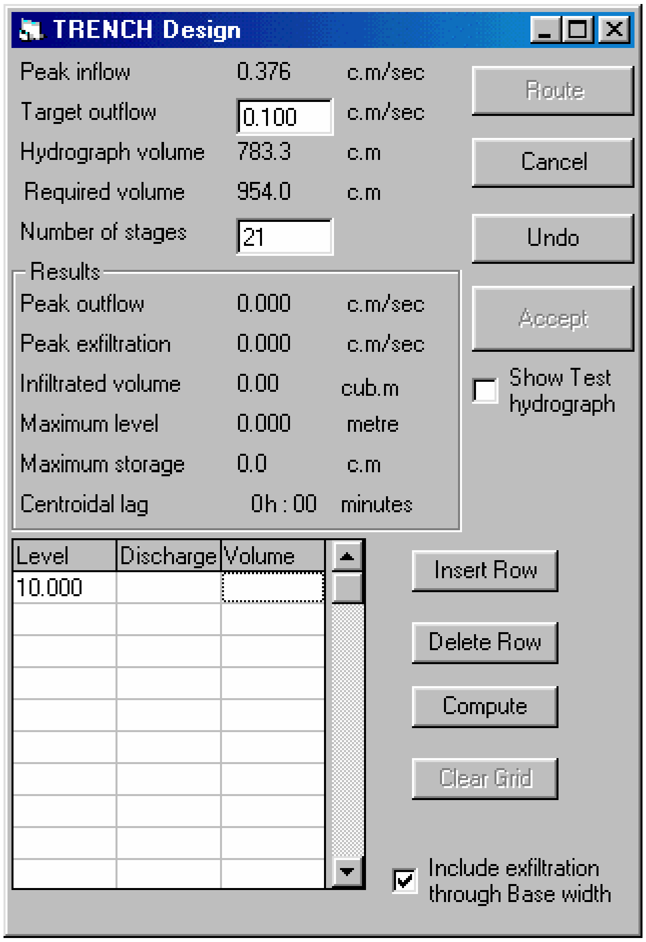
(2) the flow which is directed into the soil.

A comprehensive manual on the subject was published in 1980 by the U.S. Dept. of Transportation (see References) which describes devices such as porous pavement, basins, wells and trenches.

MIDUSS assumes the exfiltration trench to comprise a trench of rectangular or trapezoidal cross-section containing one or more perforated pipes and possibly a conventional non-perforated storm sewer surrounded by clear stone fill with a relatively high voids ratio of around 40%. To increase exfiltration the system usually has some form of outflow control device to promote high water levels within the trench.

The device can be visualized as an underground detention pond from which lateral flow is possible in addition to the downstream outflow.

4.8.1 Overview of Trench Design



Peak inflow 0.376 c.m/sec
Target outflow 0.100 c.m/sec
Hydrograph volume 783.3 c.m
Required volume 954.0 c.m
Number of stages 21

Results
Peak outflow 0.000 c.m/sec
Peak exfiltration 0.000 c.m/sec
Infiltrated volume 0.00 cub.m
Maximum level 0.000 metre
Maximum storage 0.0 c.m
Centroidal lag 0h : 00 minutes

☐ Show Test hydrograph

Level	Discharge	Volume
10.000		

☑ Include exfiltration through Base width

Figure 4_81 – The initial exfiltration Trench form

The **Design/Trench** command helps you to determine the proportions and discharge characteristics of an exfiltration trench with the object of attenuating the peak flow in a link of a drainage network to an acceptable value. As with the other design commands in this chapter, the design comprises four general steps.

1. Specify the desired peak outflow and determine an approximate storage volume (including the stone fill) which might achieve this.
2. Set up a table to define the Depth - Discharge and Depth -Storage volume relationships for the proposed trench. These functions must increase monotonically. MIDUSS contains a number of special commands to help you to define the Depth - Discharge - Storage values for a variety of standard outflow control devices and a range of geometric and physical properties of the trench system.
3. Route the inflow hydrograph through the proposed trench and get peak flow and volume of outflow and exfiltration to the soil as well as maximum depth in the trench.

4. Accept or reject the design. If rejected, edit the Depth Discharge and Storage table and/or the physical characteristics of the trench and repeat step (3) until an acceptable design is achieved.

Many of the options available in the Trench command are accessed through a special menu which is enabled when the **Design/Trench** command is selected.

A logical series of steps is summarized in the following list of topics. You may use these in an order different from that shown but the sequence illustrated here is typical.

- Set a Target Outflow for the Trench
- Set the Number of Stages in the Trench
- The Trench Geometry menu
- Trench Data
- Checking the Trench Volume
- The Trench Outflow Control
- Setting a Weir Control for the Trench
- Setting an Orifice Control for the Trench
- Setting a Pipe Control for the Trench
- Defining a Pipe in the Trench
- Defining a Pipe in the Trench
- Plotting the Trench Properties
- Routing Flow through the Trench
- Results of Trench Routing

4.8.2 The Trench Menu

Figure 4-82 – The Trench menu

When the Trench window is displayed (and has the focus as indicated by the coloured title bar) a special menu replaces the main MIDUSS menu. It includes the following items most of which are discussed in more detail in the items which follow.

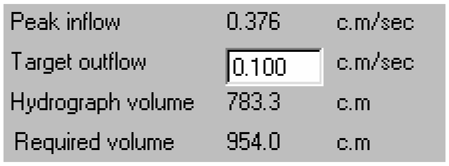
Main Menu	Lets you return to the main MIDUSS menu or use the Clipboard controls.
Geometry	Displays items to define the geometry and other physical characteristics of the trench and also set the size, position and type of any pipes in the trench
Outflow Control	Contains items to let you compute the stage-discharge values for one or more orifice controls, weir controls or outflow pipes. An outflow pipe can be located:

	(a) within the length of the trench (flow enters from the upstream manhole, or
	(b) downstream of the trench (flow enters at the downstream manhole)
Plot	Lets you display various types of plot involving the depth (or stage), discharge, exfiltration flow and storage volume.

Sometimes when the Trench window loses the focus to another window (such as a graph display or a data entry form) the main MIDUSS menu is displayed in the menu bar. To see the Trench menu you need only move the mouse pointer over the Trench form.

4.8.3 Specify Target Outflow from Trench

MIDUSS arbitrarily sets the target outflow to 20% of the peak inflow. You will usually want to change this. Click on the text box at “Target outflow” to highlight the current value and type in the desired peak outflow which you would like to achieve for this hydrograph.



Peak inflow	0.376	c.m/sec
Target outflow	0.100	c.m/sec
Hydrograph volume	783.3	c.m
Required volume	954.0	c.m

Figure 4-83 – Setting the target outflow

As shown in Figure 4-83, an approximate estimate of the required trench storage (including the stone fill) is displayed for information along with the total volume of the inflow hydrograph. The storage required will increase as the desired peak outflow is reduced.

4.8.4 Set Number of Stages for Trench

The default number of stages or water levels used to describe the Depth - Discharge - Storage functions is set at 21. If you have changed this at an earlier stage in the MIDUSS session the latest value will be displayed.

Number of stages 21

Figure 4-84 – You can use up to 50 stages

If you want to change the current value, click on the text box to highlight the current value. Type in a new value. The number of stages must be at least 3 and not more than 50. The default value of 20 depth increments will normally be sufficient.

4.8.5 The Trench Geometry Menu

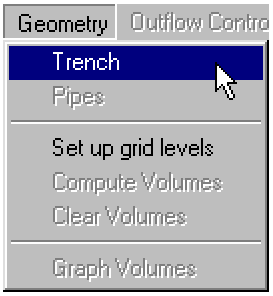
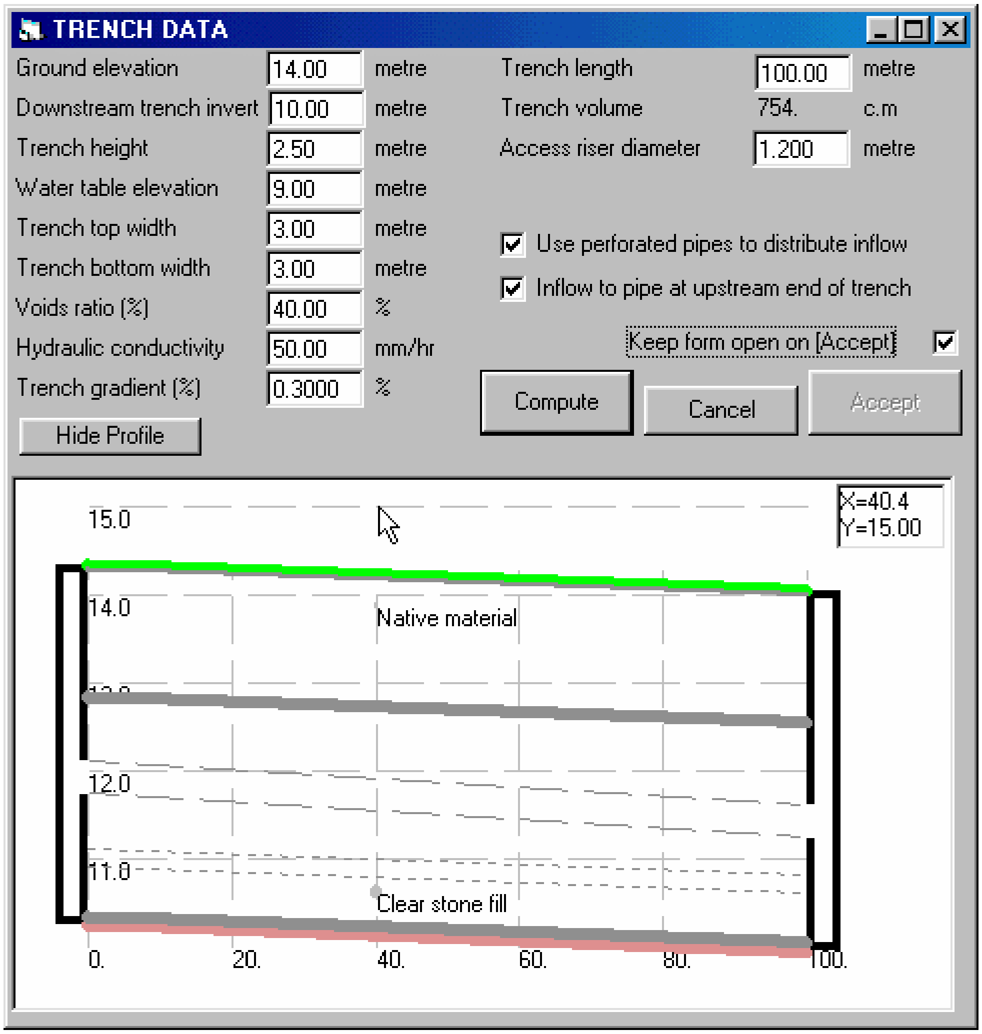


Figure 4-85 – The first step is to define the Trench geometry

You will use the Trench and Pipes options from this menu but only Trench is enabled since it is not possible to define and locate a pipe until the containing trench has been described.

Click on the Trench item as shown to open the Trench Data window.

4.8.6 Trench Data


TRENCH DATA

Ground elevation	14.00	metre
Downstream trench invert	10.00	metre
Trench height	2.50	metre
Water table elevation	9.00	metre
Trench top width	3.00	metre
Trench bottom width	3.00	metre
Voids ratio (%)	40.00	%
Hydraulic conductivity	50.00	mm/hr
Trench gradient (%)	0.3000	%
Trench length	100.00	metre
Trench volume	754.	c.m
Access riser diameter	1.200	metre

☑ Use perforated pipes to distribute inflow

☑ Inflow to pipe at upstream end of trench

Keep form open on [Accept] ☑

Figure 4-86 The Trench Data Form

The Trench Data window contains data entry boxes for a number of geometric and physical properties of the trench that must be specified at the start of the design process. After you have completed a trial design you may return to this window to change one or more of these properties.

The following quantities are shown with the current default values. Note that the 1st, 2nd and 4th items much be defined to the same datum. Also if the Top width and Bottom width are not the same the cross-section is assumed to be symmetrical.

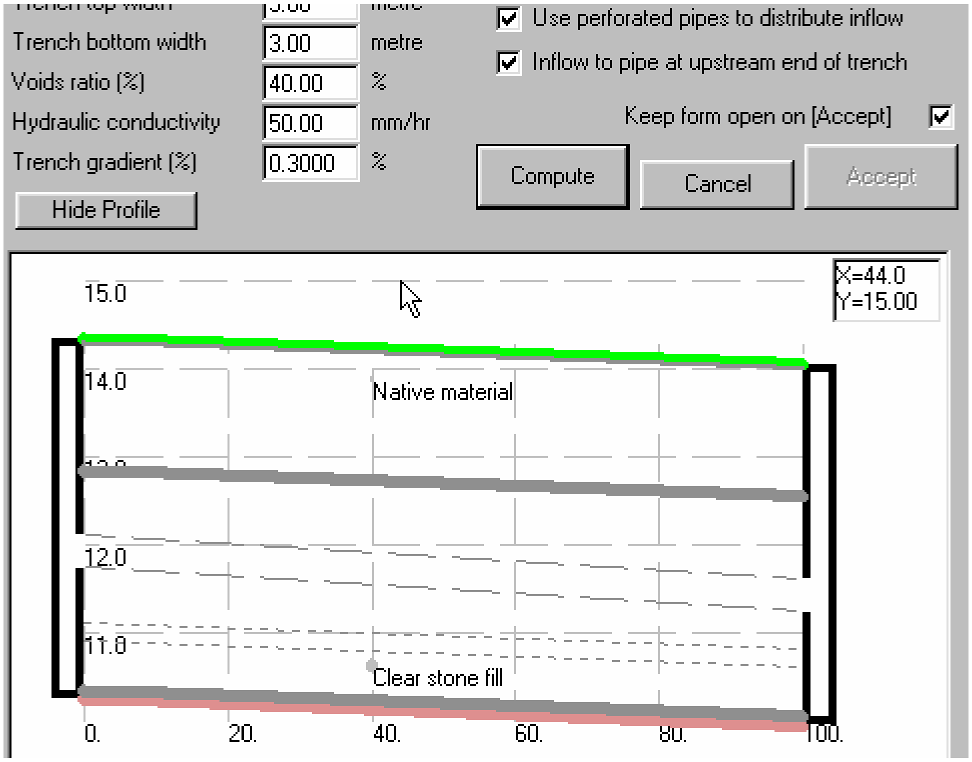
Ground elevation	Ground level at the downstream end of the trench
Trench Invert	Invert level of the trench (not the pipe) at the downstream end.
Trench Height	Total height of the stone filled trench at the downstream end
Water table elevation	Ground water elevation
Trench Top width	Width measures must be in the same units as the previous values
Trench Bottom width	Width measures must be in the same units as the previous values
Voids Ratio	Percentage of voids in the clear stone filling the trench.
Hydraulic Conductivity	Saturated hydraulic conductivity of the soil surrounding the trench
Trench Gradient	Gradient of the trench invert
Trench length	Total length of the trench.
Trench volume	Computed gross trench volume including the stone fill.

The trench volume is computed when the [Compute] button is pressed to allow comparison with the "Required trench volume" estimated when the target outflow is specified. Note that MIDUSS assumes the top water level in the stone fill to be horizontal. Thus a flatter trench gradient will provide more trench volume for storage in the voids.

When you have entered trial values for the trench data you can press the [Compute] button to check the Trench Volume.

The remaining data is defined in the next topic.

4.8.7 The Trench Profile



Trench bottom width	3.00	metre
Voids ratio (%)	40.00	%
Hydraulic conductivity	50.00	mm/hr
Trench gradient (%)	0.3000	%

☑ Use perforated pipes to distribute inflow

☑ Inflow to pipe at upstream end of trench

Keep form open on [Accept] ☑

X=44.0
Y=15.00

Fig 4-86a – Editing the Trench data form.

On the lower left of the Trench Data form the command button [Show profile] causes a graphic display of the current trench profile to be displayed. The command button is re-labeled [Hide Profile] to let you toggle this display. In selecting a design you can use two options selected by checking or clearing two checkboxes.

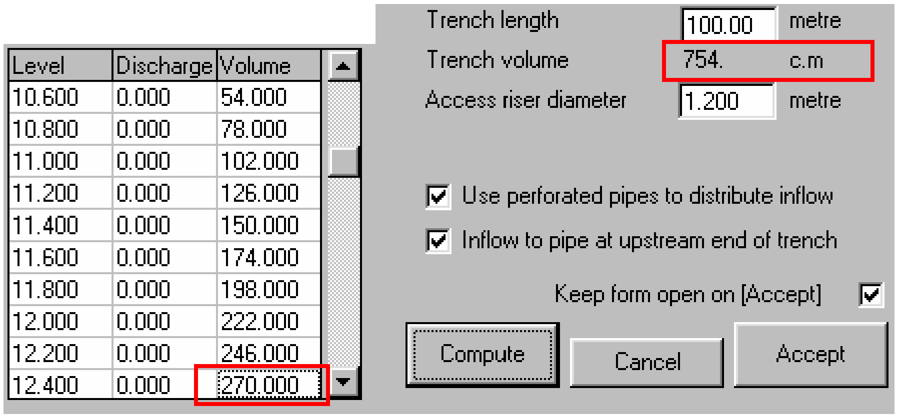
☑ Use perforated pipes to distribute inflow

This choice displays a perforated pipe over the full length of the trench but with a plug at the downstream end. The diameter, elevation and slope will be specified later; the illustration is only to provide an approximation of the design.

☑ Inflow to pipe at upstream end of trench

This choice is relevant only if you decide to use a pipe as part of the Outflow Control device. Selecting this choice shows a pipe connecting the upstream and downstream manhole chambers controlling the outflow from the trench. In most practical cases, if you decide to use either orifice or weir outflow controls you should leave this unchecked. When used, the pipe will normally form part of the minor drainage storm sewer. This type of design is often described as the Etobicoke trench after the area in Toronto, Ontario where this design was first used with considerable success.

4.8.8 Checking the Trench Volume



Level	Discharge	Volume
10.600	0.000	54.000
10.800	0.000	78.000
11.000	0.000	102.000
11.200	0.000	126.000
11.400	0.000	150.000
11.600	0.000	174.000
11.800	0.000	198.000
12.000	0.000	222.000
12.200	0.000	246.000
12.400	0.000	270.000

Trench length 100.00 metre
Trench volume 754. c.m
Access riser diameter 1.200 metre

☑ Use perforated pipes to distribute inflow
☑ Inflow to pipe at upstream end of trench
Keep form open on [Accept] ☑

Figure 4-87 – The Trench volume includes the stone fill

The maximum available trench volume (including the stone fill) is displayed in the Trench Data window when you press the [Compute] button. Some preliminary adjustment of the trench dimensions can be made at this point to provide a trench volume roughly the same as predicted in the Trench window. The predicted estimate is very approximate and it is not worth trying to match the trench volume with any precision.

In addition to showing the computed trench volume, the Volume column of the grid will be updated with the maximum volume shown at the front of the Volume column. Note that the grid values indicate the net available storage in the voids of the clear stone fill. For example, the maximum available volume of 301 c.m is 40% of the trench volume of 754 c.m.

Because of the trench gradient of 0.5 m the available volume in the voids at an elevation of 12.4 is reduced to 270.0 cu.m.

When you press the [Compute] button the [Accept] button is enabled. However, if you press the [Accept] button the Trench data form and profile graphic will be closed. If you want to keep this open, check the box labeled “Keep form open on Accept”.

In order to increase the available storage volume you can flatten the slope or increase the length or breadth of the trench. Because the water surface in the trench is assumed to be horizontal, trench lengths much in excess of 100 m or 325 ft are not very effective. It may be better to increase the trench width or split the trench into two separate reaches.

4.8.9 The Trench Outflow Control

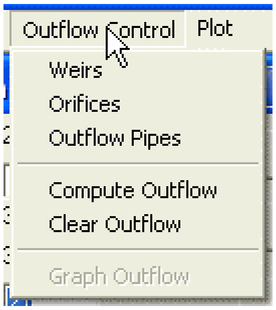
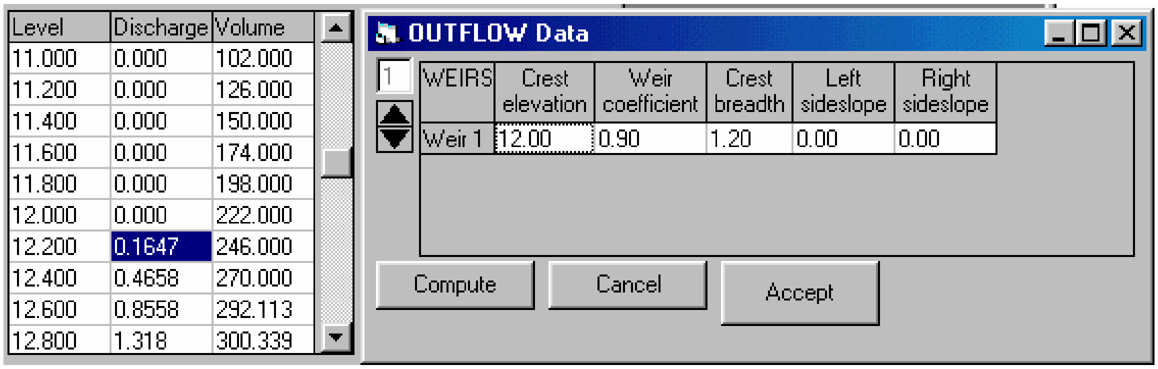


Figure 4-89 – The Trench Outflow control menu

The Outflow Control menu offers a number of options that let you design a weir, an orifice or an outflow pipe that controls the stage-discharge characteristic of the device. Up to 10 of each of these controls can be designed but normally only one or two will be required.

4.8.10 Setting a Weir Control for the Trench



Level	Discharge	Volume
11.000	0.000	102.000
11.200	0.000	126.000
11.400	0.000	150.000
11.600	0.000	174.000
11.800	0.000	198.000
12.000	0.000	222.000
12.200	0.1647	246.000
12.400	0.4658	270.000
12.600	0.8558	292.113
12.800	1.318	300.339

1

WEIRS	Crest elevation	Weir coefficient	Crest breadth	Left sideslope	Right sideslope
Weir 1	12.00	0.90	1.20	0.00	0.00

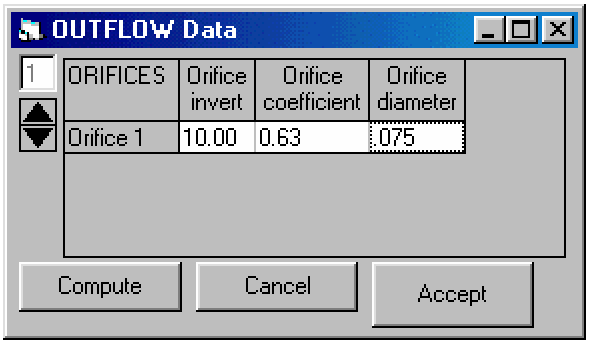
Figure 4-90 – Setting a weir for the Trench Outflow control

The table for specifying weirs or orifices is similar to that used for a regular detention pond. You must press the Up-arrow of the Spin button to open a row for the first weir. MIDUSS sets initial default values for crest elevation, discharge coefficient, crest breadth and side slopes which will pass the target outflow with reasonable ease. These default values can be edited by clicking on the appropriate cell and typing the required value. In Figure 4-90 no changes have been made from the default values.

When the data have been entered press the [Compute] button to generate the Discharge column in the grid of Depth - Discharge - Volume data. Only the cells corresponding to a depth greater than the weir crest elevation will have a finite discharge value.

Cells corresponding to levels below the weir crest will still have zero discharge.

4.8.11 Setting an Orifice Control for the Trench



1

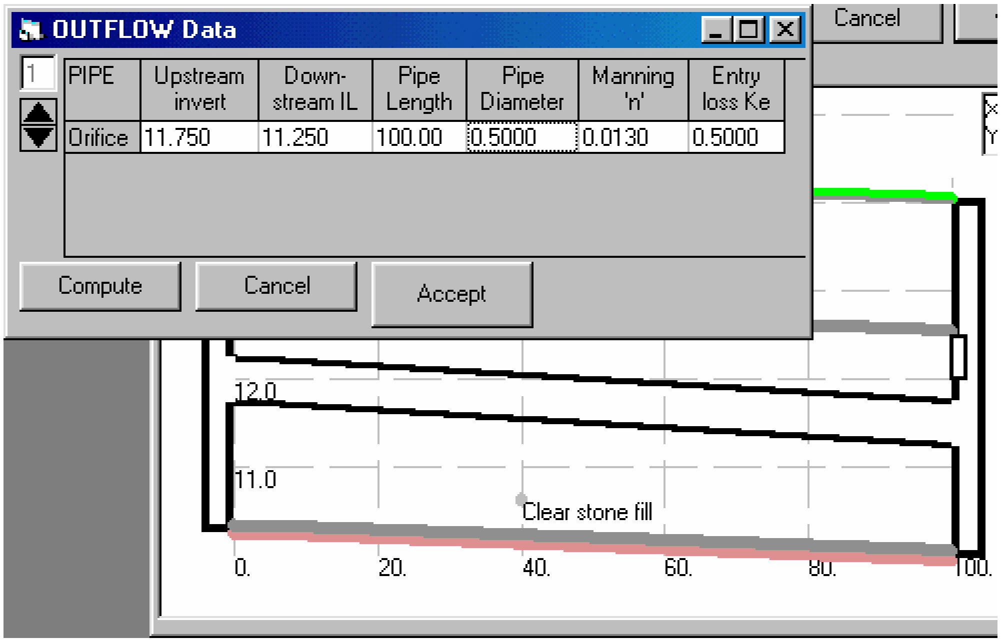
ORIFICES	Orifice invert	Orifice coefficient	Orifice diameter
Orifice 1	10.00	0.63	.075

Figure 4-92 – Setting the Orifice data

The default data provided by MIDUSS assumes an invert level at the bottom of the trench and a discharge coefficient of 0.63. It is likely that you will want to reduce the orifice diameter. Here a small orifice of 75 mm (3”) has been used.

Pressing the [Compute] button causes the vector of discharges to be updated. You can, of course, try any number of data values.

4.8.12 Setting a Pipe Control for the Trench



PIPE	Upstream invert	Down-stream IL	Pipe Length	Pipe Diameter	Manning 'n'	Entry loss Ke
Orifice	11.750	11.250	100.00	0.5000	0.0130	0.5000

Figure 4-91 – Setting the Pipe control data

Before selecting the **Outflow Control / Outflow Pipe** menu command you should indicate your choice by checking or clearing the checkbox labeled [ ] ‘Inflow to pipe at upstream end of trench’.

When you select **Outflow Pipe** a data entry form with 6 columns is displayed as shown in Fig 4-91. The estimates of upstream and downstream invert level are initial approximations, the length is set equal to the trench length and a diameter is selected based on the specified target outflow. Roughness and inlet energy loss coefficient Ke are fairly standard. When you press [Accept] the pipe is drawn on the Trench Data window.

You can experiment with these values depending on what type of outflow pipe control you have selected. Pressing [Compute] will update the discharge values for levels greater than the upstream invert level. Pressing [Accept] closes the form unless the selection **Keep form open on (Accept)** is checked ‘on’.

4.8.13 Defining Other Pipes in the Trench

Pipe geometry is defined using the second item in the Geometry menu. This item is enabled only after the general trench data has been specified.

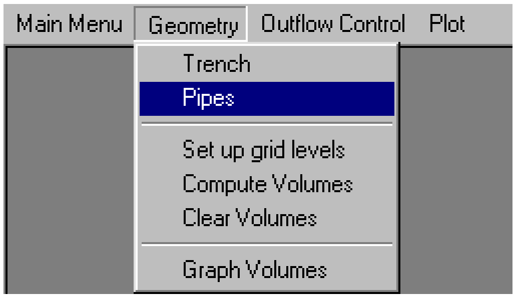
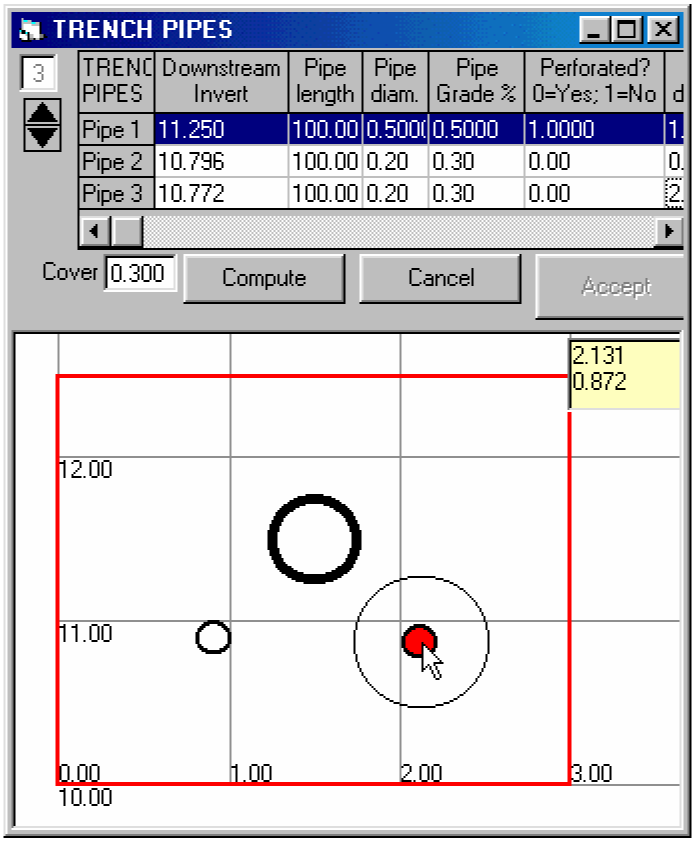


Figure 4-93 – One or more pipes can be defined in the Trench

Clicking on the **Geometry / Pipes** option opens a window which contains both a table for data entry and a graphic display of the trench cross-section on which pipes can be positioned. If an Outflow Pipe has already been defined that is located within the trench, this pipe will be shown as located at the downstream end of the trench.



TRENC PIPES	Downstream Invert	Pipe length	Pipe diam.	Pipe Grade %	Perforated? 0=Yes; 1=No	d
Pipe 1	11.250	100.00	0.500	0.5000	1.0000	1.
Pipe 2	10.796	100.00	0.20	0.30	0.00	0.
Pipe 3	10.772	100.00	0.20	0.30	0.00	2.

Cover 0.300

Figure 4-94 – The pipes can be defined graphically and fine-tuned numerically

As with the other data tables you must click on the Up-arrow of the Spin Button to open a row for the first new pipe. In addition, a perforated pipe with the default diameter is located on the centre line and at the bottom of the trench. The length and gradient of the pipe are initially set to the same values as specified for the trench. The default diameter is 200 mm or 8". The figure shows the new pipe (the second of two perforated pipes) being dragged to a location to one side of the storm sewer at an invert level of approximately 10.8 m. The circle around the pipe being dragged provides a way of ensuring a specified minimum cover between pipes. When both perforated pipes have been positioned you can refine the location by editing the data in the grid.

When you press the [Compute] button the column of available volume storage values is slightly increased because of the absence of clear stone in the volume contained by the perforated pipes.

The solid (i.e. non-perforated) storm sewer has the opposite effect of removing available storage volume due to volume occupied by the pipe. Finally, press the [Accept] button to close the window.

4.8.14 Plotting the Trench Properties

The Plot command is accessed from the special Trench menu. Options 2 and 3 which involve the exfiltration flow are not available until after the [Route] button has been pressed since exfiltration is computed as part of the routing operation.

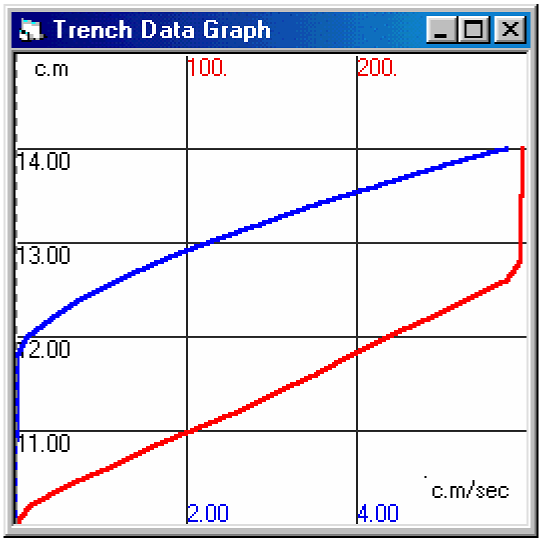


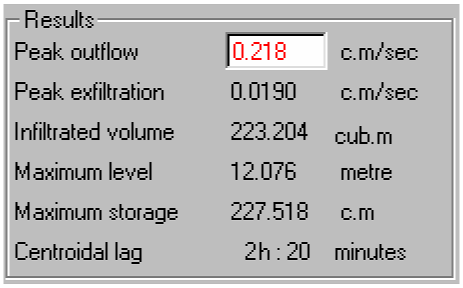
Figure 4-95 – Using the Plot menu to plot Trench properties

This plot shows the "V,Q = f(H)" option with storage volume in red and outflow in blue.

The "Q + Qinf = f(H)" option shows the exfiltration flow alone compared to the total outflow (i.e. weir, orifice and exfiltration). However, this is not available until the Inflow has been routed through the Trench which is the next step.

4.8.15 Routing the Inflow through the Trench

When you have completed the data entry you can test the effectiveness of the proposed design by clicking on the [Route] button. As with some other design commands information is presented in several ways.



Results

Peak outflow	0.218	c.m/sec
Peak exfiltration	0.0190	c.m/sec
Infiltrated volume	223.204	cub.m
Maximum level	12.076	metre
Maximum storage	227.518	c.m
Centroidal lag	2h : 20	minutes

Figure 4-96 – The results of routing the Inflow through the Trench

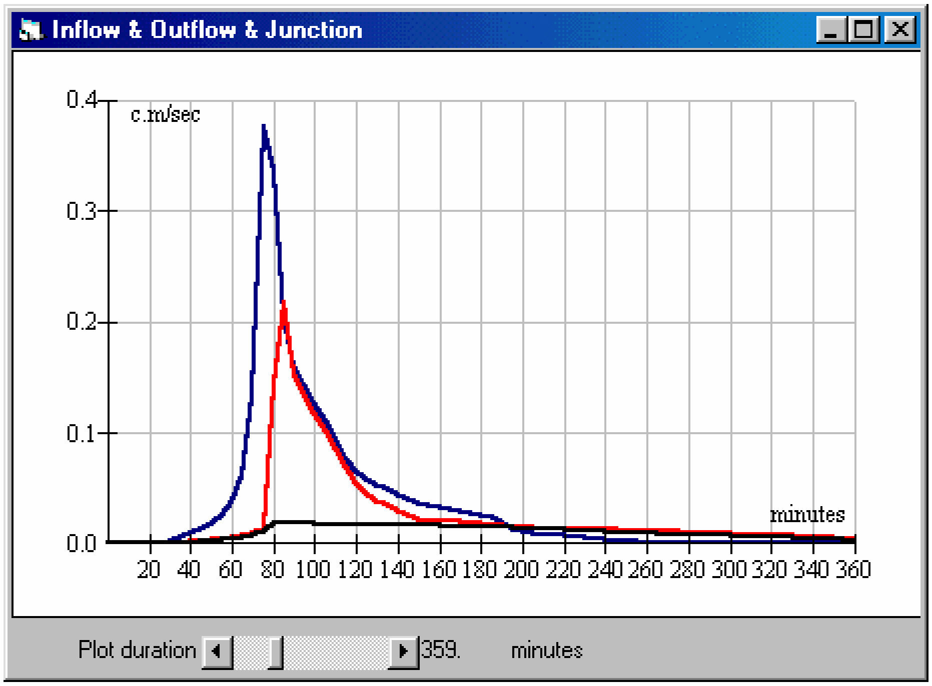
The "Results" frame within the Trench window shows the maximum values for outflow, exfiltration, water level and net storage volume in the trench. If the length of the storage 'wedge' is less than the available trench length a message is displayed. This means that the trench could be made shorter with no change in its effectiveness – at least for this inflow hydrograph.

In addition, the total exfiltrated volume is reported.

The outflow hydrograph is also displayed in tabular form from which the total outflow volume can be noted. You should be able to confirm that the sum of the outflow volume and the volume of exfiltration is reasonably close to the volume of the inflow hydrograph. Any error in continuity is usually due to the exfiltration or outflow hydrograph continuing beyond the maximum length of hydrograph specified in the **Time parameters** command.

Finally, the centroidal lag of the outflow hydrograph is reported.

4.8.16 Results of Trench Routing



Plot duration 359. minutes

Figure 4-97 – A graph of Inflow, Outflow and Exfiltration

The graphical display shows three flow hydrographs for inflow, outflow and exfiltration. The time base of the displayed hydrographs can be changed by clicking on or moving the horizontal slider below the graph area. This may be useful if you want to see how long exfiltration persists.

As with other graphical displays, the graph window can be moved by dragging the title bar or resized by dragging an edge or a corner of the graph window.

Time in minutes and values of the flows at the vertical through the mouse pointer can be displayed in the small window in the top right corner of the graph area by moving the mouse pointer over the graph. Holding down the right (secondary) button on the mouse causes a pair of cross hairs to be displayed.

The coloured dots against the flow values serve as a legend for the three hydrograph plots. You can display a set of written labels by clicking on the window containing the coloured dots to toggle this on and off.

Finally, the grid can be modified by clicking with the left (primary) mouse button in the window containing the flow values.

Should you wish to distinguish the hydrographs by line pattern instead of colour you can make use of the Show/Graph Style command in the Main Menu. You will have to set the thickness of all lines to one pixel in order to make line patterns visible since Windows does not support patterns with thick lines. You will have to repeat the [Route] command to display the new line properties.

Most of the windows displayed can be moved and re-sized and you may find it useful to construct a composite of the overall design and print a hardcopy for your file.

4.9 Culvert Design

The Design/Culvert menu command lets you model the behaviour of a culvert under various conditions of flow. Because of the many variables involved, the process is largely one of trial and error and MIDUSS does not suggest initial feasible values for the design.

Culvert design can be carried out for either steady, (i.e. time invariant) flow or for an inflow hydrograph. When inflow is in the form of a hydrograph the hydraulic design can be followed by a routing process that shows the attenuation of the inflow hydrograph caused by ponding that occurs upstream of the embankment. In such cases the peak outflow from the barrel will be less than the peak inflow and you can refine the barrel design for the reduced flow if desired or acceptable to the regulatory body. Fig 4-98 shows an example of a culvert design in profile.

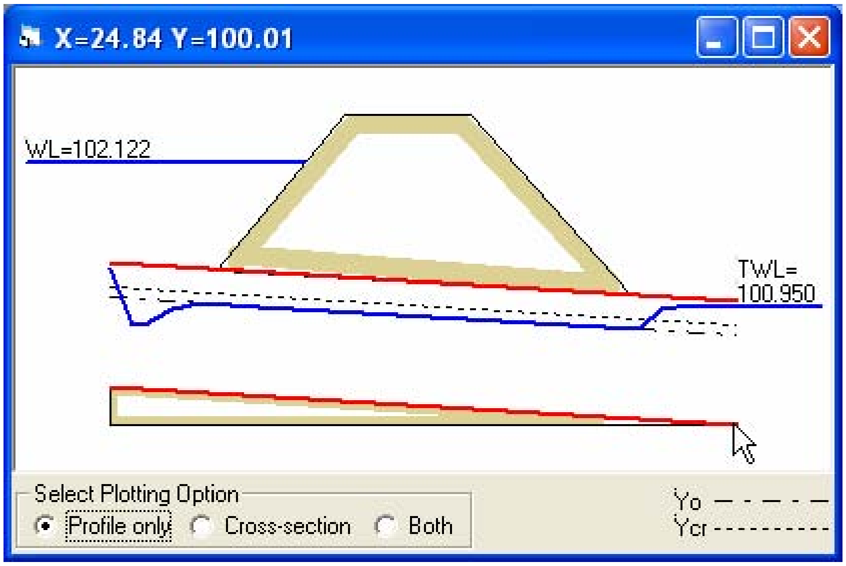


Figure 4-98 – Example of a culvert design plot.

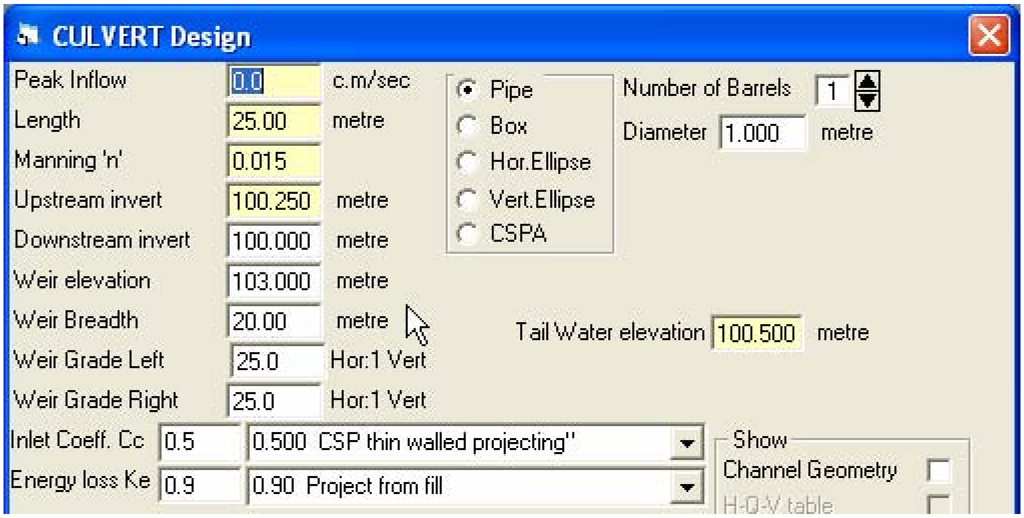
If desired, the Culvert design can be preceded by a Channel design with either a trapezoidal or complex cross-section. When this is done the cross-sectional shape of the channel is 'inherited' by the culvert design and used to describe the flow cross-section upstream of the culvert. If the inflow is a flow hydrograph, a channel design may be followed by a Channel routing process from which the channel outflow forms the inflow to the culvert.

The culvert is assumed to be located below a sag point in a highway embankment that will form an overflow weir in the event that the barrel flow capacity is sufficiently surcharged. Flow separation between barrel and weir flow is assumed to be recombined downstream of the barrel.

The cross-section of the barrel conduit may be a circular pipe, a rectangular box, a horizontal or vertical ellipse or a pipe arch. Multiple barrels may be used but cross-section and other hydraulic parameters are assumed to be the same for all barrels.

4.9.1 Culvert Data Requirements

The upper part of the Culvert design form in Fig 4-99 contains data entry fields for user defined values. The figure shows the default values in metric units. If the Culvert design was invoked when the Inflow hydrograph was zero the field Peak Inflow is shown as zero and must therefore be defined before attempting a design.



Peak Inflow	0.0	c.m/sec
Length	25.00	metre
Manning 'n'	0.015	
Upstream invert	100.250	metre
Downstream invert	100.000	metre
Weir elevation	103.000	metre
Weir Breadth	20.00	metre
Weir Grade Left	25.0	Hor:1 Vert
Weir Grade Right	25.0	Hor:1 Vert
Inlet Coeff. Cc	0.5	0.500 CSP thin walled projecting"
Energy loss Ke	0.9	0.90 Project from fill

● Pipe
○ Box
○ Hor.Ellipse
○ Vert.Ellipse
○ CSPA

Number of Barrels 1
Diameter 1.000 metre
Tail Water elevation 100.500 metre

Show
Channel Geometry ☐
H-Q-V table ☐

Figure 4-99 – Default data for Culvert design

The Downstream invert of the barrel has a default value of 100 (m. or ft) and sets a datum (either real or arbitrary) for all other elevations and inverts. Values for Weir elevation, breadth, grade left and grade right approximate the weir shape formed by the vertical curve of the highway at the sag point. This is shown by the trapezoidal black line illustrated in Fig 4-100 below.

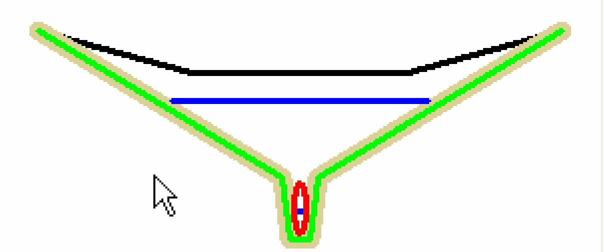


Figure 4-100 – Typical culvert cross section

Barrel cross-section is selected from the cluster of Option buttons. For Pipes, only the diameter is required but for all others both Height and Width must be defined.

If an elliptical or pipe arch barrel is selected a drop down list is displayed from which you can select one of a number of commercially available sizes.

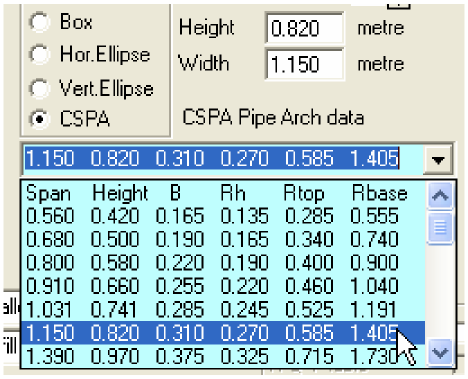
Fig 4-101 below shows the special case of a pipe arch for which 4 additional parameters must be defined.

B = height of arch springing above invert

Rh = haunch radius

Rtop = top radius

Rbase = bottom radius



○ Box
○ Hor.Ellipse
○ Vert.Ellipse
● CSPA

Height 0.820 metre

Width 1.150 metre

CSPA Pipe Arch data

1.150 0.820 0.310 0.270 0.585 1.405

Span	Height	B	Rh	Rtop	Rbase
0.560	0.420	0.165	0.135	0.285	0.555
0.680	0.500	0.190	0.165	0.340	0.740
0.800	0.580	0.220	0.190	0.400	0.900
0.910	0.660	0.255	0.220	0.460	1.040
1.031	0.741	0.285	0.245	0.525	1.191
1.150	0.820	0.310	0.270	0.585	1.405
1.390	0.970	0.375	0.325	0.715	1.730

Figure 4-101 – Selecting CSPA dimensions

The Tail Water Level at the downstream end of the barrel should also be defined.

A number of the text boxes for data entry have a light yellow background. These parameters can be modified by small increments by clicking on the data while holding down the Alt or Shift key to cause either a negative or positive increment respectively. By also holding down the Ctrl key the size of the increment is reduced for finer adjustments.

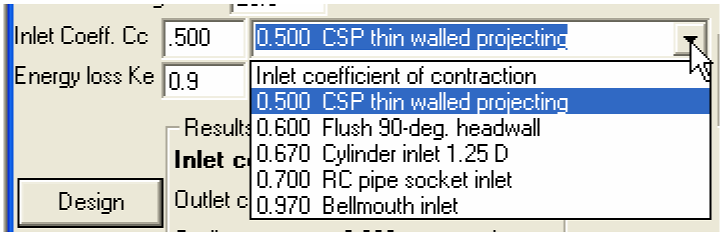
This feature is valuable for sensitivity analysis such as seeing the effect of Tail Water elevation on the location of a hydraulic jump or the effect of barrel gradient on uniform flow depth.

This feature is also available for the Inflow rate but only for the case in which there is no Inflow hydrograph.

4.9.2 Inlet Flow Conditions

The values of Inlet Coefficient of contraction Cc and the inlet Energy loss coefficient Ke may be entered directly or selected from the drop-down lists to the right of the data entry field.

The coefficient of contraction Cc defines the depth at the *vena contracta* when the inlet is submerged. Submergence is assumed when the upstream depth is greater than 1.4 times the barrel height. Cc approaches 1.0 as streamlining of the entrance is improved. Fig 4-102 shows an example of selecting the Cc.



Inlet Coeff. Cc .500

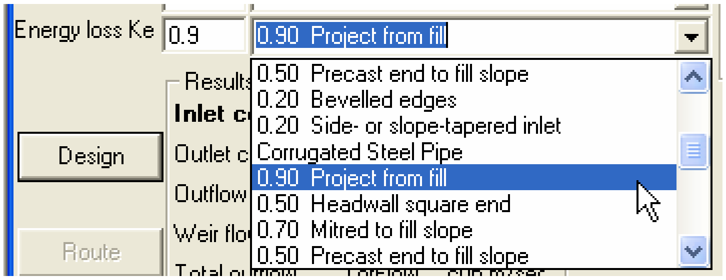
Energy loss Ke 0.9

0.500 CSP thin walled projecting

Inlet coefficient of contraction
0.500 CSP thin walled projecting
0.600 Flush 90-deg. headwall
0.670 Cylinder inlet 1.25 D
0.700 RC pipe socket inlet
0.970 Bellmouth inlet

Figure 4-102 – Selecting the Inlet Coeff. Cc

Energy loss at the entrance (Fig 4-103) is approximated as a fraction of the velocity head in the barrel. The fraction Ke is small for well-streamlined inlets and increases to 0.9 as the entrance becomes more sharp-edged.



Energy loss Ke 0.9 0.90 Project from fill

Figure 4-103 – Selecting Energy loss coefficient Ke

4.9.3 Culvert Design Trials

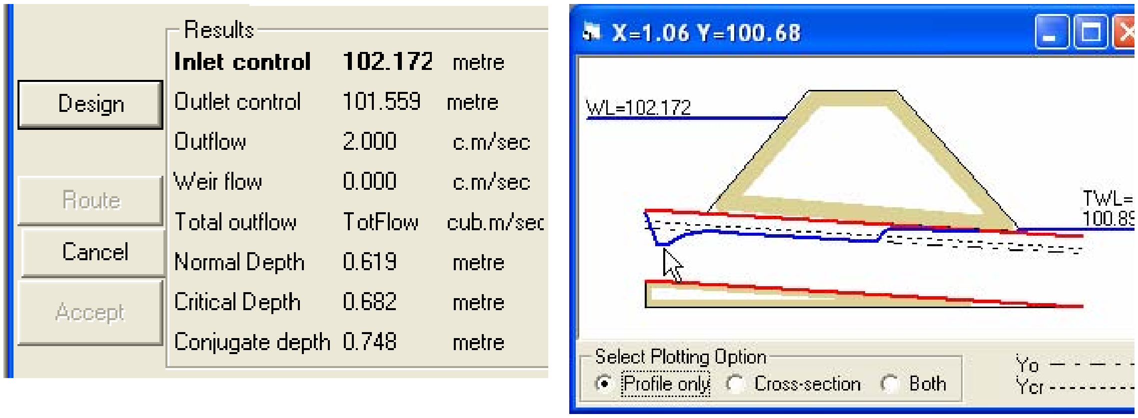
When flow and design parameters have been entered you can click the [Design] button. This carries out an analysis of flow at the barrel entrance and over the length of the culvert. Normal and critical depth are calculated and if the barrel is steep (Yo < Ycr) the location of a hydraulic jump for the current Tail Water depth is estimated.

The upstream elevation is the maximum of the two estimates obtained for Inlet control and Outlet control. As shown below, both results are displayed but the critical one is in bold type.

Two graphical displays are shown, one of the culvert profile (as shown below) and another showing a cross-section of the upstream channel with the upstream water level and the water surface in the barrel.

If the weir is overtopped an estimate of Weir flow is shown in the results frame and is also indicated on both graphical displays.

When the [Design] button has been pressed the [Accept] button is enabled and you can press this to close the Culvert design at this point. Typically, however, you will want to experiment by adjusting the design parameters, barrel configuration or tail water depth. As an aid in interpreting the results you choose to display only one or other of the two graphical windows by selecting the appropriate option button. You can also enlarge the graph window by dragging an edge or corner of the graph window. The aspect ratio of the plotted area is maintained.



Results

Inlet control	**102.172**	metre
Outlet control	101.559	metre
Outflow	2.000	c.m/sec
Weir flow	0.000	c.m/sec
Total outflow	TotFlow	cub.m/sec
Normal Depth	0.619	metre
Critical Depth	0.682	metre
Conjugate depth	0.748	metre

Select Plotting Option
● Profile only ○ Cross-section ○ Both

Figure 4-104 - Typical Culvert design results for a constant inflow

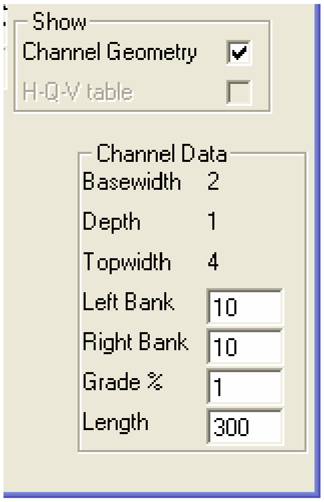
4.9.4 Routing Unsteady Flow

If the inflow is a hydrograph, completion of the hydraulic design causes the [Route] button to be enabled. Routing the inflow hydrograph is likely to show significant attenuation of the peak flow only when significant ponding has been caused by the maximum water level upstream of the culvert.

Routing is done by the same Storage-Indication method that is used for the Pond and Trench design methods. A grid of values of Elevation – Discharge - Volume is set up for a range of discharges. The volume upstream of the culvert is computed as a function of the upstream depth and several simplifying assumptions for the upstream gradient, channel length and the slope of the floodplain embankments. The channel cross-section is either inherited from a previous Channel design or based on default values based on the number and size of the barrels defined for the culvert.

Checking the check-box labeled ‘Channel Geometry’ (Fig 4-105) causes the information on the right to be displayed. Channel Basewidth, Height and Topwidth cannot be changed but the embankment slopes, channel grade and length can be modified.

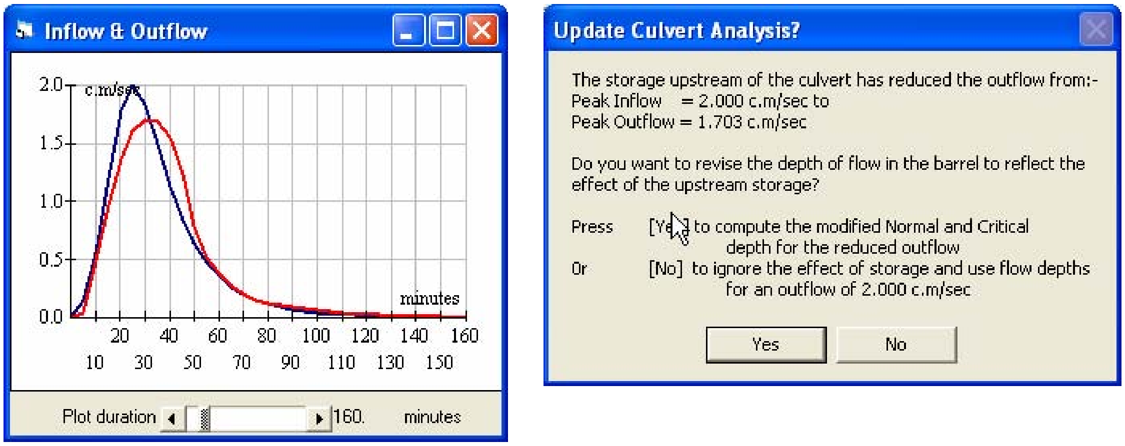
This provides enough information to compute the upstream storage volume to be calculated as a function of the downstream depth, assuming that the ponded reach has a horizontal surface.


Show
Channel Geometry ☑
H-Q-V table ☐

Channel Data
Basewidth 2
Depth 1
Topwidth 4
Left Bank 10
Right Bank 10
Grade % 1
Length 300

Figure 4-105 – Upstream channel data.

Pressing the [Route] button causes the Inflow hydrograph to be routed through the storage volume upstream of the culvert and a graph and tabular display show the resultant Outflow hydrograph.



The storage upstream of the culvert has reduced the outflow from:-
Peak Inflow = 2.000 c.m/sec to
Peak Outflow = 1.703 c.m/sec

Do you want to revise the depth of flow in the barrel to reflect the effect of the upstream storage?

Press [Yes] to compute the modified Normal and Critical depth for the reduced outflow
Or [No] to ignore the effect of storage and use flow depths for an outflow of 2.000 c.m/sec

Figure 4-106 – Results of Routing through the culvert design.

If the culvert is surcharged to a significant extent it is likely that the upstream storage volume will cause some attenuation of the inflow peak flow. Now since the design of the barrel was based on the peak Inflow it may be possible to refine the design of the barrel for the reduced peak outflow. A message giving you the option to revise the design for the reduced outflow is displayed (as shown). Note that some regulatory authorities may not allow this because a worst case scenario may allow for the possibility of the embankment having been 'washed out' by an extreme event.

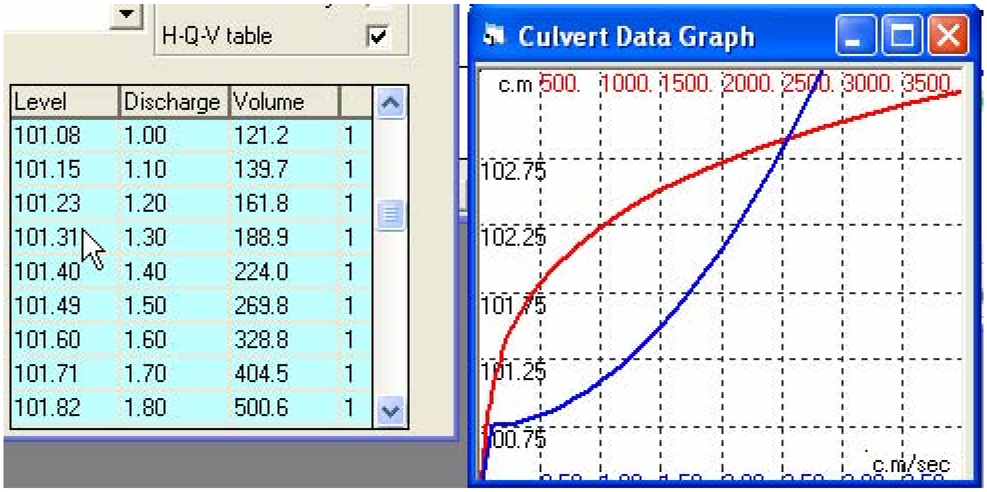
If you click [Yes] in the message box, the current culvert will be analyzed for the peak outflow.

4.9.5 Modifying the Upstream Storage

Because the estimate of upstream storage volume is based on several questionable assumptions, you may wish to modify this data if more accurate data is available. The first step is to display the

Elevation – Discharge – Volume information in tabular and graphical mode. Note that this is possible only after the initial Routing operation has been completed.

Clicking on the checkbox labeled ‘H-Q-V Table’ causes data to be displayed as shown below in Fig 4-107.



Level	Discharge	Volume	
101.08	1.00	121.2	1
101.15	1.10	139.7	1
101.23	1.20	161.8	1
101.31	1.30	188.9	1
101.40	1.40	224.0	1
101.49	1.50	269.8	1
101.60	1.60	328.8	1
101.71	1.70	404.5	1
101.82	1.80	500.6	1

Figure 4-107 – You can edit the H-Q-V data as required after the initial Routing.

The grid and the graph show values of Discharge and Volume for a range of elevations up to and exceeding the weir elevation. The range of discharge values is distributed linearly but the corresponding levels are non-linear. The fourth column of the grid indicates either ‘0’ or ‘1’ corresponding to Inlet control or Outlet control.

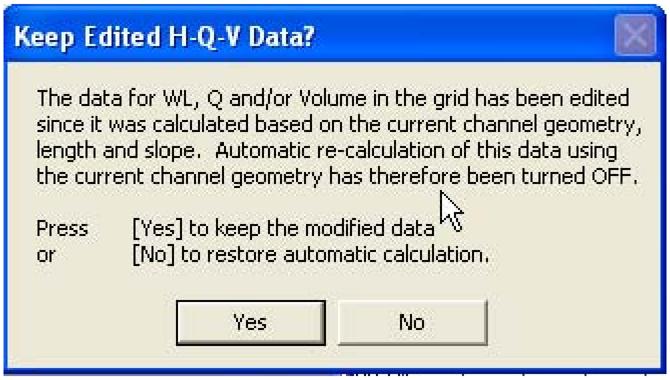
Typically you may want to improve the estimates of storage volume for the various water levels based on data (such as contour areas) available from a CAD drawing of the upstream channel topography.

You can do this in two ways.

1. You can edit the data in the grid directly by clicking on a cell to select it and then type in a new value – say of volumes. The graphical display will not show the effect of the edit until the graphic window is refreshed. The easiest way to do this is to drag one edge or corner of the graphic display.

2. By using the Edit/Copy and Edit/Paste menu commands (or Ctrl+C and Ctrl+V respectively) you can export the entire grid of values to another program such as Microsoft EXCEL, edit the values and then re-import them via the Clipboard. Note that you must define the selected area by selecting all four columns even although column 4 is of no relevance to the process. The graphical display will be updated when the graphic window is refreshed.

When you press the [Route] command button a warning message is displayed if the H–Q–V data has been modified either by direct editing or by importing data from the Clipboard. A typical message (for the former case) is shown below in Figure 4-108.



Keep Edited H-Q-V Data?

The data for WL, Q and/or Volume in the grid has been edited since it was calculated based on the current channel geometry, length and slope. Automatic re-calculation of this data using the current channel geometry has therefore been turned OFF.

Press [Yes] to keep the modified data
or [No] to restore automatic calculation.

Figure 4-108 – Warning message that the H-Q-V data has been edited.

You can return to the default H – Q – V data based on the current channel configuration by pressing the [No] button. Clicking on the [Yes] button will retain the modified data and, if Accepted, this data will be reported in the output file.

4.10 Cascade Design

On-site storage for commercial development frequently requires some underground storage facility to handle runoff from frequent storm event in order to reduce the frequency with which parking lot storage may be an inconvenience. The required storage may be provided by a 'super-pipe' or oversized storm sewer but even with a very modest slope the reach length is limited to 100 - 150m (330 - 500 ft). Two reaches of super-pipe can be used in series but then the depth in the downstream chamber may cause a backwater effect on the outflow control of the upstream chamber.

The Cascade command lets you route the current inflow hydrograph through a short cascade of storage cells formed from a variety of cross-sectional shapes such as pipes, rectangular boxes, horizontal and vertical elliptical pipes and pipe arch sections. Fig 4-109 presents the conceptual schematic of a cascade design.

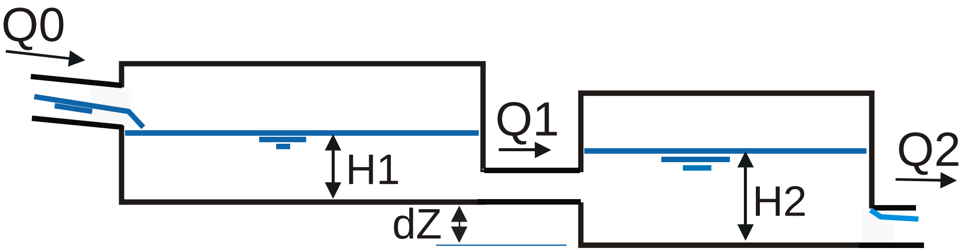


Figure 4-109 - Schematic of two element Cascade

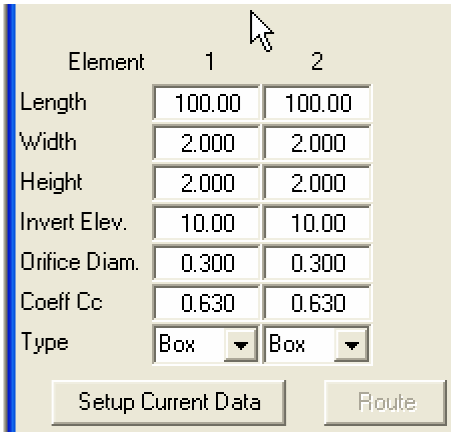
The schematic shows two elements connected by a short orifice pipe. It is clear that the outflow Q1 is dependant on the depths H1 and H2 and also on dZ, the difference in the invert elevations of the two chambers.

4.10.1 Data Entry for the Cascade

The current version of the command is limited to two elements. The form displays the data entry fields in a number of columns equal to the number of elements.

Each chamber is horizontal with a specified length, width, height and invert elevation.

The outflow control from each chamber is assumed to be an orifice of specified diameter and coefficient of contraction with the orifice invert equal to the bottom of the upstream chamber. Fig 4-110 shows an example of the data form.



Element	1	2
Length	100.00	100.00
Width	2.000	2.000
Height	2.000	2.000
Invert Elev.	10.00	10.00
Orifice Diam.	0.300	0.300
Coeff Cc	0.630	0.630
Type	Box	Box

Figure 4-110 – Setting the Cascade data

The bottom row shows a drop-down list (Fig 4-111) from which the type of cross-section can be selected. If a Pipe is selected then the data entry field for 'Width' is disabled ('grayed' out) and only Height can be specified.

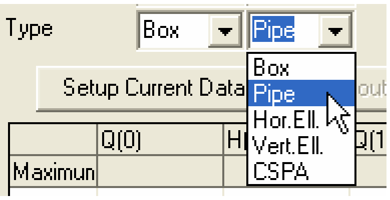
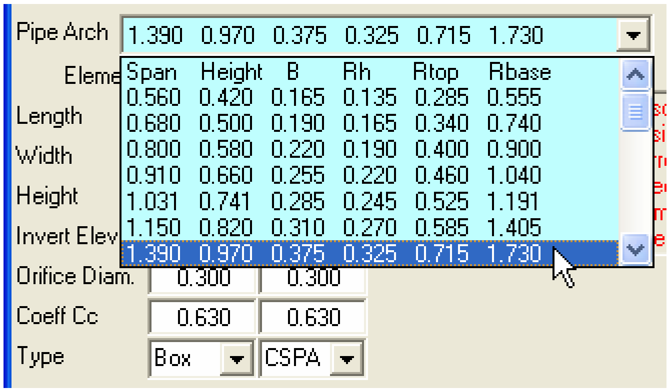


Figure 4-111 – Selecting the type of cross section

If any of the three special pipe sections are used (e.g. elliptical or pipe arch) another drop-down list on line 3 of the form is opened (Fig 4-112) to let you browse through a set of commercially available sizes. These are shown in metric or imperial sizes depending on the choice of units.

For elliptical sections only Width and Height are required. For a Pipe Arch additional data is needed for:

B = springing height
Rh = haunch radius
Rtop = top radius
Rbase = bottom radius



Pipe Arch 1.390 0.970 0.375 0.325 0.715 1.730

Span	Height	B	Rh	Rtop	Rbase
0.560	0.420	0.165	0.135	0.285	0.555
0.680	0.500	0.190	0.165	0.340	0.740
0.800	0.580	0.220	0.190	0.400	0.900
0.910	0.660	0.255	0.220	0.460	1.040
1.031	0.741	0.285	0.245	0.525	1.191
1.150	0.820	0.310	0.270	0.585	1.405
1.390	0.970	0.375	0.325	0.715	1.730

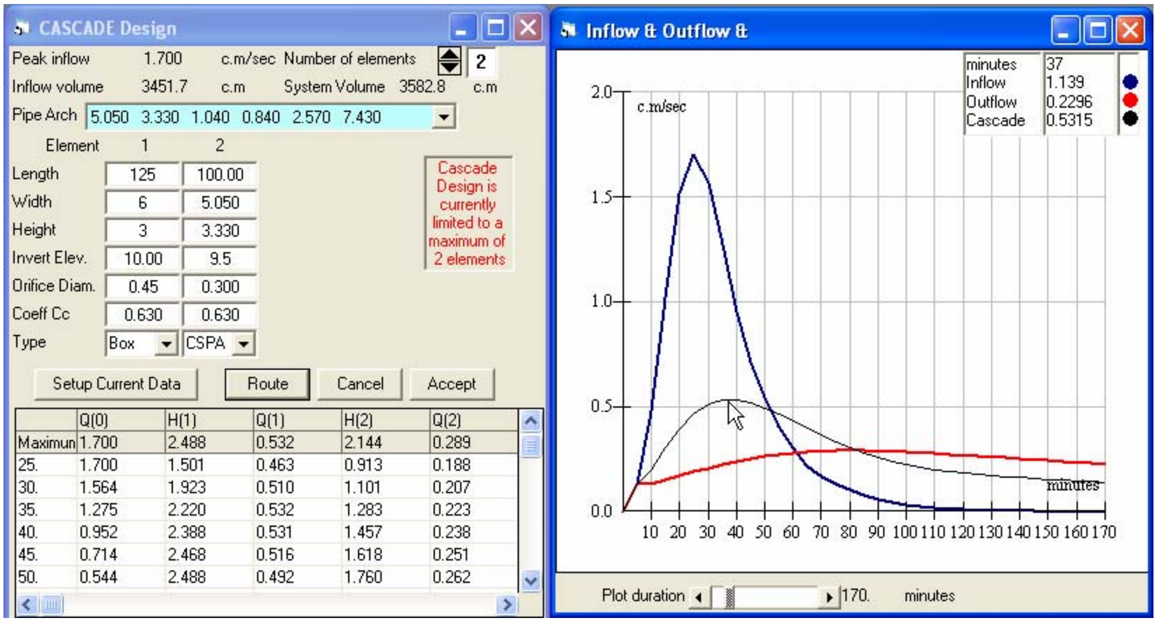
Eleme		
Length		
Width		
Height		
Invert Elev		
Orifice Diam.	0.300	0.300
Coeff Cc	0.630	0.630
Type	Box	CSPA

Figure 4-112 – Selecting pipe arch cross sections.

4.10.2 Analysis of the Design

Each iteration of the analysis requires two steps – first click on [Setup Current Data] to enable the [Route] command and then click [Route] to carry out the analysis. Pressing [Setup Current Data] also causes the total storage volume of the system to be displayed for comparison with the volume of the Inflow hydrograph.

The second row of the grid shows the peak values of the Inflow, outflow from each cell and maximum depth in each element. These correspond to the dimensions shown in the schematic of Fig 4-113.



Peak inflow 1.700 c.m/sec Number of elements 2
Inflow volume 3451.7 c.m System Volume 3582.8 c.m
Pipe Arch 5.050 3.330 1.040 0.840 2.570 7.430

Element	1	2
Length	125	100.00
Width	6	5.050
Height	3	3.330
Invert Elev.	10.00	9.5
Orifice Diam.	0.45	0.300
Coeff Cc	0.630	0.630
Type	Box	CSPA

Cascade Design is currently limited to a maximum of 2 elements

	Q(0)	H(1)	Q(1)	H(2)	Q(2)
Maximun	1.700	2.488	0.532	2.144	0.289
25.	1.700	1.501	0.463	0.913	0.188
30.	1.564	1.923	0.510	1.101	0.207
35.	1.275	2.220	0.532	1.283	0.223
40.	0.952	2.388	0.531	1.457	0.238
45.	0.714	2.468	0.516	1.618	0.251
50.	0.544	2.488	0.492	1.760	0.262

minutes	37	
Inflow	1.139	●
Outflow	0.2296	●
Cascade	0.5315	●

Figure 4-113 – Routing the flow through the cascade design.

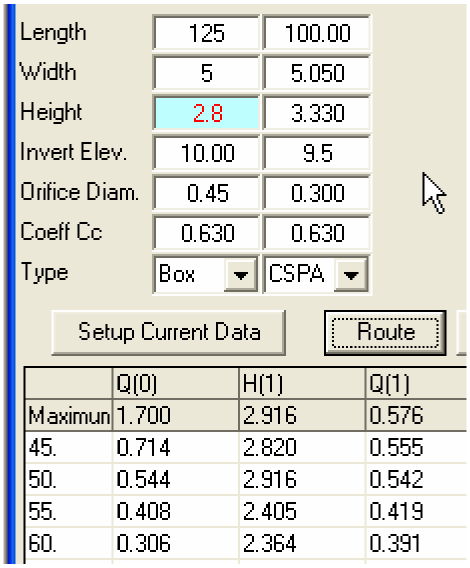
In addition to the graphical display a table of the final outflow hydrograph is shown.

4.10.3 Dealing With Surcharge

If a cell is surcharged, the data box containing the Height is highlighted to warn you that more storage or a larger orifice is required.

In Fig 4-114 below, the head H(1) reaches a peak of 2.916 m and this maximum is reported in row 2 of the grid.

The data cell holding the height of 2.8 m is highlighted to warn you that surcharging has occurred. Increasing the storage volume or increasing the orifice flow should solve the problem. In the case illustrated, increasing the length by 5 m reduces the maximum head to 2.753 m and eliminates the surcharge



Length	125	100.00
Width	5	5.050
Height	2.8	3.330
Invert Elev.	10.00	9.5
Orifice Diam.	0.45	0.300
Coeff Cc	0.630	0.630
Type	Box	CSPA

	Q(0)	H(1)	Q(1)
Maximun	1.700	2.916	0.576
45.	0.714	2.820	0.555
50.	0.544	2.916	0.542
55.	0.408	2.405	0.419
60.	0.306	2.364	0.391

Figure 4-114 – Surcharged condition is highlighted.

Chapter 5 Hydrograph Manipulation

This chapter describes the various operations which can be used in MIDUSS to manipulate the flow hydrographs which are generated during the hydrologic simulation. In addition, particular attention is given to the processes by which junction nodes are handled and some special considerations when dealing with relatively complex networks of nodes and links.

A brief description of the various commands is contained in Chapter 2 - *Structure and Scope of the Main Menu* under the heading The Hydrograph Menu. Some of the information in that section is duplicated here for convenience of reference.

The topics covered in this chapter are summarized below.

- An Introduction to Networks
    - Networks of Different Complexity
    - A Network numbering Convention
    - A Simple Tree Network
    - Representing a Circuited network
    - Hydrograph Manipulation Commands
        - The Start Command
            - The Start/New Tributary Option
            - The Start/Edit Inflow Option
            - Editing an Existing Inflow Hydrograph
        - The Add Runoff Command
        - The Next Link Command
        - The Combine Command
        - The Confluence Command
        - Handling Old Junction Files
    - Example 1 - Treatment of a Single Junction
    - Example 2 - Treatment of a Circuited network

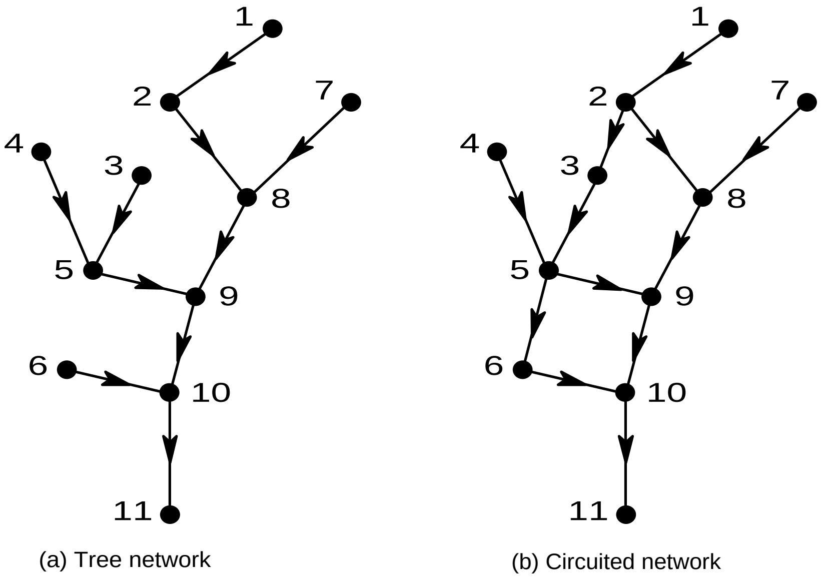
5.1 An Introduction to Networks

Drainage networks - whether they are natural drainage basins or man made stormwater systems - usually develop in the shape of a tree. Small tributaries join together to form larger tributaries leading eventually to a single, large channel or conduit at the outflow point of the watershed. A tree network can be visualized as a series of branches or links which connect a set of points with the minimum number of links. The points are referred to as nodes.

Notice that you will need to treat the major and minor drainage systems as separate networks connected at locations where there are diversion devices.

These nodes represent either:

1. junctions where two or more links join,
2. points in a single branch representing some discontinuity in conduit geometry or discharge.



(a) Tree network

(b) Circuited network

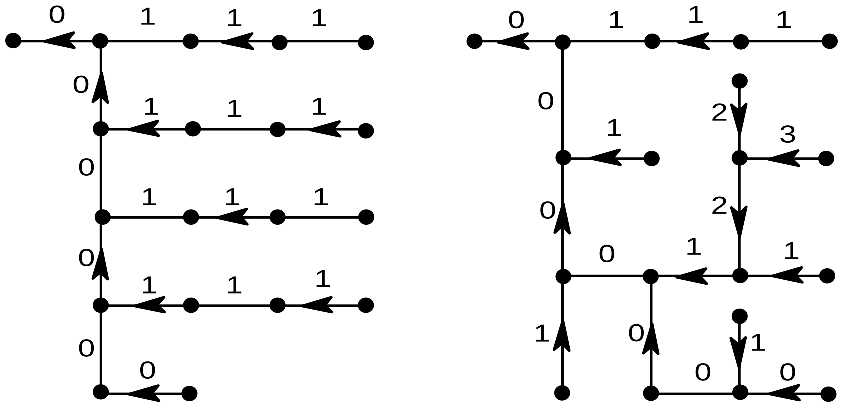
Figure 5.1 - (a) A Tree network and (b) A Circuited network.

Figure 5.1(a) shows a simple tree network comprising 11 nodes and 10 links. In a tree network the number of links is always one less than the number of nodes. Another important characteristic of a tree network is that each internal node can have several inflow links but only <u>one</u> outflow link.

Figure 5.1(b) shows the same set of 11 nodes but with 12 connecting links. The two additional links form closed paths and the network is said to be circuited. In each of the two loops that are formed, there is one node that has <u>two</u> outflow links forming a bifurcation. Circuited networks are much loved by the water (or gas) supply engineer since the extra redundant links provide additional reliability of supply for certain demand points. In storm sewers, a circuited network may result if stormwater can be partially diverted by some control device such as a side-discharge weir or even a simple catchbasin. Devices of this type can be simulated by the **Design/Diversion** command (see Chapter 4 *Design Options Available* Diversion Structure Design).

5.2 Networks of Different Complexity

Figures 5.2(a) and (b) show two typical tree networks with different levels of complexity. The tree of Figure 5.2(a) is of first order complexity because every link is no more than one junction away from the main branch. The order of the branch is indicated by the number beside it in the figure.



(a) First Order Tree

(b) Third Order Tree

Figure 5.2 - Tree networks of different order

Figure 5.2(b) shows a tree with the same number of nodes (and therefore links) but with more complex connectivity. This tree is 3rd order, but depending on the choice of main branch, it may even appear to be 4th order.

The significance of the different orders of complexity is the number of junctions that you must keep track of at any one time in order to estimate the flow in every link of the tree. In Figure 5.2(a) only one junction node need be considered at any one time. In Figure 5.2(b) as many as three or four junctions may be required to process the flows through the network.

The purpose of this chapter is to describe how MIDUSS can help you to manipulate and keep track of the flow hydrographs in each link of the drainage network. An important first step is to define a logical numbering convention for networks.

5.3 A Network Numbering Convention

MIDUSS is intended for the analysis and design of drainage systems that are tree networks. This means that strictly speaking each node should have no more than a single outflow link. To avoid ambiguity the convention is adopted whereby each link is assigned the same number as the node at its upstream end. Despite this apparent restriction, it is possible to describe a looped or circuited network as noted below (see the section later in this chapter on Representing a Circuited Network).

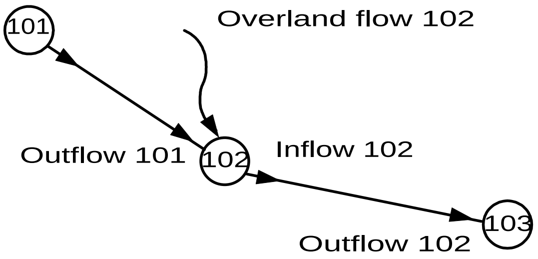


Figure 5-3 – A Network Numbering convention

As illustrated in Figure 5-3, each node is associated with a possible overland flow runoff hydrograph which is identified by using the same subscript number as the node where it enters the drainage system. Each link has associated with it an inflow and an outflow hydrograph which take the same subscript as the link (which is the same as the upstream node number). The node numbers used are quite arbitrary and used simply for reference purposes. Node numbers must be integers in the range 0 - 99999. It is usual but not required to use node numbers which increase in the downstream direction.

During analysis and design these hydrographs are stored in the three arrays for overland flow, inflow and outflow hydrographs. The nature of the design process is such that only one hydrograph of each type need be stored at any one time, so only one storage array of each type is required.

A fourth temporary hydrograph array is provided in order that an outflow hydrograph entering a junction node can be stored temporarily while other tributary branches which enter the same junction node are analyzed and designed. The special commands **Hydrograph/Combine** and **Hydrograph/Confluence** are provided to allow this type of manipulation. The peak flow values of these four hydrographs are displayed in the peak flow summary table and updated with the completion of each command - typically by pressing an [Accept] button.

In MIDUSS an additional hydrograph storage array is defined to serve as a Backup Hydrograph so that an 'Undo' command can be implemented but the current value of the backup is not displayed. Usually the presence of a Backup hydrograph is indicated by the fact that the 'Undo' menu item in the **Hydrograph** menu is enabled, otherwise it is 'grayed out'.

With this numbering convention established, we can now apply this to a simple tree network.

5.4 A Simple Tree Network

Figure 5.4 shows a simple 6 node network with 5 links. The figure also shows the various hydrographs associated with the system using the numbering convention just described. In this example, catchments contributing overland flow are located only at nodes (1), (3) and (5).

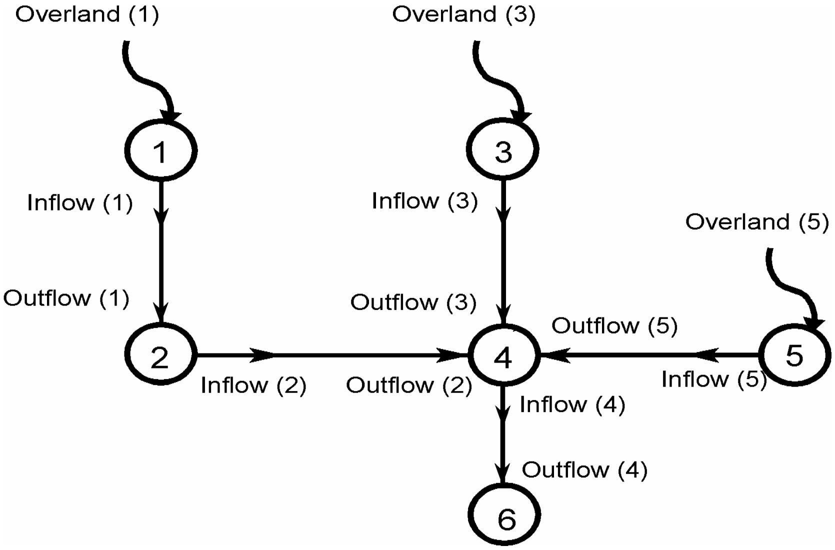


Figure 5.4 - Schematic diagram of a 6-node tree network.

The numbering convention used here can also be employed to describe a circuited network.

5.5 Representing a Circuited Network

Circuited networks may be represented by using the **Design/Diversion** command to define a structure or device which splits the inflow hydrograph into an outflow hydrograph and a diverted hydrograph. The latter is written to a file which can later be retrieved by means of the File / Load file / Hydrograph command (see Chapter 6 *Working with Files*), to create a new 'tributary' to the tree network. The idea is illustrated in Figure 5.5.

It should be emphasized that this type of 'looped' network does not have to satisfy the requirement that the algebraic sum of the head losses around a loop must be zero (i.e. Kirschhoff's 2nd law). This is because the diversion structure at the branching node involves a control section at which the head loss is unknown.

For example, in Figure 5.5 node 6 is a branching node. The two branches join again at node 12. However, the total head loss along the two routes (6)-(7)-(9)-(12) and (6)-(10)-(11)-(12) will not be equal and one of the branches entering node (12) is likely to involve a control or section of critical flow..

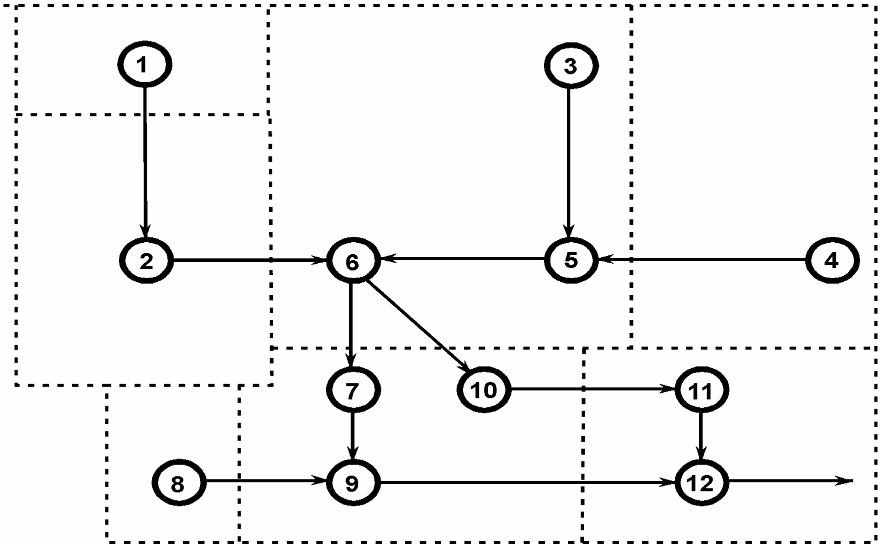


Figure 5.5 - Representing a circuited network as a tree.

Two simple examples are illustrated at the end of this chapter but before looking at these you should review the various commands available for manipulation of flow hydrographs.

5.6 Hydrograph Manipulation Commands

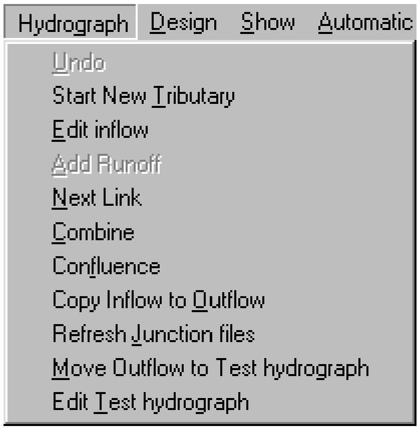


Figure 5-6 – The Hydrograph menu

As shown in this menu fragment, a number of commands are provided to let you manipulate hydrographs in various ways. These are described briefly in Chapter 2, *Structure and Scope of the Main Menu* and are summarized here as follows:

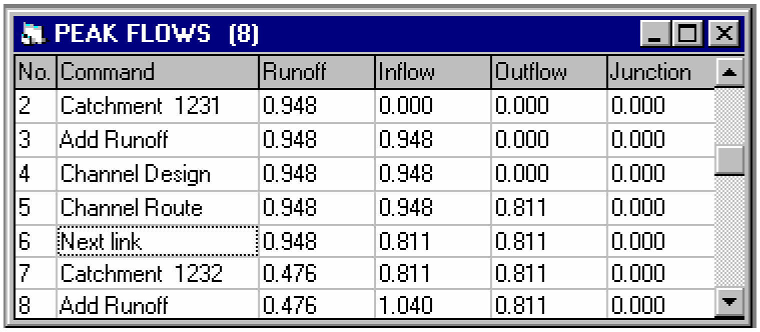
The Undo command	Reverses the most recent change to the hydrographs.
Start New Tributary	Allows you to either define the elements of the inflow hydrograph or set them all to zero. The zero option of this command should be used when starting a new tributary.
Edit Inflow	Lets you modify an existing inflow hydrograph or create a new hydrograph based on available data.
Add Runoff	Adds the overland flow hydrograph to the current inflow hydrograph; the overland flow is unchanged. This causes the runoff from a catchment to be added to the main flow in the drainage network.
Next Link	Sets the inflow hydrograph equal to the current outflow hydrograph. This command allows the design to proceed to the next downstream element of the drainage network.
Combine	Adds the outflow hydrograph to the current contents of the hydrograph at a specific junction node. If no junction hydrograph exists a new one is created. This enables the outflow from two or more branches flowing into a junction node to be accumulated. The resulting total hydrograph is stored in the temporary or junction hydrograph array.
Confluence	Takes the accumulated hydrograph at a junction node and copies it into the inflow hydrograph array. The previous contents of the inflow hydrograph are overwritten. The temporary hydrograph is set equal to zero and the junction file is deleted.

Copy Inflow to Outflow	Lets you copy the inflow to the outflow hydrograph with no attenuation or time lag.
Refresh Junction Files	lets you review the junction files in the current job directory at any time. You can remove any files that are no longer required. It is invoked automatically the first time you use the Combine command in a design session.
Move Outflow to Test hydrograph	Lets you move the outflow hydrograph into the test hydrograph.
Edit Test hydrograph	Lets you either create a test hydrograph or modify one that has been imported.

All of the above commands cause a backup copy to be made of the hydrograph which is being changed. The **Hydrograph/Undo** menu item is enabled and you may reverse the process to restore the affected hydrograph to its previous value. Only one level of backup is provided.

The operations listed above are described in more detail in the sections which follow.

Each of the above commands is followed automatically with an update of the peak flow summary table that shows the new peak flow values in all four of the hydrograph arrays.



No.	Command	Runoff	Inflow	Outflow	Junction
2	Catchment 1231	0.948	0.000	0.000	0.000
3	Add Runoff	0.948	0.948	0.000	0.000
4	Channel Design	0.948	0.948	0.000	0.000
5	Channel Route	0.948	0.948	0.811	0.000
6	Next link	0.948	0.811	0.811	0.000
7	Catchment 1232	0.476	0.811	0.811	0.000
8	Add Runoff	0.476	1.040	0.811	0.000

Figure 5-7 - A typical peak flow summary table.

5.6.1 The Start Command

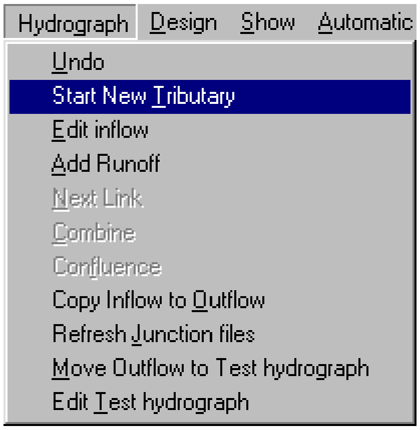


Figure 5-8 –The Start New Tributary menu

As shown above, the **Hydrograph/Start** menu item can be used in two ways. You can either:

(a) set the Inflow hydrograph to zero

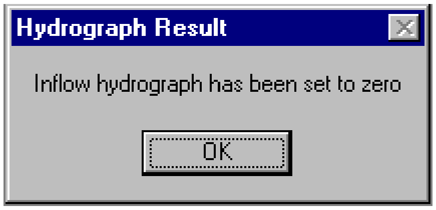
or (b) edit the current inflow hydrograph.

You should use the **Start/New Tributary** option after you have used the Combine command to store an outflow hydrograph at a junction node and wish to start a new tributary of the drainage network.

The **Start/Edit Inflow** option is less commonly used but may be useful to input a hydrograph which represents observed data for comparison with a simulated hydrograph. Alternatively, you may wish to input a hydrograph obtained by use of another model.

5.6.2 The Start New Tributary Option

This option must be used when the design of a new tributary branch is started, so that when the furthest upstream overland flow hydrograph is generated it can be loaded into an empty inflow hydrograph by means of the **Hydrograph/Add Runoff** command.



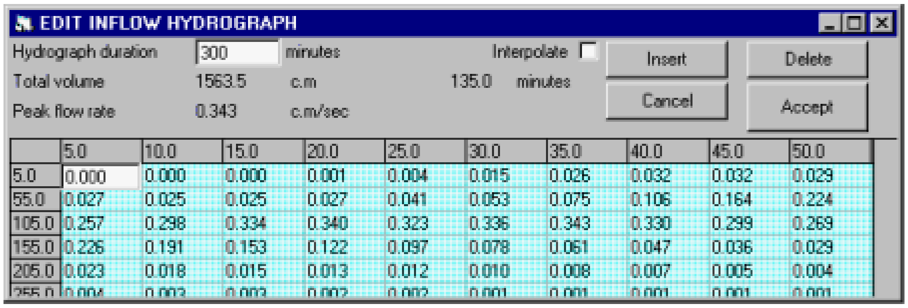
Inflow hydrograph has been set to zero

Figure 5-9 – The message following the Start/New tributary command

When you use the Start New Tributary command the message shown in Figure 5-9 is displayed. You can continue either by clicking on the [OK] button or simply pressing the space bar. The summary table of peak flows is also updated to show the current inflow peak as zero.

5.6.3 The Edit Inflow Option

You can use this option to key in the flow values during a MIDUSS design session. When you invoke the **Hydrograph/Edit inflow** command the current Inflow hydrograph is displayed in both graphical and tabular form. If the Inflow hydrograph is currently empty the graph will be a zero height, horizontal line and the table on the lower left corner will contain only zeros. The number of elements will equal the number of time steps in the maximum hydrograph duration specified at the start of the session.



Hydrograph duration 300 minutes
Total volume 1563.5 c.m
Peak flow rate 0.343 c.m/sec
135.0 minutes

	5.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	45.0	50.0
5.0	0.000	0.000	0.000	0.001	0.004	0.015	0.026	0.032	0.032	0.029
55.0	0.027	0.025	0.025	0.027	0.041	0.053	0.075	0.106	0.164	0.224
105.0	0.257	0.298	0.334	0.340	0.323	0.336	0.343	0.330	0.299	0.269
155.0	0.226	0.191	0.153	0.122	0.097	0.078	0.061	0.047	0.036	0.029
205.0	0.023	0.018	0.015	0.013	0.012	0.010	0.008	0.007	0.005	0.004
255.0	0.004	0.003	0.003	0.002	0.002	0.001	0.001	0.001	0.001	0.001

Figure 5-10 – The Inflow hydrograph editing window

If a finite Inflow hydrograph is displayed, it is likely that a large number of elements towards the end of the hydrograph will be zero and you can discard these by setting the duration to an appropriate value. In Figure 5-10 a value of 300 minutes has been specified. On the graph you can move the time slider to show as little or as much of the hydrograph as you wish.

In the table, the time value (left of the [Cancel] button) is the time in minutes for the cell under the mouse pointer. The current cell for text entry is initially at the first cell in the top row and can be moved by means of the arrow keys. When the active cell is at the right end of a row, pressing the right arrow key will cause it to wrap around to the left cell of the next row.

The Inflow hydrograph can be edited as described in the next topic Editing an Existing Inflow Hydrograph

5.6.4 Editing an Existing Inflow Hydrograph

The tabular display shown in Figure 5-10 includes a number of command buttons with the following uses.

[Insert] causes a new cell to be inserted immediately prior to the current active cell and the new cell becomes the active cell. All the cells following the new cell are moved to the right (i.e. later) by one time step and the hydrograph duration is increased by one time step. The value entered in the new cell depends on whether or not the 'Interpolate' check box is checked or not.

- If 'Interpolate is left unchecked the value in the new cell is zero.
- If 'Interpolate is checked the value is a linear interpolation of the values in the two adjacent cells. If [Insert] is clicked when the first cell in the table is the active cell, the new cell becomes the first cell and the value is one half of the cell to the right (i.e. the previous first cell)

The statistics of total volume and peak flow are updated and the duration is increased by one time step.

[Delete] causes the currently active cell to be deleted and all cells after the deleted cell are moved one time step to the left (i.e. earlier). The duration of the hydrograph is reduced by one time step. If the [Insert] button was previously disabled it is re-enabled. The statistics of total volume and peak flow are updated.

[Cancel] causes the command to be aborted and the Inflow hydrograph is restored to its value prior to executing the **Hydrograph / Edit inflow** command. The graph window is closed, the Inflow hydrograph is displayed in tabular form superimposed on the Edit Inflow table and a message is displayed to tell you that the Inflow hydrograph has been restored. A line showing the restored Inflow peak is entered in the peak flow summary table.

[Accept] causes the changes in the inflow hydrograph to be made permanent. The graph window and the Edit Inflow table are closed and a line showing the new peak value is added to the peak flow summary table. An important point to note is that the modified Inflow hydrograph is not copied to the current Output file as was the case with earlier Miduss 4.72 for DOS. If you want to use this hydrograph in future runs in Automatic mode you must save it as hydrograph file by using the **File /Save file/Hydrograph** command.

You can change the duration of the hydrograph by entering a desired duration in minutes in the text box. This must be not greater than the maximum hydrograph duration specified at the start of the session. If the specified duration is less than or greater than the current length the grid will be modified by truncating the trailing values or adding additional cells with zero values.

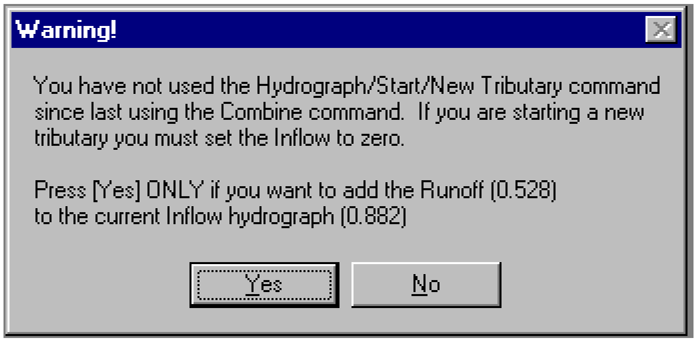
The duration can also be changed by using the [Insert] or [Delete] buttons.

To enter a new value or change an existing one make the cell to be edited the active cell either by moving to it with the arrow keys or by clicking on it with the mouse pointer. Then type the new value. As you enter values, you will see the total volume value in the table header being updated; the peak flow rate is also updated if you change the maximum value in the table. The changes you make to the data are also reflected in the graphical display.

When editing the data in this way, the Backspace key is active but the Delete key is not.

5.6.5 The Add Runoff Command

This command causes the Runoff flow hydrograph to be added to the current Inflow hydrograph array. The contents of the overland flow array are left unaltered. If you use the option **Options/Other options/Show Next logical menu** item, the Add Runoff menu is automatically highlighted and the mouse pointer positioned after you have accepted the Catchment command. The command has the effect of accumulating the most recently generated runoff hydrograph into the drainage network.



You have not used the Hydrograph/Start/New Tributary command since last using the Combine command. If you are starting a new tributary you must set the Inflow to zero.

Press [Yes] ONLY if you want to add the Runoff (0.528) to the current Inflow hydrograph (0.882)

Figure 5-11 – MIDUSS warns you against possible 'double counting'

At the start of a new tributary or branch, the **Add Runoff** command should be preceded by the **Start/New Tributary** command to ensure that the inflow hydrograph array is empty. MIDUSS detects if you have recently used the Combine command and then try to use the Add Runoff command without having set the Inflow hydrograph to zero. You will see a warning message as shown in Figure 5-11.

You will note that since the peak flows in the overland flow and inflow hydrographs are likely to occur at different times the peak flow of the sum of the hydrographs will usually be less than the sum of the peak flows of the individual hydrographs.

Runoff from last catchment area has been added to the Inflow hydrograph

Figure 5-12 – Typical message following a hydrograph operation.

On successful completion of the **Add Runoff** command the message shown in Figure 5-12 is displayed. You can delete this and continue either by clicking on the [OK] button in the message box, or by pressing any key (e.g. the spacebar) on the keyboard

5.6.6 The Next Link Command

The **Hydrograph/Next Link** command causes the current Outflow hydrograph to be copied to the Inflow hydrograph array. The command will normally be used following any of the Design commands which cause a new Outflow hydrograph to be created (e.g. Route, Pond, Trench or Diversion). In order for the design to proceed to the next downstream element of the drainage network the outflow from the current element must become the inflow for the adjacent downstream link. Hence the need for the Next Link command.

After each of the Design commands which result in a new outflow you have two choices.

(i) The outflow may enter a junction node and thus require use of the **Hydrograph/Combine** command.

(ii) The outflow may simply pass to the next link downstream and thus require you to use the **Hydrograph/Next Link** command.

5.6.7 Working with Junction Nodes

The **Hydrograph/Combine** and **Hydrograph/Confluence** commands allow you to combine hydrographs at junction nodes in a network. They make use of the Junction or Temporary hydrograph array which serves as a storage buffer in which two or more outflow hydrographs can be accumulated.

As discussed in the section An Introduction to Networks at the beginning of this chapter, it may be necessary to store the accumulated flow hydrographs at more than one junction node at any one time. To illustrate this idea, Figure 5.13 shows the same 3rd order network as Figure 5.2(b) but with node numbers added for reference.

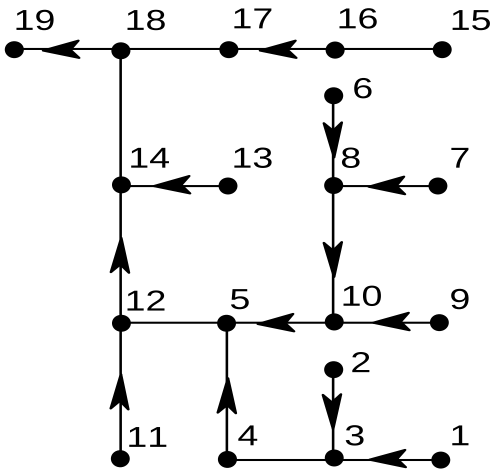


Figure 5.13 - A complex network

Assuming that modelling started with node #1 the sequence of operations might take the following form.

1 Generate flow at node 1.

2 Use Combine to create junction flow at node 3.

3 Generate flow at node 2.

4 Use Combine to add flow to junction node 3.

5 Get total flow at node 3 (using Confluence) and continue downstream.

6 Use Combine to create junction flow at node 5.

7 Generate flow at node 6.

8 Use Combine to create junction flow at node 8.

and so on.

In this case as many as three junctions may be 'active' at any one point in the modelling of the network. In order to do this, MIDUSS creates a file which serves as a backup for the temporary storage array. Each time you use the **Combine** command to create a new junction node you are prompted to specify the number of the junction node. This should be a unique integer number in the range 1 to 99999 which is used to create a file name of the form Hydnnnnn.JNC which is stored in the current Job directory.

When a junction node is defined in this way it is added to the current list of junction nodes and a file with the appropriate name is created. The outflow hydrograph array is copied (not added) to the temporary storage array, the contents are then written to a newly created disk file.

If an existing junction node is used, MIDUSS copies this hydrograph file to the temporary storage array, then adds the contents of the outflow hydrograph array to it and finally re-writes the updated contents of the temporary storage array to the same disk file. The original disk file is overwritten.

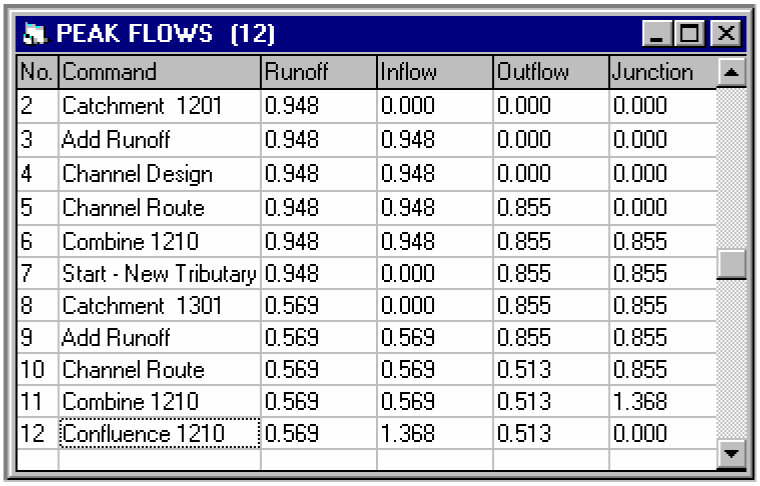
In both cases the updated contents of the temporary storage array are displayed in the table of peak flows. Refer to the topic on The Combine Command later in this chapter.

After using the **Hydrograph/Combine** command it is likely that you will want to define a new tributary. MIDUSS detects if you have recently used the **Combine** command and then try to use the **Add Runoff** command without having set the Inflow hydrograph to zero.

A peak flow summary table which illustrates the use of the Combine and Confluence commands is shown in Figure 5-14 in the next topic.

For more details on using the **Combine** command or the **Confluence** command refer to the section later in this chapter.

5.6.8 An Example of Using a Junction

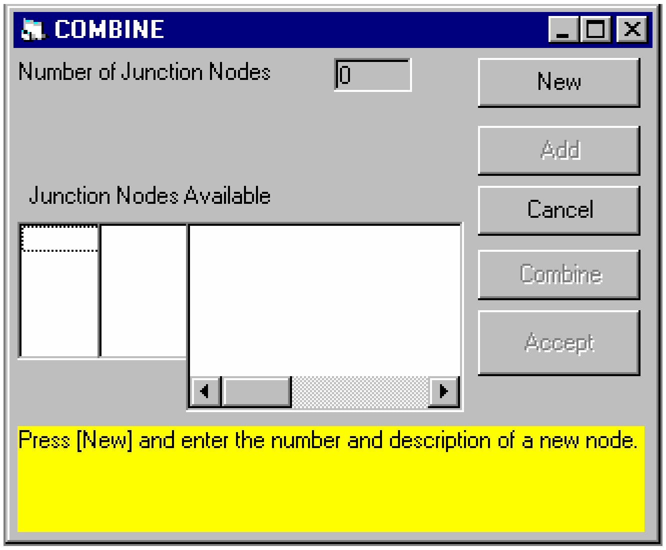


No.	Command	Runoff	Inflow	Outflow	Junction
2	Catchment 1201	0.948	0.000	0.000	0.000
3	Add Runoff	0.948	0.948	0.000	0.000
4	Channel Design	0.948	0.948	0.000	0.000
5	Channel Route	0.948	0.948	0.855	0.000
6	Combine 1210	0.948	0.948	0.855	0.855
7	Start - New Tributary	0.948	0.000	0.855	0.855
8	Catchment 1301	0.569	0.000	0.855	0.855
9	Add Runoff	0.569	0.569	0.855	0.855
10	Channel Route	0.569	0.569	0.513	0.855
11	Combine 1210	0.569	0.569	0.513	1.368
12	Confluence 1210	0.569	1.368	0.513	0.000

Figure 5-14 – The peak flow summary table illustrates use of the Combine command.

This summary table shows the peak flows resulting from two catchments 1201 and 1301 the runoff from which is conveyed by channel to a junction node number 1210. Notice that for the second catchment the same channel cross-section is used. Finally, the total flow at junction 1210 is transferred to the Inflow hydrograph by the **Confluence** command to allow the design to continue downstream. If necessary, more than two tributaries can be combined at a junction node.

5.6.9 The Combine Command



Number of Junction Nodes: 0

Press [New] and enter the number and description of a new node.

Figure 5-15 – The initial state of the Combine window

When the **Hydrograph/Combine** command is used the window shown in Figure 5-15 is displayed. If this is the first use of Combine, the list boxes will be empty and you must go through the following steps to define a junction node and add the Outflow hydrograph to it. The sequence of steps is as follows.

- Press [New] and enter a node number
- Enter a verbal description
- Press [Add] to copy this to the list box
- Select (i.e. highlight) the junction node by clicking on it in the list box
- Click [Combine] to add the current Outflow
- Click [Accept] to close the window

5.6.10 Define a Node number

Number of Junction Nodes 0

Node # 1201

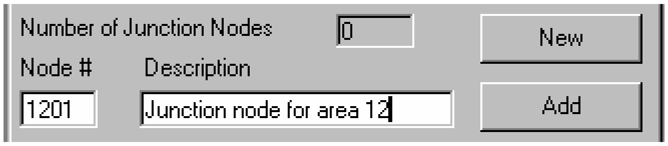
Description

Figure 5-16 – Defining a new junction node number

The Combine window will initially show no existing junction files and you must define a new one by entering a node number and description. Press the [New] button to open the text box for a node number and type any positive integer in the range 1 to 99999.

This node number is used to create a Junction file name of the form **HYD??????.JNC**. Thus in the example shown a file with the name **HYD01201.JNC** will be created. MIDUSS will not accept a node number less than 1.

5.6.11 Add a Description



Number of Junction Nodes 0

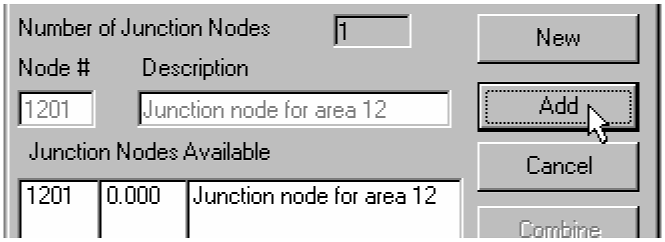
Node # 1201

Description Junction node for area 12

Figure 5-17 – Add a detailed description for future reference

The text box to contain the description is opened only after a node number has been entered. Click on the Description text box to set the focus and type in a verbal description. The length of the description can be much longer than the text box and can occupy several lines. These will be copied as a single record as part of the file header.

5.6.12 Add the Node to the List of junctions



Number of Junction Nodes 1

Node # Description

1201 Junction node for area 12

Junction Nodes Available

1201 0.000 Junction node for area 12

Figure 5-18 – Add the new node to the list of Junction Nodes Available

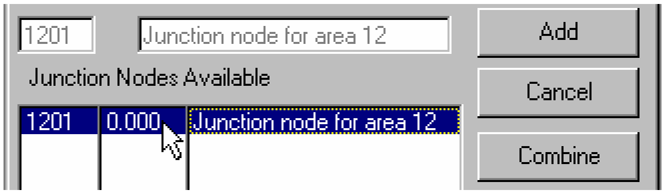
Once the node number and description of the new junction node have been defined you can click on the [Add] button. This will cause the file to appear in the 'Junction Nodes Available' list. In Figure 5-18, the new file is the only one in the list. If this was not the first junction node, it would be added at the foot of the list.

The list contains three sections defining:

1. the node number,
2. the peak flow of the accumulated hydrograph at the junction node. This is currently zero because nothing has been added yet, and
3. the node description. If the description is longer than the width of this section a horizontal scroll bar will appear at the foot of this section of the list.

The next step is to select a junction file from the list.

5.6.13 Select a Junction Node



1201 Junction node for area 12

Junction Nodes Available

1201 0.000 Junction node for area 12

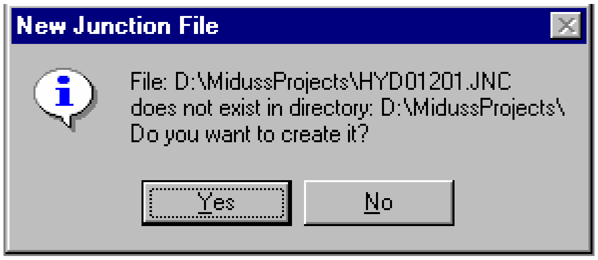
Figure 5-19 – Selecting a node enables the [Combine] button

To select a junction node from those shown in the list you must click on any part of the appropriate row with the mouse pointer. This will cause the junction file row to be highlighted as shown in Figure 5-19. Also, the [Combine] key will be enabled just to the right of the list.

Now you can add the current Outflow hydrograph to the selected junction file.

5.6.14 Add the Outflow

You can now click on the [Combine] button to add the current Outflow hydrograph to the accumulated flow at the selected junction node. If this is the first time this junction node has been used, selecting the file will cause a message (Figure 5-20) to be displayed advising you that the file does not exist and requesting confirmation to create the file.

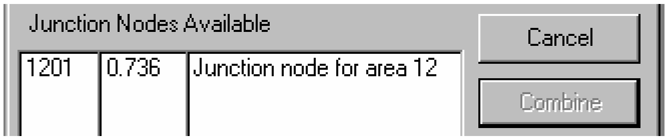

New Junction File

File: D:\MidussProjects\HYD01201.JNC
does not exist in directory: D:\MidussProjects\
Do you want to create it?

Figure 5-20 – Confirmation is required before a new file is created

The default button on the message box is [Yes] and you can accept this by clicking on the [Yes] button, or by pressing any key on the keyboard - e.g. the spacebar. Pressing [No] lets you choose another junction file.

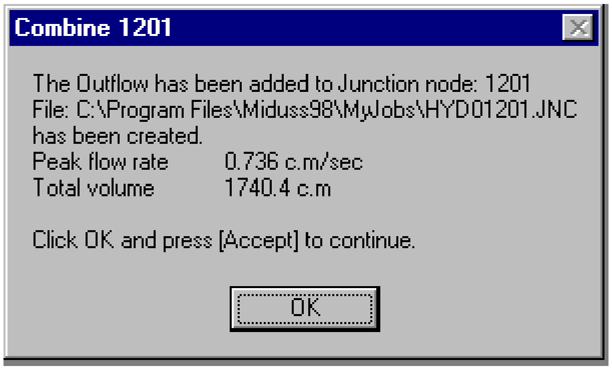
When you press the [Yes] button, the new value of the peak flow at the selected Junction node is displayed in the middle section of the list box as shown in Figure 5-21. The [Combine] button is disabled to prevent accidental double use of the Outflow hydrograph.


Junction Nodes Available

1201	0.736	Junction node for area 12

Figure 5-21 – Click [Combine] to update the peak flow at the selected junction

Finally, you are prompted to click on the [Accept] button, and a final message is displayed summarizing the updated peak flow and volume of the accumulated junction hydrograph. A typical message is shown in Figure 5-22.


Combine 1201

The Outflow has been added to Junction node: 1201
File: C:\Program Files\Miduss98\MyJobs\HYD01201.JNC
has been created.

Peak flow rate	0.736 c.m/sec
Total volume	1740.4 c.m

Click OK and press [Accept] to continue.

Figure 5-22 – MIDUSS describes the new junction hydrograph.

Now you can accept the result of the operation.

5.6.15 Accept the result

Adding the Outflow by pressing the [Combine] button enables the [Accept] button which has been "grayed out" up to now.

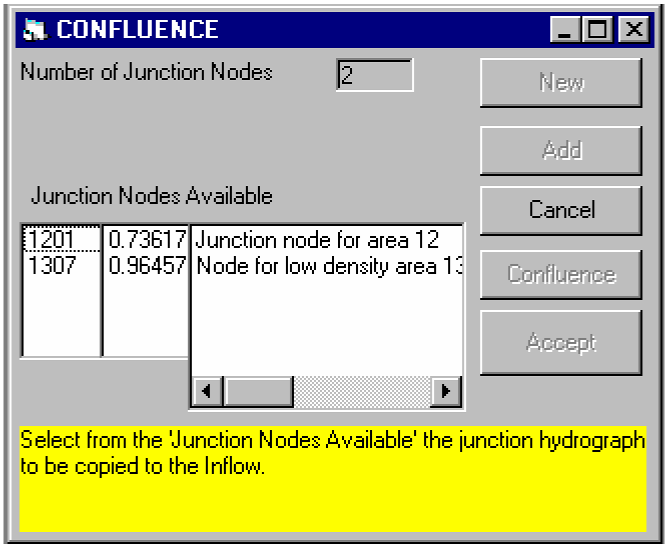
You will also have noticed helpful prompts displayed at the bottom of the Combine window. At this point you will be advised to press the [Accept] button to accept the result and close the Combine window.

When you do this you will see that the summary table of peak flows is updated with a new record noting the node number and the peak flow in the Junction or temporary hydrograph column.

5.6.16 The Confluence Command

This command is used when all the outflow hydrographs entering a junction node have been accumulated in the temporary storage array by means of two or more uses of the **Hydrograph/Combine** command. The **Hydrograph/Confluence** command causes the accumulated flow hydrograph at a particular node to be copied to the Inflow hydrograph. This enables you to continue with the design of the link (e.g. pipe or channel) immediately downstream of the junction node.

As described for the **Hydrograph/Combine** command, MIDUSS uses disk files as a backup of the temporary storage array in order that any number of junction nodes may be active simultaneously. These backup files are given names of the form **HYDnnnnn.JNC** as defined in the **Combine** command.



CONFLUENCE

Number of Junction Nodes 2

Junction Nodes Available

1201	0.73617	Junction node for area 12
1307	0.96457	Node for low density area 1

Select from the 'Junction Nodes Available' the junction hydrograph to be copied to the Inflow.

Figure 5-23 – The Confluence window will show at least one junction node

The window is basically the same as that used for the **Combine** command. Since the **Hydrograph/Confluence** command is enabled only after the **Combine** command has been used, there will be at least one Junction node shown in the list of Junction Nodes Available. Figure 5-23 shows two junction nodes from which we will choose the upper one for node 1201.

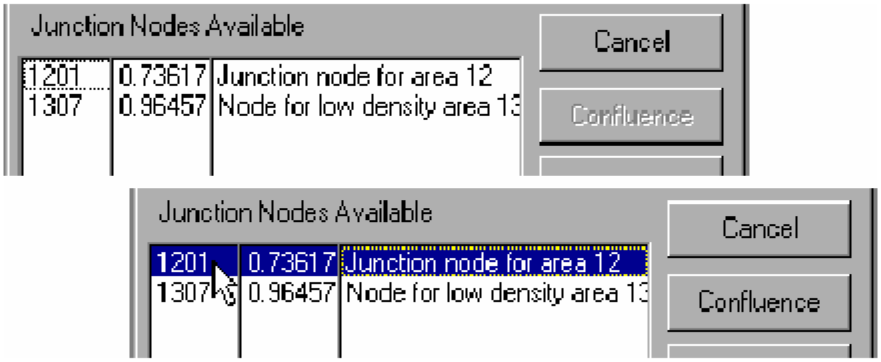
The procedure is outlined in the following steps.

- Select the Confluence node

- Copy the Junction hydrograph
- Accept the Confluence result.

A further safeguard is built into the Confluence command to reduce the potential for error in cases where the overland flow hydrograph from a sub-catchment enters the drainage network at a junction node. Because the confluence command causes the inflow hydrograph to be overwritten, it is important that the Add Runoff command be used after the Confluence command, otherwise the contribution of the local sub-catchment will be deleted. MIDUSS can detect a situation in which you may have inadvertently used the commands in the wrong order (i.e. Add Runoff followed by Confluence) and warns you that the overland flow has been deleted. You then have the opportunity to use the Add Runoff command a second time (since the overland flow hydrograph is unchanged) and thus recover from the error.

5.6.17 Select the Confluence Node



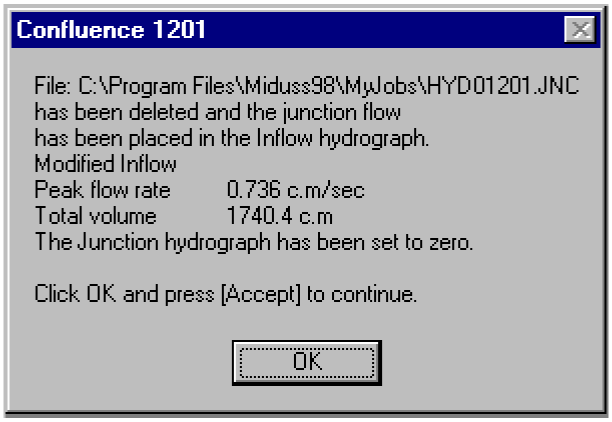
1201	0.73617	Junction node for area 12
1307	0.96457	Node for low density area 1

Figure 5-24 – Selecting a junction node enables the [Confluence] button

Figure 5-24 shows two fragments of the Confluence window containing the list of Junction Nodes Available. There are only two available in this example for junction nodes 1201 and 1307. Initially the [Confluence] button is disabled. By clicking with the mouse pointer on the row for node 1201 the row is highlighted as shown in the lower figure. The [Confluence] button is now enabled.

The next step is to copy the Junction hydrograph.

5.6.18 Copy the Junction Hydrograph



File: C:\Program Files\Miduss98\MyJobs\HYD01201.JNC
has been deleted and the junction flow
has been placed in the Inflow hydrograph.
Modified Inflow
Peak flow rate 0.736 c.m/sec
Total volume 1740.4 c.m
The Junction hydrograph has been set to zero.

Click OK and press [Accept] to continue.

Figure 5-25 – The flow and volume of the new Inflow hydrograph are displayed

When you click on the [Confluence] button, a message is displayed telling you that the transfer of the Junction hydrograph to the Inflow hydrograph has been completed. Also, the associated file has been deleted and the Temporary or junction hydrograph array has been set to zero.

As shown in Figure 5-26, the list of Junction Nodes Available has been modified to show only the remaining node. The [Confluence] button is once again disabled.

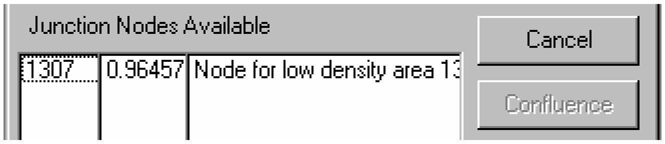


Figure 5-26 – The copied junction node is removed from the list

Now you should Accept the result of the Confluence operation.

5.6.19 Accept the Confluence Result

Pressing the [Confluence] button enables the [Accept] button which has been "grayed out" up to now.

You will also have noticed helpful prompts displayed at the bottom of the Combine window. At this point you will be advised to press the [Accept] button to accept the result and close the Confluence window.

When you do this you will see that the summary table of peak flows is updated with a new record noting the node number, the new peak flow in the Inflow hydrograph and a zero value in the Junction or Temporary hydrograph column.

5.6.20 Handling Old Junction Files

If a MIDUSS design is completed in a single session, any junction files (i.e. files with a '.JNC' extension) created by the **Hydrograph/Combine** command should be automatically deleted with the corresponding **Hydrograph/Confluence** command. However if a design is carried out in two or more sessions, or if a design session is aborted for some reason, it is possible that some junction files may be left in the currently defined Job Directory. This is particularly true if you frequently neglect to specify a Job Directory with the result that all files created by MIDUSS will accumulate in the MIDUSS folder (typically C:\Program Files\MIDUSS\ ).

In most cases junction files should be deleted since in repeating a run in automatic mode there is a danger that a flow hydrograph will be doubled if the **Hydrograph/Combine** command encounters an old file with the same node number.

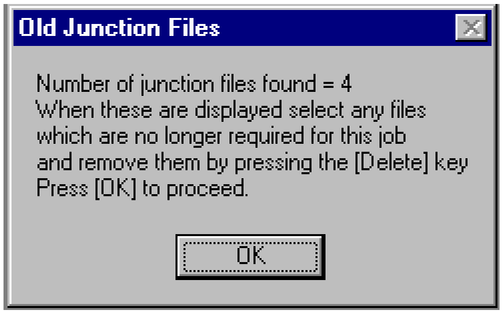
To help you deal with this potential problem MIDUSS makes a special check the first time the Combine command is used during a design session and detects all of the files in the current Job Directory which have an extension of *.JNC and gives you the chance to either remove them or retain them.

The procedure can be summarized in the following steps.

- Check for Junction Files
- Reviewing the Junction Files Available
- Selecting a File to Delete
- Continue to the Combine Command
- Confirming File Deletion on Exit

5.6.21 Check for Junction Files

The first time you use the **Hydrograph/Combine** command during a design session MIDUSS detects any junction files in the current Job Directory. If one or more are found the message shown below is displayed.



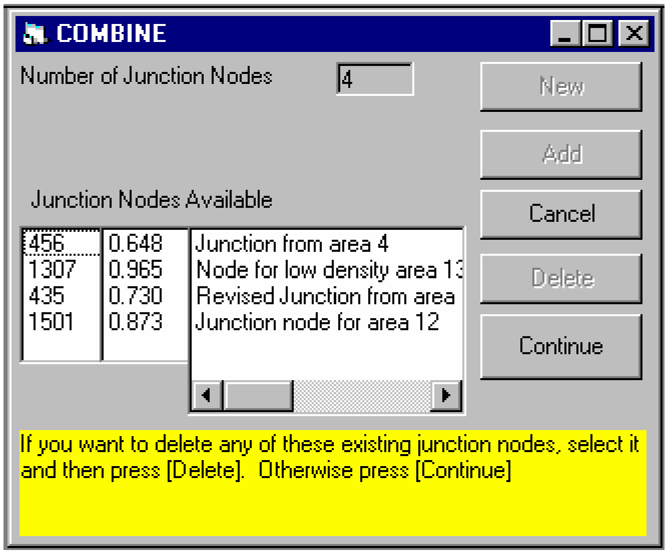
Number of junction files found = 4
When these are displayed select any files
which are no longer required for this job
and remove them by pressing the [Delete] key
Press [OK] to proceed.

Figure 5_27 – MIDUSS checks for previously created junction files

Close the message box by clicking on the [OK] button or press the spacebar (or any key) on the keyboard.

MIDUSS then displays the Junction Files Available.

5.6.22 Review the Junction Files Available



Number of Junction Nodes: 4

456	0.648	Junction from area 4
1307	0.965	Node for low density area 13
435	0.730	Revised Junction from area
1501	0.873	Junction node for area 12

If you want to delete any of these existing junction nodes, select it and then press [Delete]. Otherwise press [Continue]

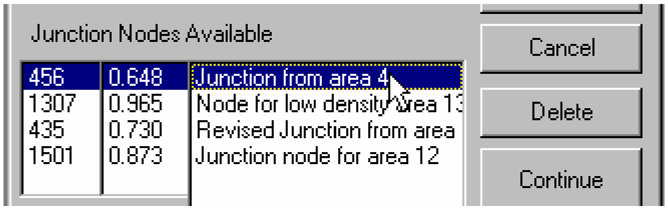
Figure 5-28 – All of the currently available junction nodes are displayed.

The **Hydrograph/Refresh Junction files** command uses the same window as is used for the **Combine** and **Confluence** commands as shown in Figure 5-28. In this example the 'Junction Nodes Available' list contains four files or nodes. In reviewing the files for possible deletion you will quickly learn the value of giving a full and meaningful description when junction files are first created.

Let's assume that only the bottom two (nodes 435 and 1501) are of use since they were created during a previous design session which was only partially completed and which is now being continued.

Now you can select a file to be deleted .

5.6.23 Select a File to Delete

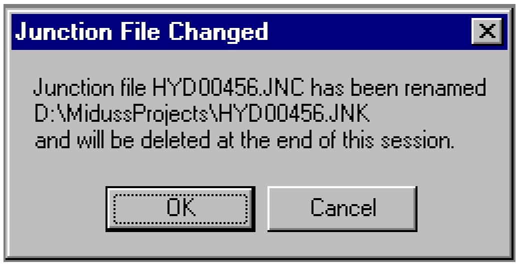


456	0.648	Junction from area 4
1307	0.965	Node for low density area 13
435	0.730	Revised Junction from area
1501	0.873	Junction node for area 12

Figure 5-29 – Selecting an old junction node to be deleted.

Select the file for node 456 by clicking on it with the mouse pointer. As shown, the row is highlighted and the [Delete] button is enabled.

Now press the [Delete] button to remove the selected junction. MIDUSS does not immediately delete the file **DIV00456.JNC** since it is possible that you may change your mind and recover the file. The file is therefore re-named by changing the extension to *.JNK . When the list is refreshed this file will no longer appear and the number of Junction Nodes will be reduced to three. This information is shown in a message box as follows.



Junction file HYD00456.JNC has been renamed
D:\MidussProjects\HYD00456.JNK
and will be deleted at the end of this session.

Figure 5-30 – MIDUSS advises you that the deleted file has only been re-named.

Should you change your mind at this point you can press the [Cancel] button and you will see a message that the junction file has been restored. When you close the message box the Combine window will be closed, but when you invoke the **Hydrograph/Combine** command again you will find the file appears once again in the list of Junction Nodes Available.

The process is repeated until only the junction files that you need for the current session are retained. Now you can finish with this house-keeping operation and continue to the Combine Command .

5.6.24 Continue to the Combine Command

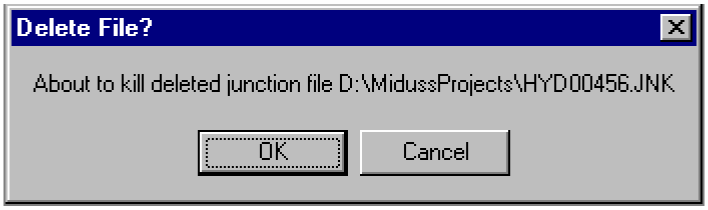
Press the [Continue] button as suggested in the prompts shown at the bottom of the window. The window will be restored to the normal **Combine** format with the [New] and [Add] command buttons re-enabled. You can now proceed as described in the **Hydrograph/Combine** Command.

However, you will have one last chance to recover a junction file that was removed from the list at the end of the MIDUSS session. See Confirm File Deletion.

5.6.25 Confirm File Deletion on Exit

As explained above, the "removed" junction files have merely been renamed and will not be physically erased from your hard disk until the end of the current session.

When you use the **Files/Exit** command to close MIDUSS you will see a message box as shown in Figure 5-31 for each of the removed junctions. If you press the [OK] button the file is erased but pressing [Cancel] will cause the file to be restored as a junction node by changing the extension back to ***.JNC**.



About to kill deleted junction file D:\MidussProjects\HYD00456.JNK

Figure 5-31 – When you finish the session you have a last chance to recover the 'deleted' file

5.6.26 The Copy Inflow to Outflow Command

This command causes the current Inflow hydrograph to be copied to the Outflow hydrograph overwriting the current contents. This may be useful if the current inflow is to be added to a new or existing junction node without the need to use a Design option such as the Pond command or the Pipe & Route commands. It is equivalent to using the **Design/Route** command with a pipe or channel of zero or negligible length.

The action can be reversed by means of the **Hydrograph/Undo** command.

5.6.27 Treatment of a Single Junction

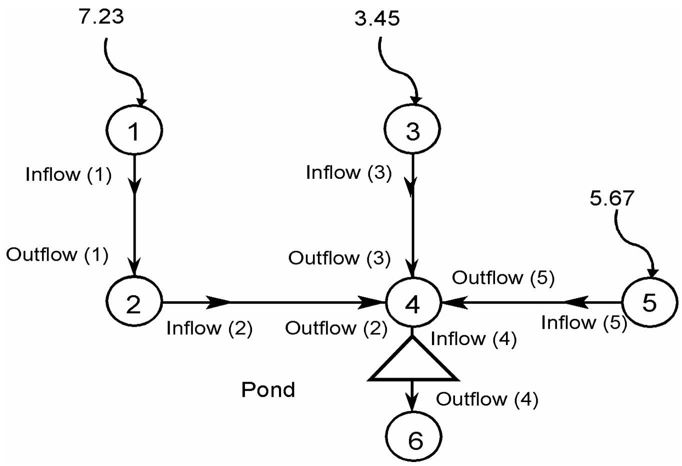


Figure 5.32 - A simple tree network with a single junction

Example 1: This example is for illustration and does not represent an actual MIDUSS design session. Figure 5.32 shows a simple tree network comprising 6 nodes and 5 links. Three branches meet at a single junction node (4). The flows entering at nodes (1), (3) and (5) are assumed to have peak values as listed below and shown in the figure.

Node (1) - Peak flow = 7.23

Node (3) - Peak flow = 3.45

Node (5) - Peak flow = 5.67

The table below illustrates how the **Combine** and **Confluence** commands might be used to analyze this network. For brevity, the design of the links is represented by a combined Pipe/Route command resulting in some small and arbitrary attenuation. Each step in the design process will produce a complete hydrograph that is represented in the table by the peak flow. Note that in order to clarify the arithmetic it is assumed that the peaks are coincident in time. In an actual design the sum of two hydrographs would have a peak slightly less than the sum of the two respective peak flows.

Table 5.1 - Commands and peak flows for Example 1

Action	Runoff	Inflow	Outflow	Junction

Catchment 1	**7.23**		0.00		0.00		0.00
Add Runoff	7.23	- - - >	**7.23**		0.00		0.00
Pipe/Route	7.23		7.23	- - - >	**7.10**		0.00
Next Link	7.23		**7.10**	< - - -	7.10		0.00
Pipe/Route	7.23		7.10	- - - >	**6.90**		0.00
Combine at 4	7.23		7.10		6.90	- - - >	**6.90**
Start New tributary	7.23		**0.00**		6.90		6.90
Catchment 3	**3.45**		0.00		6.90		6.90
Add Runoff	3.45	- - - >	**3.45**		6.90		6.90
Pipe/Route	3.45		3.45	- - - >	**3.20**		6.90
Combine at 4	3.45		3.45		3.20	- - - >	**10.10**
Start New tribute...	3.45		**0.00**		3.20		10.10
Catchment 5	**5.67**		0.00		3.20		10.10
Add Runoff	5.67	- - - >	**5.67**		3.20		10.10
Pipe/Route	5.67		5.67	- - - >	**5.40**		10.10
Combine at 4	5.67		5.67		5.40	- - - >	**15.50**
Confluence at 4	5.67		**15.50**	< - -	< - -	< - -	0.00
Pond	5.67		15.50	- - - >	**10.35**		0.00

The sequence of steps is straightforward and can be summarized as follows.

1. Runoff from sub-area 1 is generated and added to the Inflow
2. Pipe #1 is designed and flow routed to node (2)
3. The outflow from pipe #1 is made the Inflow for Pipe #2 by the Next Link command.
4. Pipe #2 is designed and flow is routed to node (4)
5. Outflow from pipe #2 is stored a junction node (4)
6. The Inflow is set to zero by the Start/New Tributary command
7. Runoff from sub-area 3 is generated and added to the Inflow.
8. Pipe #3 is designed and flow routed to node (4).
9. Outflow from pipe #3 is added to the flow at junction node (4).
10. The Inflow is set to zero by the Start/New Tributary command
11. Runoff from sub-area 5 is generated and added to the Inflow.
12. Pipe #5 is designed and flow is routed to node (4).
13. Outflow from pipe #5 is added to the flow at junction node (4).
14. The total flow at junction node (4) is placed in the Inflow hydrograph (overwriting the previous contents) by using the Confluence command. The junction file is deleted.
15. A detention pond downstream of node (4) is designed and the flow routed to node (6).

5.6.28 Treatment of a Circuited network

Example 2: Figure 5.33 illustrates a circuited network comprising 12 nodes and 11 links. The loop defined by nodes (6)-(7)-(9)-(12)-(11)- (10) is formed as a result of a Diversion structure at link #6 which produces a bifurcation of the flow to nodes (7) and (10).

Overland flow is assumed to enter the system at nodes (1), (2), (3), (4), (8),(9) and (11) with the following peak values.

Node (1) - Peak flow = 1.23

Node (2) - Peak flow = 0.63

Node (3) - Peak flow = 0.56

Node (4) - Peak flow = 2.34

Node (8) - Peak flow = 0.45

Node (9) - Peak flow = 0.75

Node (11) - Peak flow = 0.82

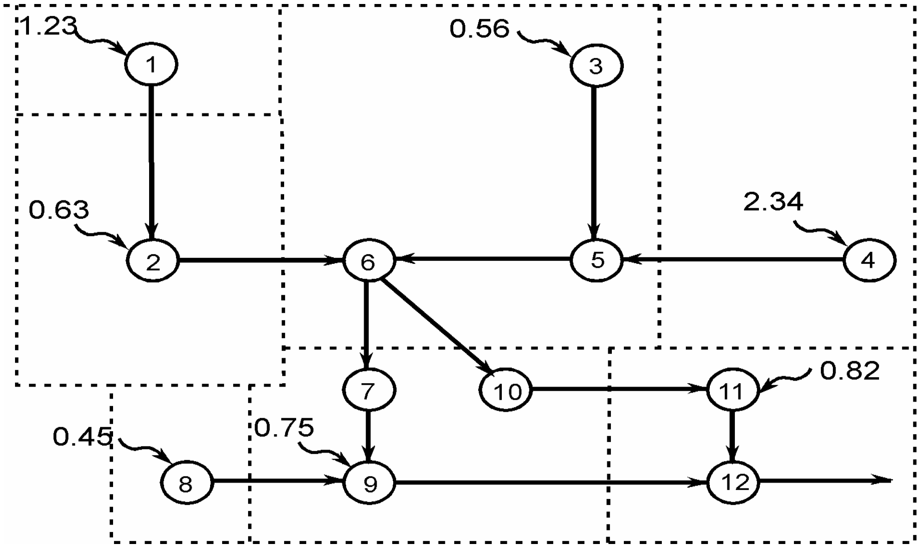


Figure 5.33 - A circuited network with a diversion structure

As in the previous example, all of the hydrographs are represented by their respective peak flow values. Also, the time to peak is assumed to be the same for all hydrographs in order to make the arithmetic more explicit. Refer to the table of commands shown below when reading the following explanation of the design procedure.

Table 5.2 - Commands and peak flows for Example 2

Action	Runoff		Inflow		Outflow		Junction
Catchment 1	**1.23**		0.00		0.00		0.00
Add Runoff	1.23	- - - >	**1.23**		0.00		0.00
Pipe 1/Route	1.23		1.23	- - - >	**1.20**		0.00
Next Link	1.23		**1.20**	< - - -	1.20		0.00
Catchment 2	**0.63**		1.20		1.20		0.00
Add Runoff	0.63	- - - >	**1.83**		1.20		0.00
Pipe 2/Route	0.63		1.83	- - - >	**1.79**		0.00
Combine at 6	0.63		1.83		1.79	- - - >	**1.79**
Start New tributary	0.63		**0.00**		1.79		1.79
Catchment 3	**0.56**		0.00		1.79		1.79
Add Runoff	0.56	- - - >	**0.56**		1.79		1.79
Pipe 3/Route	0.56		0.56	- - - >	**0.52**		1.79
Combine at 5	0.56		0.56		0.52	- - - >	**0.52**
Start New tribute…	0.56		**0.00**		0.52		0.52
Catchment 4	**2.34**		0.00		0.52		0.52
Add Runoff	2.34	- - - >	**2.34**		0.52		0.52
Pipe 4/Route	2.34		2.34	- - - >	**2.29**		0.52
Combine at 5	2.34		2.34		2.29	- - - >	**2.81**
Confluence at 5	2.34		**2.81**	< - -	< - -	< - -	0.00
Pipe 5/Route	2.34		2.81	- - - >	**2.76**		0.00
Combine at 6	2.34		2.81		2.76	- - - >	**4.55**
Confluence at 6	2.34		**4.55**	< - -	< - -	< - -	0.00

The design starts with the design of links #1 and #2 (i.e. the links immediately downstream of nodes (1) and (2)) which, after routing through the pipes, produces a hydrograph with a peak flow of 1.79 entering the junction node (6).

This is stored in the Temporary or Junction hydrograph and also in a file called **`HYD00006.JNC`** - to allow links #3, #4 and #5 to be designed.

The hydrographs from links #3 and #4 meet at a second junction at node (5) which now occupies the Temporary or Junction hydrograph and causes a file **`HYD00005.JNC`** to be created.

After link #5 has been designed, the outflow hydrograph from this link can be added to the one already stored in file **`HYD00006.JNC`** by means of the **Combine** command.

The combined flow hydrograph entering link #6 is obtained by the **Confluence** command which sets the Junction hydrograph to zero and also deletes the file for this junction since it is no longer needed. Note that all the file manipulation is done automatically by MIDUSS and that you need only use the **Combine** and **Confluence** commands consistently - i.e. with the correct junction nodes.

Table 5.2 - (Example 2 continued)

Action	Runoff	Inflow	Outflow	Junction
Diversion at 6	2.34	4.55 - - - >	**1.15**	0.00
(DIV00006.005)	< - -	< - -	< - -	**3.40**
Next Link	2.34	**1.15** < - - -	1.15	0.00
Pipe 7/Route	2.34	1.15 - - - >	**1.09**	0.00
Combine at 9	2.34	1.15	1.09 - - - >	**1.09**
Start New tribute…	2.34	**0.00**	1.09	1.09
Catchment 8	**0.45**	0.00	1.09	1.09
Add Runoff	0.45 - - - >	**0.45**	1.09	1.09
Pipe 8/Route	0.45	0.45 - - - >	**0.41**	1.09
Combine at 9	0.45	0.45	0.41 - - - >	**1.50**
Confluence at 9	0.45	**1.50** < - -	< - - < - -	0.00
Catchment 9	**0.75**	1.50	0.41	0.00
Add Runoff	0.75 - - - >	**2.25**	0.41	0.00
Pipe 9/Route	0.75	2.25 - - - >	**2.21**	0.00
Combine at 12	0.75	2.25	2.21 - - - >	**2.21**
File I_O (DIV00006.005)	- - >	**3.40**	2.21	2.21
Pipe 10/Route	0.75	3.40 - - - >	**3.32**	2.21
Next Link	0.75	**3.32** < - - -	3.32	2.21
Catchment 11	**0.82**	3.32	3.32	2.21
Add Runoff	0.82 - - - >	**4.14**	3.32	2.21
Pipe 11/Route	0.82	4.14 - - - >	**4.08**	2.21
Combine at 12	0.82	4.14	4.08 - - - >	**6.29**
Confluence at 12	0.82	**6.29** < - -	< - - < - -	0.00
Pipe 12/Route	0.82	6.29 - - - >	**6.18**	0.00

Link #6 is a diversion structure with a threshold flow of 0.55 and an overflow ratio of 0.85. Thus the peak outflow from the diversion structure is given by:

$$Q_{out} = 0.55 + (1 - 0.85) \times (4.55 - 0.55) = =0.55 + 0.6 = 1.15$$

The discarded flow with a peak of 3.40 is stored in a file called **`DIV00006.xxx`** which is created by MIDUSS to be recalled by the user at some future point in the design.

Design of link #7 proceeds as usual with the routed flow hydrograph (with a peak of 1.09) being stored at the junction node (9). A new tributary is started at node (8) and link #8 is designed and the outflow added to the hydrograph stored in junction node (9) by means of the **Combine** command.

The total flow entering node (9) (with a peak value of 1.50) is recalled by the **Confluence** command and used to design link #9. Notice that the **Catchment** command at node (9) could have been used before the **Confluence** command, as long as the **Add Runoff** command was used after it.

After generating and adding the overland flow from sub-area 9 and routing the flow through link #9, the outflow hydrograph (2.21 peak flow) is again stored at junction node (12) to allow links #10 and #11 to be designed.

The **Hydrograph/FileI_O** command is used to read the discarded hydrograph (from the diversion structure in link #6) and then places this in the Inflow hydrograph.

Links #10 and #11 are designed in the usual way, with sub-area 11 being added at node (12). The outflow link #11 is added to the hydrograph at node (12). Finally, the **Confluence** command copies the total flow hydrograph into the inflow array to allow design of link #12 to proceed.

This example demonstrates the importance of sketching the layout of the system and numbering the nodes before embarking on the design.

5.6.29 Move Outflow to test Hydrograph

Several commands in MIDUSv2 have a checkbox which causes a user-defined Test hydrograph to be displayed graphically with the other hydrographs computed by the command. This allows some calibration of the MIDUSS model to improve the agreement between a hydrograph observed in the field and the hydrograph computed by MIDUSS.

This command Moves a hydrograph that has been imported into the Outflow hydrograph array to the Test Hydrograph array.

5.6.30 Edit Test Hydrograph

Several commands that display a computed hydrograph have a check box added which causes the current Test hydrograph to be added to the graphical display. A compatible observed hydrograph can be defined at any point during the session by importing a file or by editing the Test hydrograph.

This command displays the current contents of the Test hydrograph in both tabular and graphical mode and allows the user to edit the individual cell-values. In addition, cells can be Inserted or Deleted with automatic adjustment of the hydrograph length, maximum flow value and total volume.

Chapter 6 Working with Files

Programs and data normally reside in your computer's memory only as long as power is supplied. To keep a relatively permanent copy of computer-readable information, the programs, data etc. are stored in some form of physical medium. By far the commonest form is a surface covered with magnetic oxide in the form of permanently installed hard disks or exchangeable, floppy disks. Optical storage devices using recordable CDs are also becoming common. The notion of a file is a collection of information represented in some digital code and which can be read into the memory of the computer when it is needed. Most devices of this type are also capable of creating new files by outputting the information to the disk. The process of reading and writing files in a permanent form is fundamental to the storage, modification and distribution of packages of information.

Although you can use MIDUSS without a detailed knowledge of computer operations, it is recommended that you becomes reasonably well informed with the way in which your computer manipulates files of information. Windows 95, Windows 98 and Windows NT offer some new features concerning the naming of files.

This chapter is concerned with the reading and writing of data files of various types and it is presumed that you have a basic understanding of the conventions for naming files and the folder or directory structure into which they are organized.

The following topics cover general information about file use in MIDUSS.

- Types of Files and where they Reside
- Commands that use Files
- Storage Arrays that Interact with Files
- File Names
- File Formats

Most file operations which you will require can be done using File / Load or Save / Hydrograph Command. Refer to this topic for complete details.

6.1.1 Types of Files and where they Reside

Two types of file are used by MIDUSS. These are MIDUSS system files and job-specific files. MIDUSS system files include the following:

In the Job Folder:

default.out	a default output file used if you do not define a job-specific output file.
Miduss.log	a log of errors trapped during a MIDUSS session.
Main.log	an optional log file created by the license manager
Qpeaks.txt	the data in the peak flows summary table for possible backup use.
Miduss.Mdb…	the Input database for running in Automatic mode.

The following files reside in the MIDUSS folder i.e. C:\Program Files\MIDUSS\ or an alternate location that you defined during installation:

GrParams.dat	default options for line and fill patterns and colours in the Show/Graph window.
LagCurve.dat	an empirical curve used in the Lag & Route command
*.mrd…..	a mass rainfall distribution such as Huff2.mrd or SCS_Type3_24hr.mrd

Other files will reside in the current default job directory and, in addition to the list of 5 above, will generally comprise either storm hyetographs or flow hydrographs. These will normally have the extension of *.stm or *.hyd for easy identification when using the File I/O command.

Two special types of hydrograph file should be mentioned. Files of the general form DIV *****.hyd are hydrograph files created by the Diversion command. Files with the extension *.JNC are junction files created by the Combine command.

After acceptance of the Storm command you are prompted to enter a descriptor that will be added to the name of any hydrograph file created to allow it to be associated with a specific storm event.

6.1.2 Commands that use Files

Almost all commands write information to the current Output file in order to allow creation of an Input database file for use in Automatic mode. Apart from this, only certain MIDUSS commands make use of files. These are listed below for each of the commands which use them.

Menu Command	**Menu item**	**File name**
File	Open input Database	Miduss.Mdb
File	Output file…	*.out
File	Save Database As	*.Mdb, *.stm
Hydrology	Storm	*.mrd
Hydrograph	Combine	*.jnc
Hydrograph	Confluence	*.jnc
Design	Design/Log	Design.log
Design	Diversion	DIV?????.hyd
Show	Design Log	Design.log
Show	Output File	*.out
Show	Flow Peaks	Qpeaks.txt
Show	Graph	*.bmp
Automatic	Create Input Database	*.out, Miduss.Mdb
Automatic	Edit Input Database	Miduss.Mdb
Automatic	Save Database As	Miduss.Mdb
Automatic	Run input Database	Miduss.Mdb
Tools	Add Comment	*.out
Tools	Notepad	*.txt, *.*
Tools	WordPad	*.doc, *.*
Help	Contents	*.hlp, *.cnt
Help	Tutorials	Tutorial*.exe

Note that Tutorial files on the MIDUSS CD are read-only files.

6.1.3 Storage Arrays that Interact with Files

When you define the time parameters you set up the size of storage arrays to hold the hyetographs and hydrographs for the current MIDUSS session. For example

Nstm = Maximum Storm Length / Time step

Nhyd = Maximum hydrograph / Time Step

The arrays used in MIDUSS are then set up as follows.

Name	**Size**	**Content**
RainTemp()	Nstm	Temporary storm until accepted
Rain()	Nstm	Defined storm
RainEffI()	Nstm	Effective rain on Impervious fraction
RainEffP()	Nstm	Effective rain on Pervious fraction
OvHyd()	Nhyd	Total runoff
OvHydI()	Nhyd	Runoff from impervious area
OvHydP()	Nhyd	Runoff from Pervious area
Inflow()	Nhyd	Inflow to a facility
Outflow()	Nhyd	Outflow from a facility
TempHyd()	Nhyd	Temporary or Junction hydrograph
BkupHyd()	Nhyd	Backup to allow "Undo" recovery.

6.1.4 File Names

The Microsoft Windows operating systems offer great flexibility in the way files can be named. File names have only a few "illegal" characters, can include spaces, can have more than one 'period' separator and can be longer than the 11 character "nnnnnnnn.eee" format used in Microsoft DOS.

The following characters are not allowed.

double quote	"
back slash	\
forward slash	/
colon	:
asterisk	*
question mark	?
less than	<
greater than	>
vertical/pipe	|

See the Hydrology/Storm Descriptor topic in Chapter 2 for information on applying a special storm-specific tag to hydrograph names. This allows files in the same directory or folder to describe hydrographs at the same node but which result from different storm events.

Although it is tempting to use long and self-explanatory file names, try to make names reasonably unique in the first 8 characters. This will ensure that if the file names are displayed in an application which expects DOS names you do not get a list of files differentiated only by the "~1" or "~2" following of the first 6 characters of the original name.

6.1.5 File Formats

Storm hyetograph and flow hydrograph files are simple ASCII (i.e. text) files that can be read, edited and created using a text editor such as Notepad. Each file contains a total of (N+6) records comprising 6 records of file header information followed by N records each of which holds a single value of rainfall intensity or hydrograph flow rate.

The file format is:

Record	**Content**	**Example**
1	Description	"Hydrograph #1"
2	File type	4
3	Peak value	"0.441"
4	Gravity	9.81
5	Time step	5
6	No. of values	36
7	Value 1	0
8	Value 2	3.970632E-03
:		
N+6	Value N	7.172227E-04

where

Description is an alphanumeric string enclosed in ".."

File Type is an integer value:

1 - 3	for	Storm, Impervious and Pervious effective rainfall hyetographs
4 - 7	for	Runoff, Inflow, Outflow and Temporary (or Junction) hydrograph

Peak value is the maximum ordinate expressed as a string

Gravity = 9.81 for metric or SI and 32.2 for imperial or U.S. Customary units

Time step is nominal time step in minutes

No. of values is the number of records following the header records.

Mass Rainfall Distribution files have only two header lines. The file format is:

Record	**Content**	**Example**
1	Description	"Hydrograph #1"
2	No. of values	21
3	Value 1	0.000
4	Value 2	0.063
:		
N+2	Value N	1.000

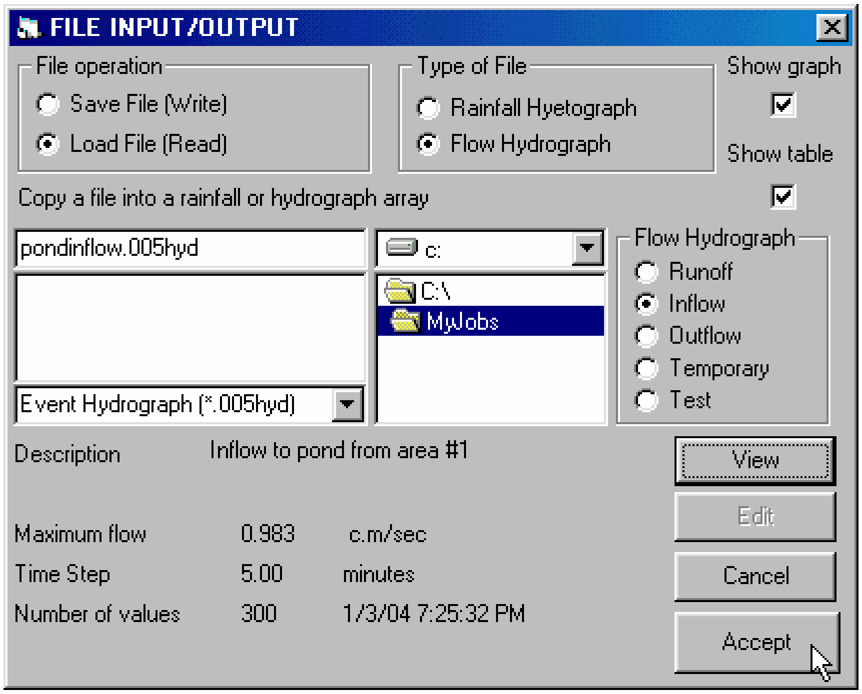
The first and last values of an **.mrd** file are always 0.000 and 1.000 respectively. Usually the number of values is odd defining an even number of increments. The files **`Huff.mrd`** have 21 values; the SCS storm files **`SCS_Type3_24hr.mrd`** have 97 values.

If you want to create a new file it is important that you follow the defined format. The simplest way is to make a copy of an existing file of the same type in Notepad and edit the values as required. You can quickly test the integrity of such a file by importing it into an appropriate array using the File / Load file command and then display it using either the **Show/QuickGraph** or **Show/Tabulate** commands.

New ***.mrd** files can be prepared with the Edit Storm Tool.

(Hint: Notepad may add its default extension '.txt' to a file you have edited thus creating – for example – a file called PondInflow.005hyd.txt. Check the folder if in doubt.)

6.2 The File Input / Output Command



File operation
○ Save File (Write)
● Load File (Read)

Type of File
○ Rainfall Hyetograph
● Flow Hydrograph

Show graph ☑

Show table ☑

Copy a file into a rainfall or hydrograph array

pondinflow.005hyd

c:

C:\
MyJobs

Event Hydrograph (*.005hyd)

Flow Hydrograph
○ Runoff
● Inflow
○ Outflow
○ Temporary
○ Test

Description Inflow to pond from area #1

Maximum flow 0.983 c.m/sec
Time Step 5.00 minutes
Number of values 300 1/3/04 7:25:32 PM

Figure 6-1 – The File Input-Output window.

This form is not directly accessible from the Hydrograph menu. Instead, it is called by the File/Load file/ and File/Save file/ in the File menu.

This general form lets you read or write disk files from or to any of the three rainfall hyetograph arrays or the four hydrograph arrays. For any selected file the header information is displayed. If the file is not recognized by MIDUSS you can use a text editor to view the file assuming it is in ASCII code. Options are available to let you display the contents of the file as a Graph or in Tabular form or in both modes.

The following topics describe specific parts of the File I/O window and provide a logical sequence of steps to use the command.

- The Hydrograph/File I/O menu
- The FileIO window
- The File Operation Options
- The File Type Options
- The Rainfall Hyetograph Options
- The Flow Hydrograph Options
- Choosing a Drive and Directory
- Setting the File Name Filter
- Selecting and Editing the File name
- Pre-viewing the File Header

Some of the above information is automatically detected by the File Open or File Save dialogue boxes that are opened by the File/Load file or File/Save file commands prior to opening the File Input/Output form. However, for completeness the various steps are described in detail in the sections that follow.

6.2.1 The File / Save and Load file menu

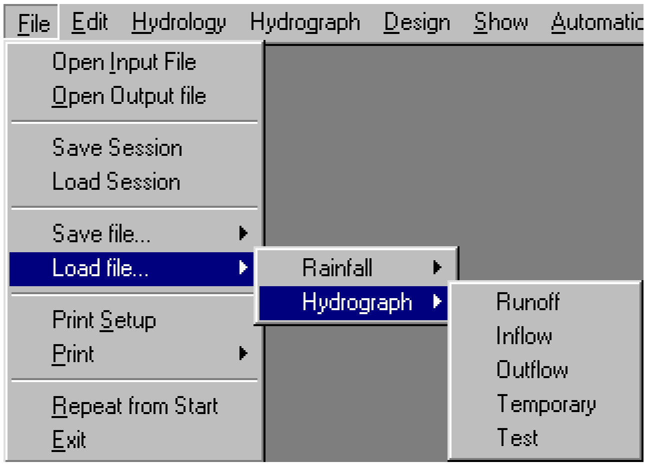


Figure 6-2 – The initial state of the Hydrograph menu

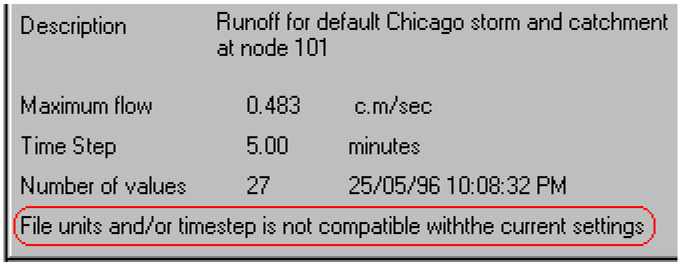
The File menu includes the Save file and Load file items.

Both of these cause the File Open or File Save dialogue box to be opened to let you define the file to be processed. With this information, the File Input/Output window is then partially populated with the data and opened.

Selection of the Hydrograph menu is not enabled until you have specified the system of units to be used and the time parameters. Even then, as shown here, only the Start and File I/O options are enabled. Other options are enabled as various hyetograph and hydrograph arrays are created during the design session.

Each time you attempt to load a file, MIDUSS checks to make sure that the file contents are compatible with the current system parameters and it is for this reason that you are required to pre-define these values before attempting to import a file.

If you select a file which is recognized by MIDUSS but which is not compatible with the current units or time step the information is displayed in the lower part of the window. The [View] button is enabled when you select a file but if you press the [View] button a warning message is displayed below the file header information. This is circled in red in Figure 6-3 below.

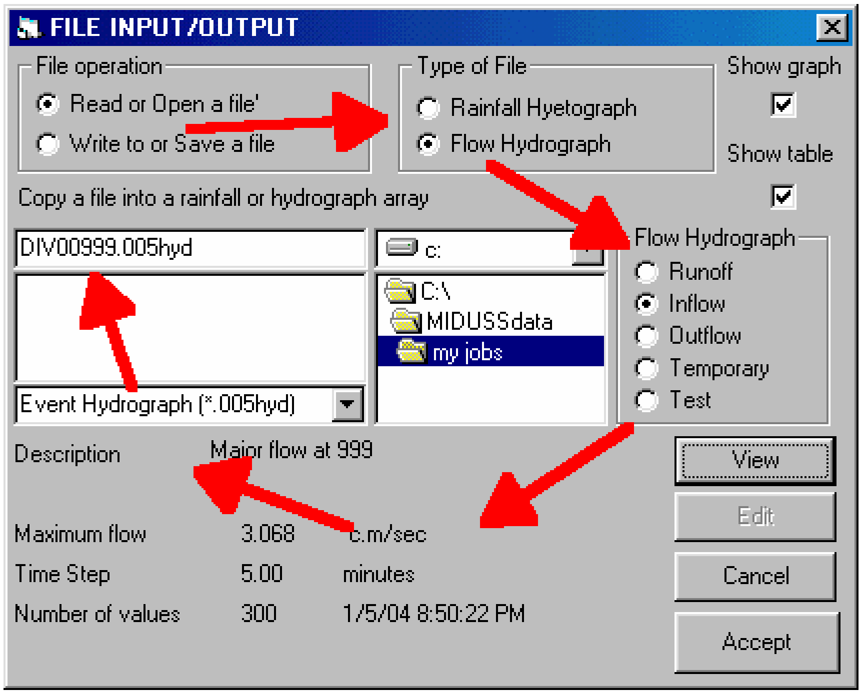


Description	Runoff for default Chicago storm and catchment at node 101	
Maximum flow	0.483	c.m/sec
Time Step	5.00	minutes
Number of values	27	25/05/96 10:08:32 PM

File units and/or timestep is not compatible withthe current settings

Figure 6-3 – MIDUSS checks that file characteristics are compatible.

6.2.2 The File Input / Output Window


FILE INPUT/OUTPUT

File operation
● Read or Open a file'
○ Write to or Save a file

Type of File
○ Rainfall Hyetograph
● Flow Hydrograph

Show graph ☑
Show table ☑

Copy a file into a rainfall or hydrograph array

DIV00999.005hyd
c:
C:\
MIDUSSdata
my jobs
Event Hydrograph (*.005hyd)

Flow Hydrograph
○ Runoff
● Inflow
○ Outflow
○ Temporary
○ Test

Description Major flow at 999
Maximum flow 3.068 c.m/sec
Time Step 5.00 minutes
Number of values 300 1/5/04 8:50:22 PM

Figure 6-4 – The required steps proceed logically in a clockwise direction

The red arrows of Figure 6-4 suggest that you proceed through the various options on the File I/O window in a clockwise direction. First select the file operation required (input or output, read or write), then choose the file type (hyetograph or hydrograph). The latter choice will cause the appropriate set of options to be displayed from which you can select a hyetograph or hydrograph array. Next you accept or define the device and directory or folder in which the file exists or is to be stored. From here, proceed to the File filter to select the appropriate file extension that in turn will cause the appropriate files to be displayed. Finally you can define a filename - either that of an existing file, an edited version of an existing file or a completely new file name.

Each of these steps is discussed in the topics that follow. The first step is to decide on the File Operation option.

6.2.3 The File Operation Options

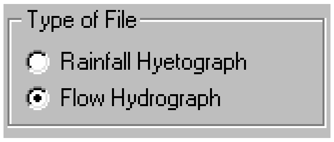

File operation
● Read or Open a file'
○ Write to or Save a file

Figure 6-5 – The first step is to select a Read or Write operation

The initial default choice is to write a file to the disk. Click on the desired option. If you change this option the message displayed below the frame changes to confirm the type of operation.

The next step is to decide on the type of file.

6.2.4 The File Type Options


Type of File

○ Rainfall Hyetograph

● Flow Hydrograph

Figure 6-6 – Next, choose between hyetographs and hydrographs

The initial default option is for a rainfall hyetograph and the three arrays holding a hyetograph are displayed. Changing the choice causes the appropriate set of arrays to be displayed.

If the selected options are to write a rainfall hyetograph file to the Storm array and you have not yet defined a storm you will see the warning message shown below.

The Storm array may be empty or may have been made obsolete by a previous operation

Figure 6-7 – The initial defaults may not be appropriate if no storm has been defined

This simply warns you that there is no information to write. The mouse pointer will be conveniently placed on the [OK] button and you can either click the primary (left) mouse button or press the space bar to close the message box.

The next step depends on the choice made here. You will see either a set of three options for Rainfall hyetographs or, if you have selected Flow hydrographs as the file type, you will see a set of four Flow hydrograph options.

6.2.5 The Rainfall Hyetograph Options


Rain Hyetograph

● Storm

○ Impervious

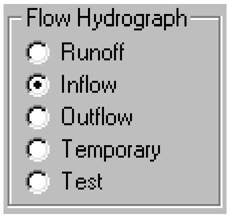
○ Pervious

Figure 6-8 – Select which hyetograph to use.

The initial default option is for the Storm hyetograph. If you have selected to read a file into a rainfall array, all three of the options will be enabled. However, if you intend to write a rainfall hyetograph file only the options corresponding to arrays that have finite data will be enabled. If you have used the Hydrology/Storm command the Storm option will be enabled. If you have used the Hydrology/Catchment command the other two options will also be enabled.

If you want to review the Flow hydrograph options refer to the next topic, otherwise proceed to the next logical step which is to confirm or select the drive and directory to be used..

6.2.6 The Flow Hydrograph Options



Flow Hydrograph

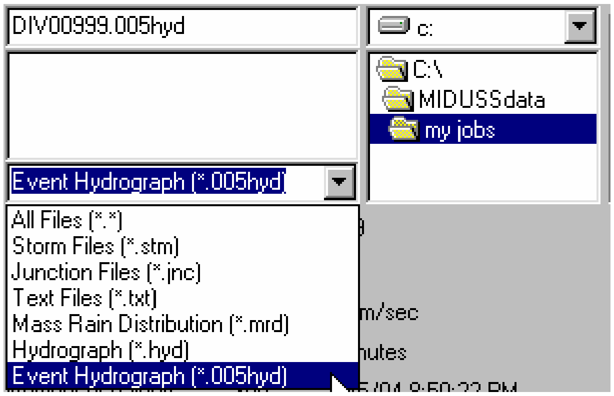
○ Runoff
● Inflow
○ Outflow
○ Temporary
○ Test

Figure 6-9 – All four hydrographs are available for a file Read operation.

The initial default option is for the Runoff hydrograph. If the File operation is set to read a file into one of these arrays, all four options will be enabled. However, if you want to write a hydrograph file only the options that correspond to an array containing finite data will be enabled. Clearly this depends on the previous actions taken during the design session.

Now proceed to the next logical step which is to confirm or select the drive and directory to be used..

6.2.7 Choosing a Drive and Directory



DIV00999.005hyd

c:

C:\
MIDUSSdata
my jobs

Event Hydrograph (*.005hyd)

Figure 6-10 – The default drive and folder should be your Job directory

The drive selector and the default directory are set equal to the directory containing the current output file. If you have defined a job specific output file (which is recommended) it will be shown here. If you didn't have time to define a job directory and output file the output will be directed to a file called "default.out" which resides in the Miduss98 directory. During installation this will normally default to C:\Program Files\Miduss98\ unless you have elected to load MIDUSS to another directory of your choice (for instance if your C: drive is getting really full.)

Selecting a drive and directory is done easily by first selecting the drive (i.e. device) and then navigating to the desired directory. The File I/O command presently does not support devices and directories (or folders) on a network.

The next step is to select an appropriate file name filter (or file extension) which will limit the number of files displayed in the file list box.

6.2.8 Setting the File Name Filter

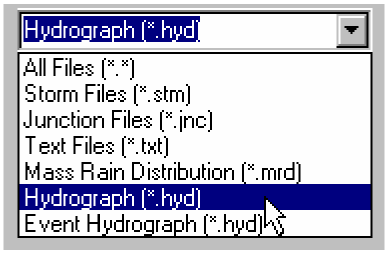


Figure 6-11 – Select the appropriate filter for the type of file being used.

If you have selected a file type of Rainfall hyetographs the default filter will be set to *stm. If the file type is a Flow hydrograph the default filter is *.hyd as shown in Figure 6-11. If a storm descriptor has been defined, it will be added to the Event Hydrograph filter and this will be the default for hydrograph operations.

To change the filter you should click on the down arrow at the right side of the Filter box to open the list. The available options are shown in Figure 6-11. Click the mouse pointer on the desired filter as shown here and it will be selected and the drop-down list box will be closed.

The final step is to select or edit or otherwise define the file name you want to use.

6.2.9 Selecting and Editing the File name

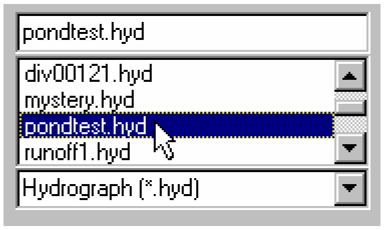


Figure 6-12 – You can select an existing hydrograph to read or over-write

Figure 6-12 illustrates a situation in which the user wants to read a flow hydrograph. Typically this is useful when a flow hydrograph has been created in a previous design session and you now want to refine the design of pond or other facility.

If you intend to write a file from an existing array of data you can either:

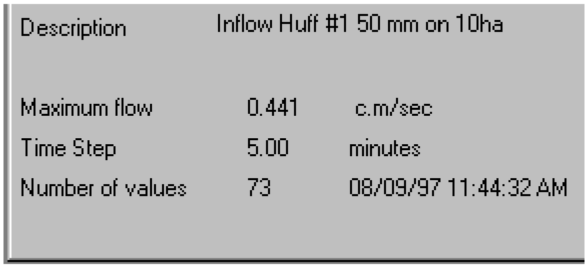
- Select an existing file which you want to overwrite,
- Select an existing file name and then edit it to create a new file, or
- Type in the name of the file you want to create.

Hint: To add a filename to the default '*.hyd' do the following:

- Click on the default name to highlight it
- Press [Home] on the keyboard to move the text entry point to the left end of the text box
- Press [Delete] on the keyboard to remove the asterisk '*'.
- Type in the desired file name.

This completes the choices to read or write a file. The last step is to review the contents of the file header.

6.2.10 Pre-viewing the File Header



Description	Inflow Huff #1 50 mm on 10ha	
Maximum flow	0.441	c.m/sec
Time Step	5.00	minutes
Number of values	73	08/09/97 11:44:32 AM

Figure 6-13 – Typical file header information.

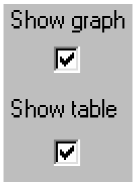
This portion of the window will also vary depending on whether the file operation is to read or write a file. Figure 6-13 shows the situation when a hydrograph file is to be read in (i.e. imported) to an array. The main difference is in whether a description is already known for an existing file (as shown here) or, if a new file is to created, the description must be typed into a text box.

The other data show the maximum flow rate or rainfall intensity, the time step, the system of units employed (as indicated by the legend following the peak value - e.g. c.m/sec or c.ft/sec) and the number of values to be read or written. If the file already exists the date and time of its creation are also shown.

If you are creating a new file you are well advised to type in a meaningful description. Use two rows if you wish. The only restriction is that you cannot use the double quote character (") often used as the symbol for inch because this is used as the delimiter for the description. If you want to use a symbol for inch use two single quotes.

To see the properties of the file being read or written press the [View] command button.

6.2.11 Using the [View] Command



Show graph ☑

Show table ☑

Figure 6-14 – You can choose to show or suppress the graph and table.

The actual file operation is carried out by pressing the [Accept] button but before this is enabled you must press the [View] button. If the appropriate check boxes are ticked MIDUSS displays a graph and a tabular display of the hyetograph or flow hydrograph. If you are reading an existing file the properties are displayed prior to reading it into the requested array. If you are writing a new file from a selected array the graph and table show the array to be written. This can be an alternate way to view the properties of the current hyetographs and flow hydrographs generated during the design session. See the **Show/Quick Graph** and **Show/Tabulate** commands for other methods.

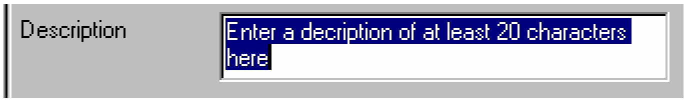
Note that whereas the title bar of the table corresponds to file type, the Title bar of the graph display shows "Backup" for a file being read in or imported. This is because the Backup array is used as temporary storage until you confirm that the file can be accepted.

If a file is being read in you may see a warning message advising you that the hyetograph or hydrograph will be "padded with zeros" or "will be truncated". These cases will occur if you are importing a file with fewer elements than can be stored in the current array sizes, or conversely, if the array size is not able to hold the total file size. In the latter case you can find out if this is significant by dragging up the top edge of the Table containing the file which is to be read.

When you are satisfied that all is well you can press the [Accept] button to accept the file Operation.

6.2.12 Accepting the File Operation

Before a new file is created you are prompted to enter a description of at least 20 characters.

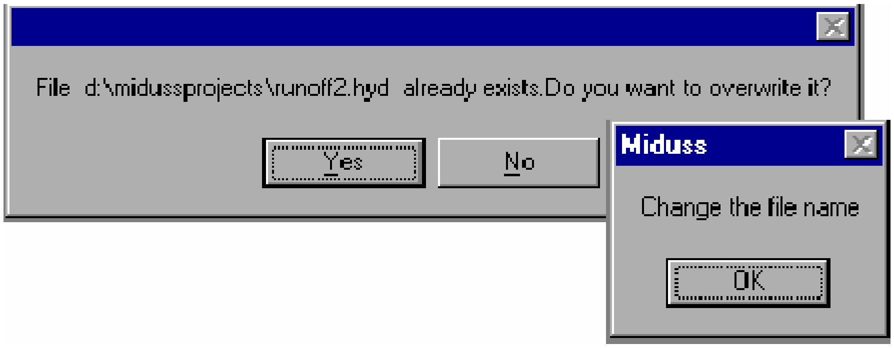


Description: Enter a decription of at least 20 characters here

Figure 6-15 – It is worth while to provide a detailed description of the file

Use as much space as you need to provide a clear and unambiguous description as the data entry text box will support multiple lines (although you can see only two lines at a time.

When you press the [Accept] button you may see a warning message as shown below advising you that MIDUSS has detected a file of the same name and directory as the one you are attempting to write.



File d:\midussprojects\runoff2.hyd already exists.Do you want to overwrite it?

Miduss

Change the file name

Figure 6-16 – MIDUSS protects you from accidental over-writes.

If you press the [Yes] button the operation will continue and the existing file will be overwritten. If you want to save the existing file press [No]. If you press [No] you will see the message box on the right of Figure 6-16 and on clicking on the [OK] button or pressing the space bar the File name data entry box will be re-activated and you can edit the name for the new file.

Notes:

Chapter 7 Hydrological Theory

The hydrology incorporated in MIDUSS is based on relatively simple and generally accepted techniques. There are four commands to control the fundamental operations.

1. STORM - This command allows you to define a rainfall hyetograph either of the synthetic, design type or a historic storm. It should be noted that as an alternative to using the STORM command, a previously defined rainfall hyetograph may be read in from a disk file, by means of the File/Load file/Storms/ command.

2. CATCHMENT - This command prompts you to define a single sub-catchment and computes the total overland flow hydrograph for the currently defined storm. The runoff hydrographs from the pervious and impervious areas are computed separately and summed. The roughness, degree of imperviousness and surface slope of both the pervious and impervious fraction are defined in this command. The effective rainfall on these two fractions is computed and stored for future use. The runoff hydrograph from the two fractions is computed separately and added to give the total runoff.

3. LAG and ROUTE . This command is useful for modelling the runoff from very large sub-catchments without having to resort to specifying unrealistically long overland flow lengths. The command computes the lag time in minutes of a hypothetical linear channel and linear reservoir through which the runoff hydrograph is routed. Typically this results in a smaller, delayed runoff peak flow.

4. BASE FLOW - This command lets you specify a constant value of base flow to be added to the current inflow hydrograph.

Some details of the hydrological techniques used in these commands are given in the detailed discussion which follows. If further information is required, reference should be made to any standard text on the subject. See the references for other suggested reading.

7.1 Theory of Design Storms

For those readers who wish a more detailed description of the methods used to define design storm hyetographs, this section contains a brief theoretical derivation of three of the methods described previously in the Hydrology/Storm menu command.

The following sections are provided:

- Derivation of the Chicago Storm
- Using a Mass Rainfall Distribution
  - Derivation of the Huff Storm
  - Derivation of the User Defined Storm
- Derivation of the Canadian 1-hour storm

7.1.1 Derivation Of The Chicago Storm

The synthetic hyetograph computed by the Chicago method is based on the parameters of an assumed Intensity-Duration-Frequency relationship, i.e.

[7.1] $$i = \frac{a}{\left(t_d + b\right)^c}$$

where *i* = average rainfall intensity (mm/hr or inch/hr)

td = storm duration (minutes)

a,b,c = constants dependent on the units employed and the return frequency of the storm.

The asymmetry of the hyetograph is described by a parameter *r* (where $0 < r < 1$) which defines that point within the storm duration td at which the rainfall intensity is a maximum.

Imagine a rainfall distribution (with respect to time) such as that shown by the dashed curve of Figure 7-1, *i.e.* with a maximum intensity *imax* at the start of rainfall at *t=0*, which then decreases monotonically with elapsed time *t*, according to some function *f(t)* which is, as yet, unknown. If the duration of such a storm is *td* then it is easy to see that the total volume of rainfall is represented by the area under the curve from *t=0* to *t=td.* The average rainfall intensity for such an event could be estimated as *iave = Volume/td* as illustrated by the shaded rectangle of Figure 7-1.

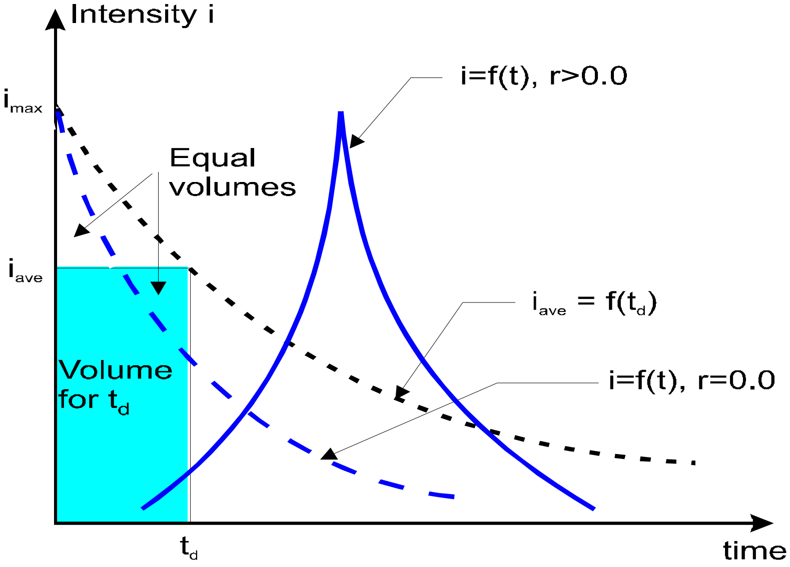


Figure 7-1 - Development of the Chicago storm

Several storms with different durations *td* but with the same time distribution of intensity would produce values of *iave* which decrease as *td* increases, leading to the dotted curve of Figure 7-1. Thus:-

[7.2] $$i_{ave} = \frac{Volume}{t_d} = \frac{1}{t_d}\int_0^{t_d} f(t)dt$$

If the average intensity *iave* over an elapsed time *t* can be described by an empirical function such as equation [7-1], then by combining [7-1] and [7-2], the functional form of *f(t)* can be obtained by differentiation, i.e.

[7.3]

$$f(t) = \frac{d}{dt}\left[\frac{at_d}{\left(t_d + b\right)^c}\right]$$

or

[7.4]

$$i = f(t) = \frac{a\left[\left(1-c\right)t_d + b\right]}{\left(t_d + b\right)^{1+c}}$$

Now if the value of r is in the range $0<r<1$ the time to peak intensity for a given duration is $tp = r.t$ The time distribution of rainfall intensity can then be defined in terms of time after the peak $ta = (1-r).t$ and time before the peak $tb = r.t$ by the following two equations.

[7.5]

$$i_a = \frac{a\left[\frac{\left(1-c\right)t_a}{\left(1-r\right)} + b\right]}{\left(\frac{t_a}{\left(1-r\right)} + b\right)^{1+c}}$$

[7.6]

$$i_b = \frac{a\left[\frac{\left(1-c\right)t_b}{r} + b\right]}{\left(\frac{t_b}{r} + b\right)^{1+c}}$$

The solid curve of Figure 7-1 shows the time distribution of rainfall using a value of r greater than zero ($r = 0.4$ approximately).

Calculation of the discretized rainfall hyetograph is carried out by integrating these equations to obtain a curve of accumulated volume as illustrated in Figure 7-2 below. For convenience this curve is computed so that volume V is zero at $t=tp$ and is defined in terms of the elapsed time after and before tp. The expressions for volume after and before tp are then given by equations [7-7] and [7-8] respectively.

[7.7]

$$V_a\left(t_a\right) = \frac{a.t_a}{\left(\frac{t_a}{r-1} + b\right)^c} = \frac{a\left(t - t_p\right)}{\left(\frac{t_p - t}{1-r} + b\right)^c}$$

[7.8]

$$V_b\left(t_b\right) = \frac{a.t_b}{\left(\frac{t_b}{r} + b\right)^c} = \frac{a\left(t_p - t\right)}{\left(\frac{t_p - t}{r} + b\right)^c}$$

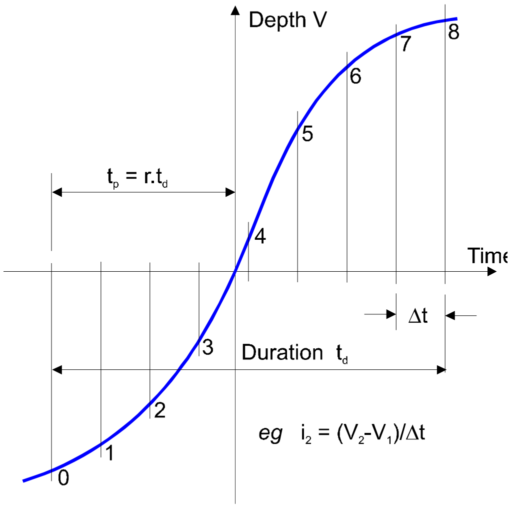


Figure 7-2 – Discretization of an integrated volume curve.

Discretized values of rainfall intensity can now be obtained by defining a series of 'slices' of equal timestep *Δt*. The time step at the peak intensity is positioned relative to the *tp* position so that it is disposed about the peak in the ratio *r* to *(1-r)*. In general, this means that the commencement of the storm may not be precisely defined by t = - r td and the storm duration is therefore not disposed about the peak exactly in the ratio *r* to *(1- r)*. However because the rainfall intensities at the extremities of the storm are generally very small this approximation is unlikely to lead to significant error.

7.1.2 Storm From a Mass Rainfall Distribution Curve

This method is used both for the four quartile Huff distributions and also for the user defined Mass Rainfall Distribution function. These are treated very similarly but are discussed separately in the two sections that follow.

7.1.3 Derivation of the Huff Storm

Based on data from watersheds in the mid-western USA, Huff (see References) suggested a family of non-dimensional, storm distribution patterns. The events were divided into four groups in which the peak rainfall intensity occurs in the first, second, third or fourth quarter of the storm duration. Within each group the distribution was plotted for different probabilities of occurrence. MIDUSS uses the median curve for each of the four quartile distributions. The non-dimensional curves are illustrated in Figure 7-3 below and are tabulated in Table 7-1.

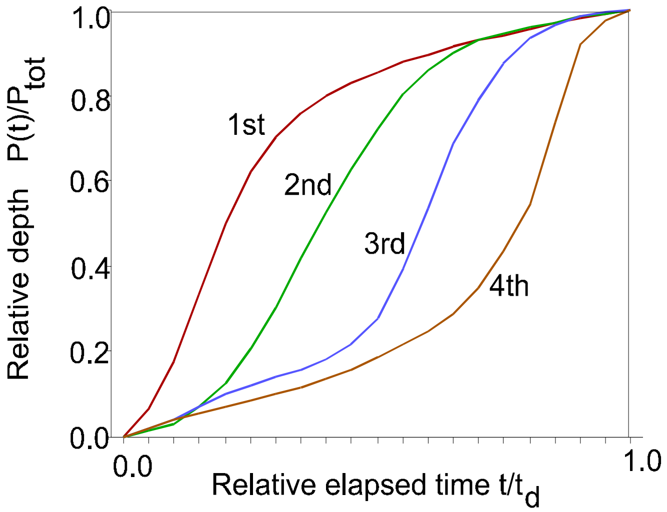


Figure 7-3 - Huff's four storm distributions.

To define a storm of this type you must provide values for the total depth of rainfall (in millimetres or inches), the duration of the storm (in minutes) and the quartile distribution required (i.e. 1, 2, 3 or 4). The duration must not exceed the maximum storm duration defined in the **Hydrology/Time parameters** menu command and, as with the Chicago storm option, an error message is displayed if this constraint is violated. Once the parameter values have been entered and confirmed by pressing the [Display] command button, the hyetograph is displayed in both graphical and tabular form. You can experiment by altering any of the data - even the type of storm - and re-using the [Display] button until you press the [Accept] button to save the storm and close the Storm command.

The four quartile Huff distributions are approximated by a series of chords joining points defined by the non-dimensional values in the table referenced below. Figure 7-4 shows a typical curve (not to scale) which for clarity uses only a very small number of steps. The time base for the NH dimensionless points defining the ‘curve’ is subdivided into dimensionless time steps defined by:

[7.8] $$\Delta\tau = \frac{(NH - 1)}{NDT}$$

where NH = number of points defining the Huff curve (shown as NH = 7 but usually much more)

NDT = number of rainfall intensities required (shown as only 15 in Figure 7-4).

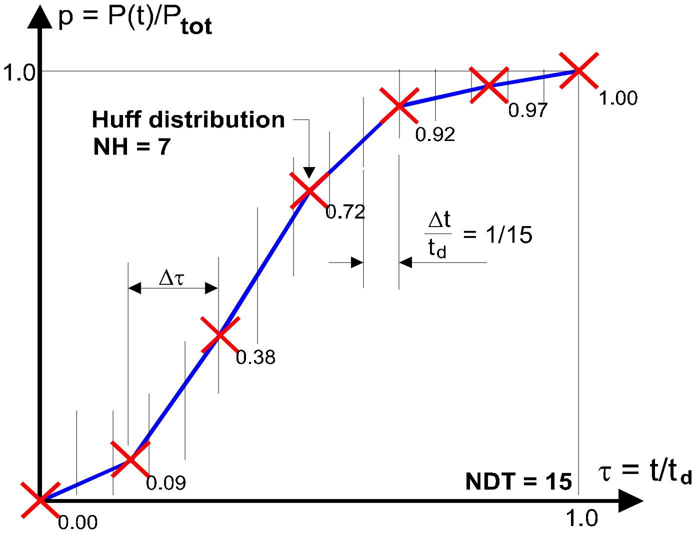


Figure 7-4 - Discretization of a Huff curve.

The values of the dimensionless fractions P_k and P_{k+1} at the start and finish of each time-step are obtained by linear interpolation and the corresponding rainfall intensity is then given as:

$$[7.10] \qquad i_j = \frac{(p_{k+1} - p_k)P_{tot}}{\Delta t}$$

where

$$p_{k+1} = p_m + (p_{m+1} - p_m)(h - m)$$

$$m = INT(h)$$

$$h = j.\Delta\tau + 1$$

For the example shown in Figure 7-4, the Huff 2nd quartile curve is approximated by NH=7 points with a storm duration which is divided into 15 time steps. Then $\Delta\tau$ = (7-1)/15 = 0.4. The calculation of the rainfall fractions P_{k+1} required for eq. [7.10] is then carried out as shown in the table below.

J	1	2	3	4	5	…	12	13	14
h=j Δτ+1	1.4	1.8	2.2	2.6	3.0		5.8	6.2	6.6
M	1	1	2	2	3		5	6	6
Pm	0.00	0.00	0.09	0.09	0.38		0.92	0.97	0.97
pk+1	0.036	0.072	0.148	0.264	0.340		0.960	0.976	0.988

7.1.4 Derivation of the user defined storm

The user defined Mass Rainfall Distribution function is defined in exactly the same way as for the Huff storm but with the difference that the number of points is not limited to 51 to define the curve. Since the initial and final points must have values of 0.00 and 1.00 respectively this means that there can be up to (Npts-1) equally spaced segments in the definition of the Mass Rainfall curve.

A number of popular rainfall distributions are described in *.mrd files which are included with MIDUSS. These include the various SCS 24-hour and 6-hour storms and regulatory storms such as Hurricane Hazel (Southern Ontario, Canada) and the Timmins storm for more northerly parts of Ontario. These files are simple text files and can be printed out to show the values used.

7.1.5 P(t)/Ptot for Four Huff Quartiles

t/td	**P(t)/Ptot for quartile**			
	1st	**2nd**	**3rd**	**4th**
0.00	0.000	0.000	0.000	0.000
0.05	0.063	0.015	0.020	0.020
0.10	0.178	0.031	0.040	0.040
0.15	0.333	0.070	0.072	0.055
0.20	0.500	0.125	0.100	0.070
0.25	0.620	0.208	0.122	0.085
0.30	0.705	0.305	0.140	0.100
0.35	0.760	0.420	0.155	0.115
0.40	0.798	0.525	0.180	0.135
0.45	0.830	0.630	0.215	0.155
0.50	0.855	0.725	0.280	0.185
0.55	0.880	0.805	0.395	0.215
0.60	0.898	0.860	0.535	0.245
0.65	0.915	0.900	0.690	0.290
0.70	0.930	0.930	0.790	0.350
0.75	0.944	0.948	0.875	0.435
0.80	0.958	0.962	0.935	0.545
0.85	0.971	0.974	0.965	0.740
0.90	0.983	0.985	0.985	0.920
0.95	0.994	0.993	0.995	0.975
1.00	1.000	1.000	1.000	1.000

7.1.6 Canadian 1-Hour Storm Derivation

Recent work by Watt *et al* (see references) has suggested a simple two parameter design storm which has a linear rising portion followed by an exponentially decreasing recession curve. Watt *et* *al* also suggest the possibility of reversing the linear and exponential segments but this option is not supported by MIDUSS. Figure 7-5 shows a definition sketch of this design hyetograph.

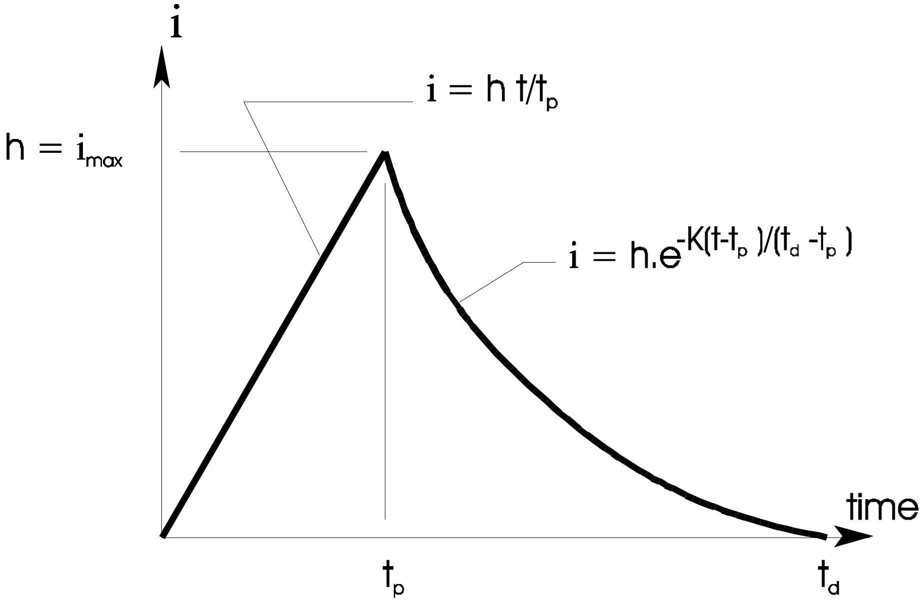


Figure 7-5 - The Canadian AES 1-hour design storm.

The parameter values which you are prompted to supply for this option are the total depth of rainfall (millimetres or inches), the duration in minutes, the time to peak intensity (minutes) *tp* and the decay coefficient *K*. The decay coefficient K is usually in the range 5 to 7. As with the other options, the maximum duration and the time step used are as defined in the **Hydrology/Time parameters** menu command.

It should be noted that the design storm as suggested by Watt *et al* is intended to be used for storms of 1 hour duration only, since the data used for the work was limited to this duration. However, MIDUSS allows you to define other values of duration. Care should be taken if suggested values for the time to peak are taken from Watt *et al* as these are intended specifically for 60 minute storms.

The rising and falling limbs of the hyetograph suggested by Watt *et al* are defined by equations [7-11] and [7- 12] respectively.

$$[7.11] \qquad i = h\frac{t}{t_p} \qquad \text{for } t < t_p$$

$$[7.12] \qquad i = e^{-K\frac{(t-t_p)}{(t_d-t_p)}} \qquad \text{for } t > t_p$$

For specified values of the parameters t_d, t_p, K and the total depth of rainfall $Rtot$ the peak intensity h (see Fig. 7-5) can be obtained as follows.

$$[7.13] \qquad h = \frac{R_{tot}}{0.5t_a + \frac{1}{K}\left(t_d - t_p\right)\left(1-e^{-K}\right)}$$

The total depth for any time t can then be obtained by integration as shown in equations [7-14] and [7-15].

$$[7.14] \qquad Vol = \frac{h}{2} . \frac{t^2}{t_p} \qquad \text{for} \quad t < t_p$$

$$[7.15] \qquad Vol = \frac{h}{2} t_p + \frac{1}{K} h(t_d - t_p)\left(1 - e^{-K\frac{(t-t_p)}{(t_d-t_p)}}\right) \qquad \text{for} \quad t > t_p$$

By computing the volumes Vk and $Vk+1$ at the beginning and end of a time step, the intensity during the interval is then defined by equation [7-16].

$$[7.16] \qquad i_k = \frac{V_{k+1} - V_k}{\Delta t}$$

Suggested values for K and tp are shown in two tables referenced below. These are based on data published by Watt *et al*. Values in minutes of the time to peak tp are for 60 minute storms only and should not be used for storms of different duration. The time to peak as a proportion of duration is provided for guidance. See the topics listed below.

7.1.7 Suggested K values for Canadian Provinces

Province	**K value**
B.C.(coastal region)	5
Yukon, New Brunswick, Nova Scotia, Newfoundland	6
B.C.(interior), Alberta, Saskatchewan, Manitoba, Ontario, Quebec	7

(from Watt *et al* - see References)

7.1.8 Suggested tp values for locations in Canada

Location	tp (minutes)	tp /Duration
Yukon	20	0.33
B.C.(coast)	28	0.47
B.C.(interior), Prince George	13	0.22
Alberta	17-18	0.29
Saskatchewan	23-24	0.39
Manitoba (Brandon, Churchill)	31	0.52
Manitoba (Winnipeg)	25	0.42
Ontario (Timmins, Thunder Bay)	24-25	0.41
Ontario (Ottawa, Kingston, Windsor)	26-27	0.44
Ontario (Toronto, Sudbury)	21	0.35
Quebec (Montreal)	27	0.45
Quebec (Val D'Or, Quebec City)	23	0.38
New Brunswick (Fredericton)	17	0.28
Nova Scotia, Newfoundland	26-28	0.45

(from Watt *et al* - See References)

Note: These times to peak were obtained from 60 minute duration storms.

7.2 Calculating Effective Rainfall

Effective rainfall - sometimes called excess rainfall - is the component of the storm hyetograph which is neither retained on the land surface nor which infiltrates into the soil. The effective rainfall produces overland flow that results in the direct runoff hydrograph from a sub-area of a catchment. The difference between the storm and the effective rainfall hyetographs is termed the abstractions or rainfall losses. Abstractions are made up of one or more of the following three main components:-

1. interception by vegetation or tree canopy
2. infiltration into the soil
3. storage in surface depressions and hollows

In the absence of field observations it is usually necessary to employ some form of mathematical model to represent the abstractions. This must be done for the pervious and the impervious fractions of the catchment.

MIDUSS currently lets you choose from three methods to define the infiltration model.

(1) The Soil Conservation Service (SCS) infiltration method or the SCS Method

(2) The 'moving curve' Horton Equation for infiltration.

(3) The Green and Ampt Method

These options are available for both the pervious and impervious fractions of the catchment. However the choice of infiltration model must be made on the Pervious tab of the Catchment form. When the Impervious tab is displayed there is a note to remind you which option is in use. Refer to the Catchment command; Data for the Pervious Area for details and a view of the relevant forms.

The choice of infiltration method is made after the selection of the overland routing option. It is important to note that if you intend to use the SWMM/Runoff option for the generation of the overland flow hydrograph, the SCS infiltration option is not available and you must choose between the Horton equation and the Green & Ampt method.

The descriptions of the infiltration models given in the sections that follow apply equally to both impervious and pervious fractions of the catchment. Details are provided for:

The SCS Infiltration method

The Horton Infiltration equation

and The Green & Ampt algorithm.

7.2.1 The SCS Method

In 1972 the U.S. Soil Conservation Service suggested an empirical model for rainfall abstractions which is based on the potential for the soil to absorb a certain amount of moisture. On the basis of field observations, this potential storage S (millimetres or inches) was related to a 'curve number'

CN which is a characteristic of the soil type, land use and the initial degree of saturation known as the antecedent moisture condition.

The value of S is defined by the empirical expression [7-17] or [7-18] depending on the units being used.

[7.17] $$S = \frac{1000}{CN} - 10$$ (inches)

[7.18] $$S = \frac{25400}{CN} - 254$$ (millimetres)

Typical values for the SCS Curve Number *CN* as a function of soil type, land use and degree of saturation can be found in most texts on hydrology (*e.g.* See references such as Viessman, 1977 or Kibler, 1982) or from the section on Pervious Data requirements. in Chapter 3 *Hydrology Used in MIDUSS.*

In some texts you may see values of *CN* quoted as a function of the percentage of impervious area. These are usually calculated as a weighted average assuming *CNimpervious* = 98 and *CNpervious* equal to the value for 'Pasture in good condition' for the various soil types A, B, C or D. See Chapter 3 *Hydrology used in MIDUSS*, eq. [3.10].

Values of *CN* estimated in this way are intended to be applied to the **total** catchment assuming other parameters to be the same for both pervious and impervious areas. Many programs (including MIDUSS) compute the runoff from the pervious and impervious fractions separately and then add the two hydrographs. In such cases, it is most important that you **do not use** a composite value of *CN* since this would 'double count' the impervious fraction and greatly exaggerate the runoff prediction.

The effective rainfall is computed by the equation:

[7.19] $$Q(t) = \frac{\left(P(t) - I_a\right)^2}{\left(P(t) + S - I_a\right)}$$

where $Q(t)$ = accumulated depth of effective rainfall to time t

$P(t)$ = accumulated depth of rainfall to time t

Ia = initial abstraction

S = potential storage in the soil

All of the terms in equation [7-19] are in units of millimetres or inches. Note that the effective rainfall depth or runoff will be zero until the accumulated precipitation depth P(t) exceeds the initial abstraction *Ia*.

The original SCS method assumed the value of the initial abstraction *Ia* to be equal to 20% of the storage potential S, but many engineers now regard this as unacceptably high for most stormwater management situations. MIDUSS uses an initial default value of 10% but allows you to specify the ratio of $fa = Ia / S$ when you are entering the data for rainfall losses.

Alternatively, MIDUSS lets you define the initial abstraction Ia explicitly as a depth. For suggested values, see the section on Pervious Data requirements in Chapter 3 *Hydrology Used in MIDUSS.*.

When you enter a value for the SCS Curve Number, MIDUSS calculates the equivalent volumetric runoff coefficient (C) and displays this for information. You can also enter a value for the runoff coefficient and MIDUSS will compute and display the corresponding value of CN. The SCS CN value is a function of runoff coefficient C, the total rainfall depth and the initial abstraction ratio $fa = Ia./S$. The relationship used is as follows.

$$[7.20] \qquad CN = \frac{1000}{10 + P_{tot}\left[\frac{1}{f_a} + \frac{1-f_a}{2f_a^{\,2}}.C\left[1-\left(1+\frac{4f_a}{(1-f_a)^2}\frac{1}{C}\right)^{1/2}\right]\right]}$$

Sometime this is a useful way to 'guesstimate' a value for CN in the absence of other information.

It is worth digressing a little at this point to explain a feature of MIDUSS which you may notice when you are reviewing an output file. If you specify a runoff coefficient C for a particular sub-catchment, both the values of C and CN are copied to the output file. However, if you run the program in Automatic mode MIDUSS uses the CN value as the basis for estimating rainfall losses. The reason for this is as follows. Typically, in designing a minor drainage system the engineer will use a relatively modest storm (say 5 year return interval) for which a reasonable estimate of C might be made based on records or previous experience with the rational method.

When the design is completed it is usual to subject the system to a more severe storm with a much larger depth of precipitation $Ptot$. For the same ground conditions, the severe storm will produce a much higher runoff coefficient than the 5- year storm. Now since the CN value is a measure of ground conditions it is preferable to use the CN value rather than the runoff coefficient C, which, if used with the severe storm, would greatly under-estimate the runoff. Of course, if the output file is used as input for a subsequent run in which the 5- year storm is used again, the result will be identical to that which would have been obtained using the runoff coefficient. After specifying values for Manning's 'n' and the SCS curve number CN (or runoff coefficient C) MIDUSS displays the current value of the ratio $fa = Ia /S$ as well as the initial abstraction depth Ia in inches or millimetres. You have the option to accept the current values or alter the ratio Ia /S or the initial abstraction Ia by entering values in the appropriate text boxes.

If either the ratio $fa = Ia /S$ or the initial abstraction depth Ia is altered, the displayed values of both Ia /S and Ia are updated. These values become the default for future uses of the Catchment command but these are not retained for future design sessions with MIDUSS.. However, if the output file is later used as an input data file in Automatic mode the correct values will be used.

In both the Pervious and Impervious forms, pressing the [Display] button causes a tabular display of the effective rainfall to be displayed together with a graph showing the storm rainfall and one or both of the two effective rainfall hyetographs.

7.2.2 The Horton Equation

One of the first attempts to describe the process of infiltration was made by Horton in 1933. He observed that the infiltration capacity reduced in an exponential fashion from an initial, maximum rate $f0$ to a final constant rate fc.

The Horton equation for infiltration capacity $fcapac$ is given by equation [7-21] which shows the variation of the maximum infiltration capacity with time t.

$$[7.21] \qquad f_{capac} = f_c + (f_0 - f_c)e^{-t/K}$$

where $fcapac$ = maximum infiltration capacity of the soil

$f0$ = initial infiltration capacity

fc = final (constant) infiltration capacity

t = elapsed time from start of rainfall

K = decay time constant

At any point in time during the storm, the actual infiltration rate must be equal to the smaller of the rainfall intensity $i(t)$ and the infiltration capacity $fcapac$. Thus the Horton model for abstractions is given by equations [7-22] and [7-23].

$$[7.22] \qquad f = f_{capac} \qquad \text{for } i > fcapac$$

$$[7.23] \qquad f = i \qquad \text{for } i <= fcapac$$

where f = actual infiltration rate (mm/hr or inches/hr)

i = rainfall intensity (mm/hr or inches/hr).

Figure 7-8 below shows a typical problem in which the average rainfall intensity in each time step is shown as a stepped function. It is clear that if the total volume of rain in time step 1 (say) is less than the total infiltration volume in that time step it is more reasonable to assume that the reduction in f is dependent on the infiltrated volume rather than on the elapsed time. It is therefore usual to use a 'moving curve' technique in which the ft curve is shifted by an elapsed time which would produce an infiltrated volume equal to the volume of rainfall.

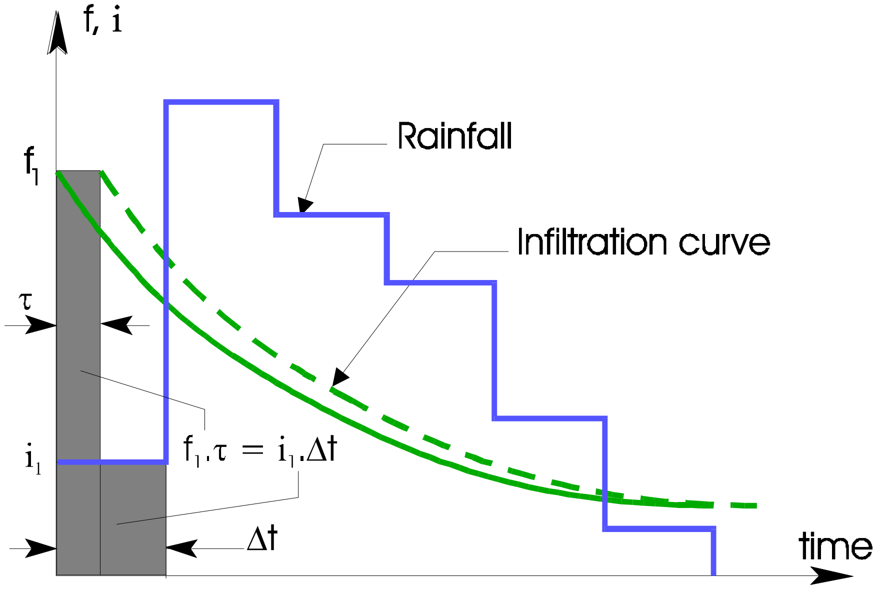


Figure 7-6: Representation of the moving curve Horton equation

Figure 7-6 shows a dashed infiltration curve shifted by a time τ which is defined as follows.

Let

$$\Delta F = \int_{t}^{t+\Delta t} f_{capac}\,dt$$

If $i.dt \geq \Delta F$

then $\tau = \Delta t$ and

[7.24] $$f_0(new) = f_c + \left(f_0 - f_c\right)e^{-\frac{\Delta t}{K}}$$

If $i.dt < \Delta F$

then τ is defined implicitly by the equation

$$\int_{t}^{t+\tau} f_{capac}\,dt = i.\Delta t$$

and

[7.25] $$f_0(new) = f_c + \left(f_0 - f_c\right)e^{-\tau/K}$$

Solving for τ involves the implicit solution of equation [7-26]

[7.26] $$f_c\tau + K\left(f_0 - f_c\right)\left(1 - e^{-\tau/K}\right) = i.\Delta t$$

Application of equations [7-24] - [7-26] to every time step of the storm results in a hyetograph of effective rainfall intensity on either the impervious or pervious fraction. If the surface has zero surface depression storage, this is the net rainfall that will generate the overland flow. However, if the depression storage is finite, this is assumed to be a first demand on the effective rainfall and the depth must be filled before runoff can occur.

You are prompted to enter a total of five parameters comprising Manning's 'n', the initial and final infiltration rates $f0$ and fc (mm/h or inch/h), the decay time constant K (in hours, not 1/hrs) and the depression surface storage depth (millimetres or inches). For the impervious fraction you can enter either very small or zero values for all the parameters except the Manning roughness coefficient 'n'.

7.2.3 The Green and Ampt Method

The basic assumption behind the Green and Ampt equation is that water infiltrates into (relatively) dry soil as a sharp wetting front. Figure 7-7 below illustrates the variation in moisture content θ with depth z below the surface, at a point in time when the front has progressed a distance L.

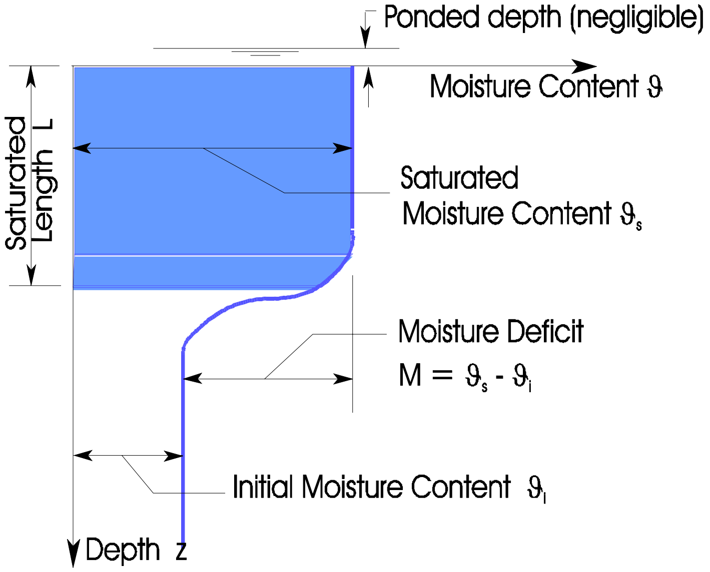


Figure 7-7 - Wetting front of the Green & Ampt model

The passage of this front causes the moisture content to increase from an initial value θi to a saturated value θs. This difference is defined as the moisture or water deficit M, *ie*

[7.27] $$M = \theta_s - \theta_i$$

Typically for dry soils M has a value in the range $0.2 < M < 0.5$ depending on the soil voids ratio, with lower values for pre-wetted soil.

If the hydraulic conductivity of the soil is K (mm/hour or inches/hr) then by Darcy's law,

[7.28] $$f = \frac{dL}{dt} = -K.\frac{\partial h}{\partial z}$$

where $(\partial h/\partial z)$ represents the hydraulic gradient.

The head causing infiltration is given by equation [7-29].

[7.29] $$h = h_0 + L + S$$

where *h0* = depth of surface ponding (usually neglected)

L = depth of water already infiltrated

S = suction head at the wetting front.

The suction head *S* (millimetres or inches) is due to capillary attraction in the soil voids and is large for fine grained soils such as clays and small for sandy soils.

The total infiltrated volume between the surface of the soil and the wetting front is defined by equation [7-30].

[7.30] $$F = L \times M$$

The infiltration rate $f = dF/dt$ is then given by [7-31].

[7.31] $$f = K\left(1 + \frac{MS}{F}\right)$$

In order to calculate the effective rainfall, this equation must be solved for each time step in the storm hyetograph. As illustrated in the Figure 7-8, three cases must be considered in which the infiltration rates at times t and $(t + \Delta t)$ are denoted by f_1 and f_2 respectively, and the rainfall intensity i is assumed to be constant during the time step. Each case is considered separately.

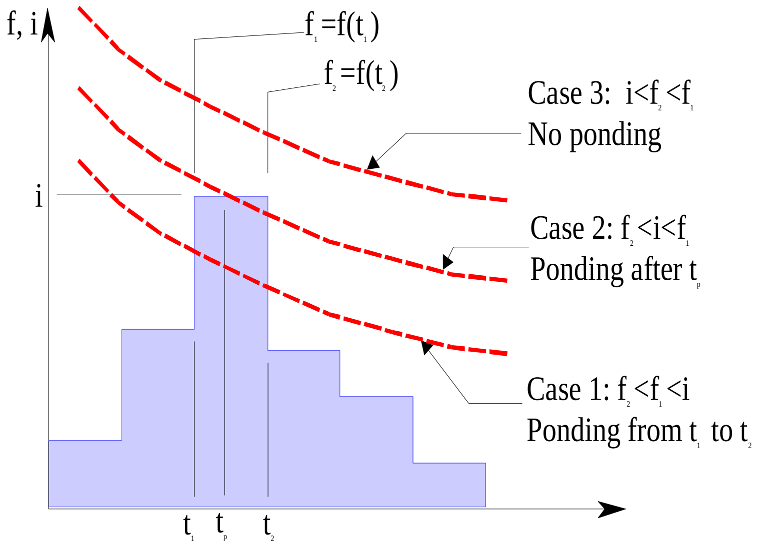


Figure 7-8 - Three cases of the Green & Ampt model.

Case (1) $$f_2 < f_1 < i$$

i.e. the rainfall intensity exceeds the infiltration capacity of the soil throughout the whole time step so that ponding must occur for the entire time Δt.

Case (2) $f_2 < i < f_1$

i.e. at the beginning of the time step Δt the infiltration capacity f_1 exceeds the rainfall intensity but this changes before the time step is completed. Ponding will start during the time Δt.

Case (3) $i < f_2 < f_1$

i.e. the rainfall infiltrates for the entire time step and no ponding occurs.

The solution algorithm used can be summarized as follows.

(i) If $ij>fj$ then case (1) holds and

$$[7.32] \qquad F_{j+1} = F_j + K\Delta t + MS \ln\left[\frac{(F_{j+1} + MS)}{(F_j + MS)}\right]$$

The effective rainfall is then given by [7.33].

$$[7.33] \qquad i_{eff} = i - \frac{(F_{j+1} - F_j)}{\Delta t}$$

If $ij \leq fj$ then either case (2) or case (3) applies. If we assume that case (3) applies - i.e. all the rainfall infiltrates during time Δt - then we can estimate:

$$[7.34] \qquad F_{J+1} = F_j + i\Delta t$$

and

$$[7.35] \qquad f_{j+1} = K\left(1 + \frac{MS}{F_{j+1}}\right)$$

(iii) Test if $i_j \leq f_{j+1}$ also. If so, then case (3) is true and:

$$[7.36] \qquad i_{eff} = 0$$

If $i_j > f_{j+1}$ as computed in step (ii) then case (2) holds. The volume required to cause surface ponding to occur is calculated as:

$$[7.37] \qquad F_p = \frac{KMS}{(i_j - K)}$$

The time to the start of ponding δt can then be found from equation [7-38].

$$[7.38] \qquad \delta t = \frac{(F_p - F_j)}{i}$$

Then:

[7.39] $$F_{j+1} = F_j + K\left(\Delta t - \delta t\right) + MS\ln\left[\frac{\left(F_{j+1} + MS\right)}{\left(F_j + MS\right)}\right]$$

and the effective rainfall can be estimated as:-

[7.40] $$i_{eff} = i - \frac{\left(F_{j+1} - F_j\right)}{\Delta t}$$

The application of this algorithm to each time step in the storm hyetograph produces an effective rainfall hyetograph for either the impervious or pervious surface. If the surface depression storage is finite this is subtracted from the initial elements of the hyetograph. The remaining effective rainfall produces the direct runoff hydrograph.

In the Green and Ampt method you are prompted to supply values for a total of five parameters. These are:

1. Manning's 'n' roughness coefficient
2. Water (or Moisture) deficit M (say 0.0 to 0.6)
3. Suction head S (mm or inches)
4. Soil conductivity K (mm/hour or inches/hour)
5. Surface depression storage depth (mm or inches)

7.2.4 Parameters for the Green & Ampt equation

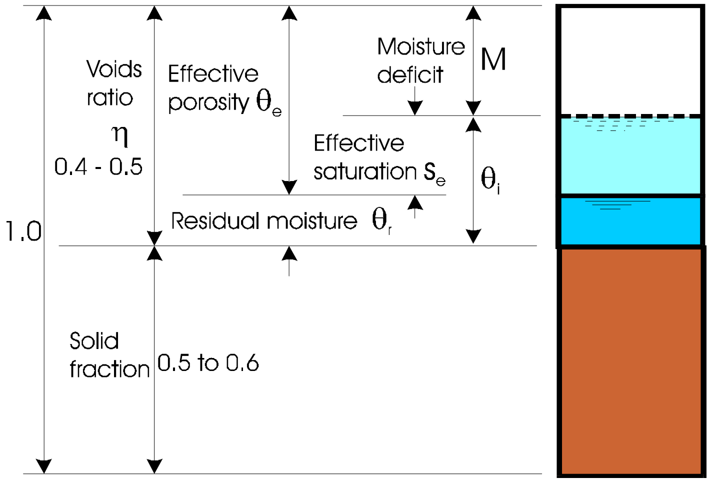


Figure 7-9 – Schematic representation of the Green & Ampt parameters

The schematic of Figure 7-9 shows the fractions of solid material, moisture and air or vapour in the soil. The voids ratio of the soil η is typically between 0.4 to 0.5. Within the voids of a dried sample there is a certain volume of residual moisture θr. The remaining 'fillable' voids comprise the effective porosity $\theta e = \eta - \theta r$ and typically varies from 0.31 to 0.48. Now if the initial moisture is denoted by θi the soil moisture deficit $M = \theta e - \theta i$.

The effective saturation is denoted by $Se = (\theta - \theta r)/ \theta e$ and can be used to estimate the suction head at the wetting front as described by Brooks and Corey, 1964. - (see references)

Some typical values suggested by Rawls, Brakensiek and Miller (1983) (see references) are shown below.

Soil type	**Porosity**	**Effective porosity**	**Suction head**	**Hydraulic conductivity**
			mm	**mm/h**
Sand	0.437	0.417	49.5	117.8
loamy sand	0.437	0.401	61.3	29.9
sandy loam	0.453	0.412	110.1	10.9
Loam	0.463	0.434	88.9	3.4
silt loam	0.501	0.486	166.8	6.5
sandy clay loam	0.398	0.330	218.5	1.5
clay loam	0.464	0.309	208.8	1.0
silty clay loam	0.471	0.432	273.0	1.0
sandy clay	0.430	0.321	239.0	0.6
silty clay loam	0.479	0.423	292.2	0.5
Clay	0.475	0.385	316.3	0.3

7.3 Calculating the Runoff

7.3.1 Catchment Shape

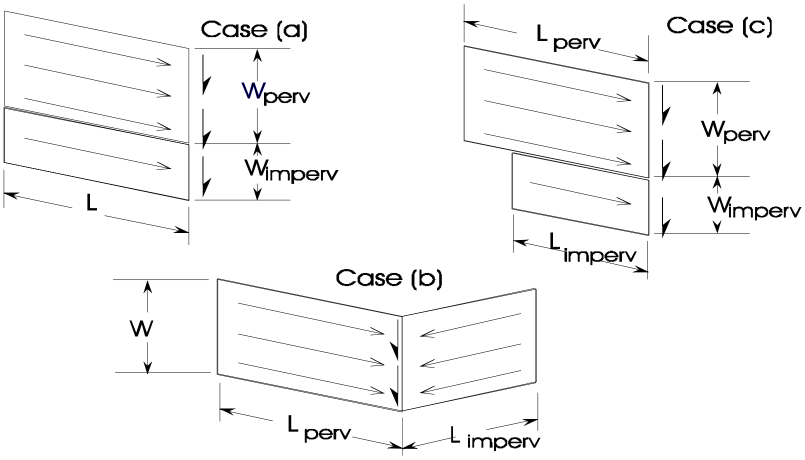


Figure 7-10 - Alternative definitions of catchment shape.

The catchment is assumed to be represented by two idealized, rectangular inclined planes - one for the pervious surface and the second for the impervious fraction. The two planes are commonly assumed to be inclined at the same gradient, but MIDUSS lets you define this and all other characteristics to be different.

For each catchment you must first specify the total catchment area and the percentage of that area which is impervious. MIDUSS then provides three options to define the shape of the two surfaces. Each of these is illustrated in a rather idealized way in Figure 7-10.

The default assumption is that the length of overland flow on the impervious surface is the same as that specified for the pervious fraction. This case is shown in Figure 7-10(a).

Alternatively, you may choose an option which assumes that the width of both rectangles is the same. This is equivalent to assuming that the overland flow lengths are in the same proportion as the areas of the two fractions, and is illustrated in Figure 7-10(b).

The third option allows you to define a specific length of overland flow for each of the two rectangles, so that neither length nor width need be the same. This case is shown in Figure 7-10(c).

From the sketches of Figure 7-10 it should be clear that the overland flow length is the distance from the boundary of the idealized rectangle to the drainage conduit (pipe or channel). It is along the overland flow length that the surface gradient should be estimated. The idealized catchments of Figure 7-10 are shown as non-symmetrical (i.e. with all the pervious or impervious area on one side of the drainage conduit) only to illustrate the concept.

In practice, it is usual for both pervious and impervious surfaces to be distributed more or less symmetrically about the drainage conduit.

Avoid the mistake of estimating overland flow length and slope between the outflow point and the point on the catchment boundary which is furthest from the outlet. This overestimates the time of concentration and underestimates the peak outflow.

If the catchment area is symmetrically distributed around the drainage network, an approximate value for the overland flow length can be found by dividing the area by twice the length of the drainage channel. If the catchment is unsymmetrical so that the drainage channel is along one edge of the catchment, the overland flow length can be approximated as (Area/Channel length). The two cases of symmetrical and one-sided catchments are illustrated in Figures 7-11(a) and (b) respectively. If neither of these cases applies then you must either make a subjective judgment or simulate the area as two separate sub-catchments.

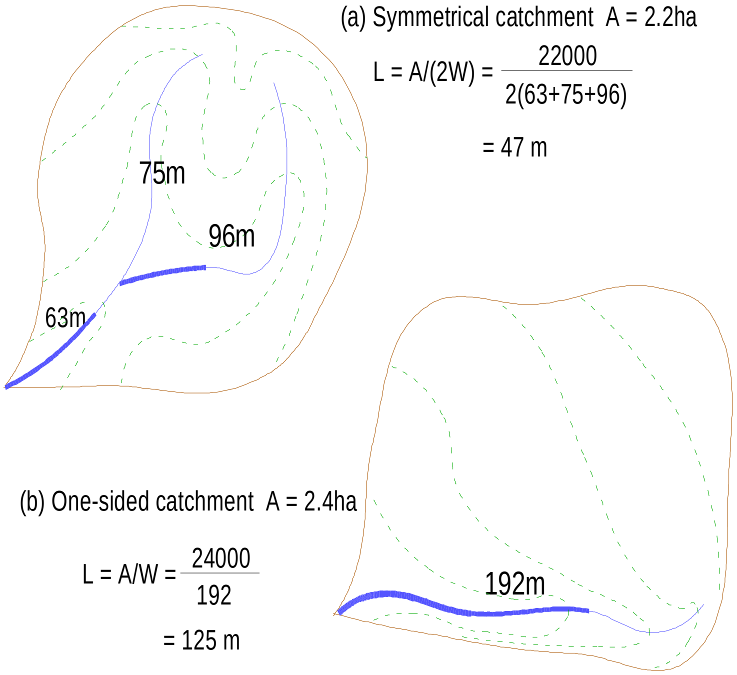


Figure 7-11 - Estimating overland flow length in symmetrical and one-sided catchments.

Another point to note is that in MIDUSS the impervious fraction is assumed to be directly connected to the drainage network. This means that flow from the impervious areas does not pass over a pervious area before reaching the drainage channel. In some urban drainage models the impervious area is further subdivided into directly and indirectly connected fractions but these methods assume that runoff from the indirectly connected impervious area is uniformly distributed over the pervious fraction. In practice, such runoff is usually concentrated over a relatively small pervious area thus reducing the potential for infiltration. The assumption in MIDUSS therefore leads to a conservative estimate of the total runoff from the catchment.

The Manning 'n' value is used to estimate the time of concentration (see equation [7.41]) for any specific intensity of effective rainfall. Typical values of ‘n’ for overland flow on pervious surfaces should be in the range 0.2 - 0.35 and do not represent realistic values of 'n' that might be used in channel flow calculations.

In addition to the above description, parameters must be defined which describe the infiltration process and rainfall abstractions on the pervious area. These will depend on the infiltration model selected and are as described in the section Calculating Effective Rainfall .

7.3.2 The Idealized Catchment

The catchment area is divided into various components as indicated in Figure 7.16. In MIDUSS both the pervious and impervious areas are assumed to be directly connected and are further assumed to be described by lumped parameters - i.e. each fraction is assumed to be homogeneous.

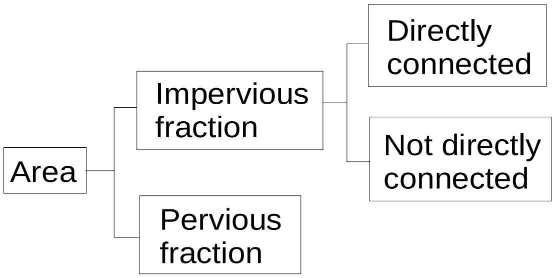


Figure 7-12 - Various components of an idealized catchment.

7.3.3 Conceptual Components of Rainfall

For each fraction of the catchment (pervious and impervious) the rainfall loss is the difference between the rainfall depth and the depth of runoff. This is made up of various components as illustrated in Figure 7-13. Not all methods of modelling rainfall losses use all of these components.

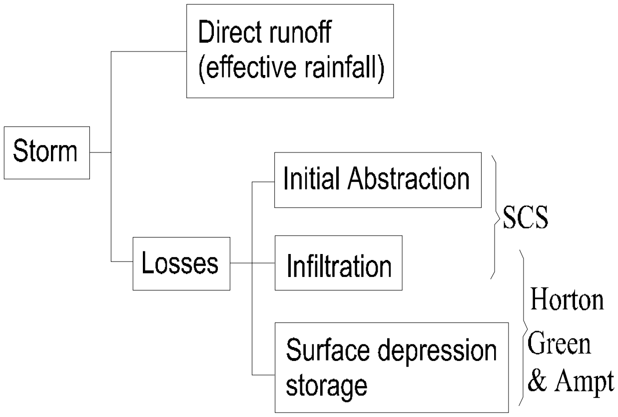


Figure 7-13 – Rainfall abstraction models use different components

The initial abstraction Ia may be defined explicitly as an average depth over the area (in mm or inches) or implicitly as a fraction of the potential storage depth in the soil (e.g. $Ia = 0.1\ S$). The notion of initial abstraction is used in the SCS infiltration method, but not by either the Horton equation or the Green & Ampt methods. The initial abstraction depth is treated as a first demand on the storm rainfall; surface depression storage is a first demand on the surface water excess leading to runoff.

The infiltration capacity is assumed to decrease continuously throughout the storm as the storage potential in the soil is progressively reduced by the volume of infiltration. The reduction in infiltration capacity is a function of the infiltrated volume and not of the elapsed time from the start of rainfall. In release 1 of MIDUSS no provision is made for 'recovery' of infiltration potential during periods of zero or very low rainfall. For single event modelling this is not likely to be significant. MIDUSS models the infiltration process by

- the SCS method,
- the 'moving curve' Horton equation or
- the Green & Ampt model.

Surface depression storage is represented by an average depth distributed uniformly over the surface area. The usual assumption made is that when rainfall intensity exceeds the infiltration capacity the depth of excess water on the surface must attain a value greater than the surface depression storage depth before runoff can occur. The concept of surface depression storage depth is not used in the SCS method but plays a significant role in the implementation of the Horton or Green & Ampt methods.

7.3.4 Processing the Storm Rainfall

The significance of the components of rainfall loss is illustrated in Figures 7-14 and 7-15. For this comparison the storm used is a 2nd quartile Huff distribution with a total depth of 50 mm over a period of 120 minutes. The rainfall abstractions have been modelled using the SCS Curve Number method with *CN* = 88.

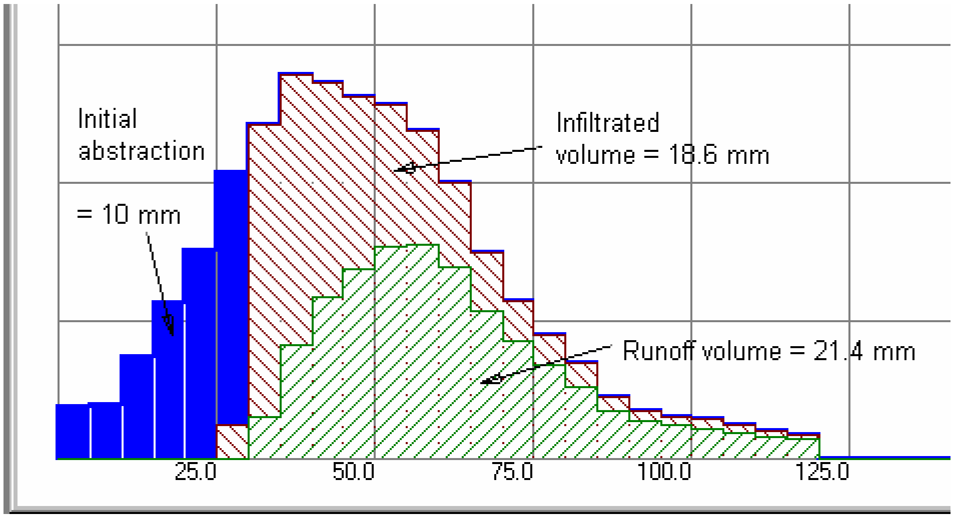


Figure 7-14 – Rainfall loss components with Ia = 10 mm, Yd = 0.0

Fig 7-14 above shows the normal application of the SCS method in which an initial abstraction Ia = 10 mm has been applied. It is clear that this is a first demand on the storm hyetograph. The remaining 40 mm of rainfall is split into an infiltrated volume of 18.6 mm leaving 21.4 mm of direct runoff.

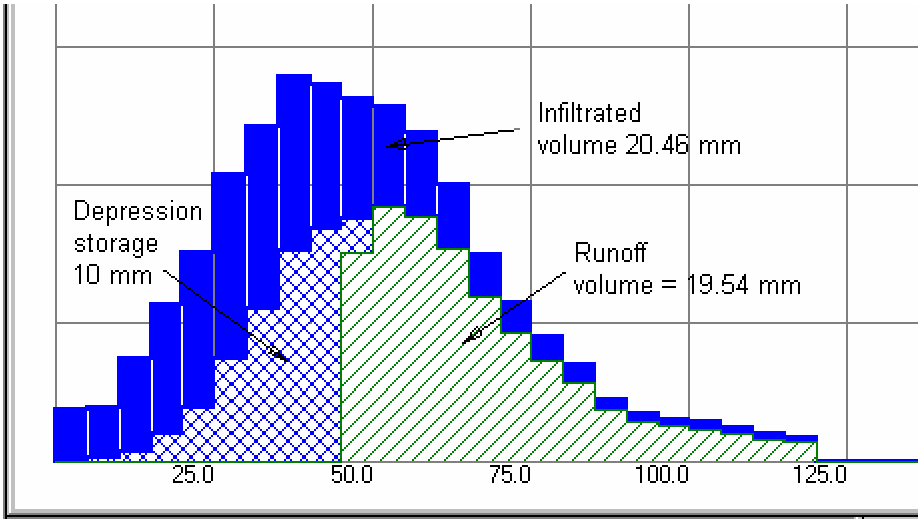


Figure 7-15 – Rainfall loss components with Ia = 0.0 and Yd = 10.0 mm.

Fig 7-15 shows an unusual application of the SCS method developed by means of some of the options in MIDUSS. In this case the initial abstraction is zero so that an infiltration volume of 20.46 mm is the first demand on the storm hyetograph. The remaining 29.54 mm of rainfall is divided between 19.54 mm of direct runoff and 10 mm which is detained as surface depression storage.

Note the difference in volume, peak intensity and shape of the direct runoff component. This would certainly be reflected in the resulting overland flow hydrograph. In this example the differences have been exaggerated by using a relatively large depth for Ia or yd. You will find it instructive to recreate this experiment using the Horton method to model the infiltration process or with smaller values of Ia and yd.

7.3.5 Rainfall Runoff Models

Rainfall runoff models may be grouped in two general classifications that are illustrated in Figures 7-16 and 7-17. The first approach uses the concept of effective rainfall in which a loss model is assumed which divides the rainfall intensity into losses and an effective rainfall hyetograph. The effective rainfall is then used as input to a catchment model to produce the runoff hydrograph. It follows from this approach that the infiltration process ceases at the end of the storm duration.

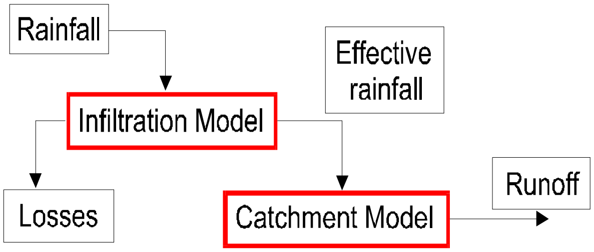


Figure 7-16 - A rainfall-runoff models using effective rainfall.

An alternative approach that might be termed a surface water budget model, incorporates the loss mechanism into the catchment model. In this way, the incident rainfall hyetograph is used as input and the estimation of infiltration and other losses is made as an integral part of the calculation of runoff. This approach implies that infiltration will continue to occur as long as the average depth of excess water on the surface is finite. Clearly, this may continue after the cessation of rainfall.

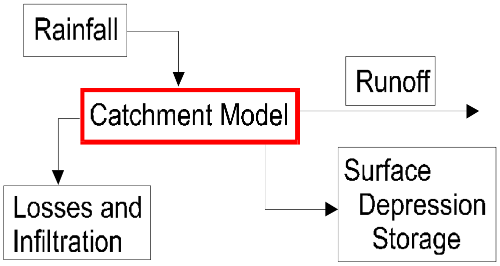


Figure 7-17 – A rainfall-runoff model using a surface water budget

MIDUSS allows you the option to use particular implementations of both these techniques. The effective rainfall approach is employed in a convolution algorithm that uses response functions of different shape. For the case of triangular or rectangular response functions the time base is computed by a kinematic wave equation which involves the intensity of the effective rainfall.

The convolution process is therefore nonlinear in that the response function changes throughout the storm but the principle of superposition is retained. These two approaches are embodied in the ‘Rectangular’ and ‘Triangular SCS’ options of the **Hydrology/Catchment** command.

The third convolution option uses a response function which is obtained by routing a rectangular input of duration Δt and height $A/\Delta t$ through a linear reservoir with a lag or storage coefficient $KL=tc/2$. For this case, the time of concentration tc is computed using the kinematic wave equation [7-41] but for the maximum value of effective rainfall intensity.

An example of a surface water budget model is also made available in the form of the SWMM/RUNOFF algorithm and can be implemented by using the ‘SWMM method’ option. It is important to note that if the ‘SWMM method’ option is to be employed it is necessary to use the Horton or Green and Ampt infiltration models to represent the rainfall losses. The four options are described in the sections that follow.

7.3.6 A Rectangular Response Function

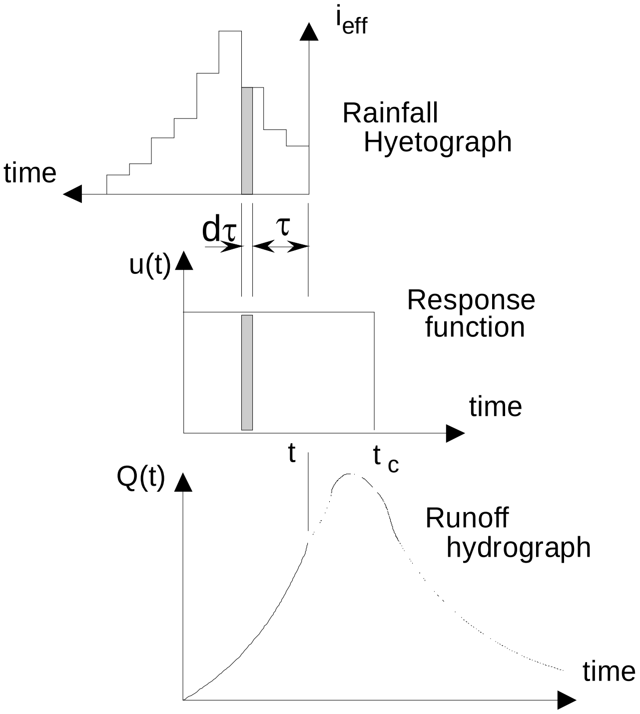


Figure 7-18 - Convolution using a rectangular response function.

Figure 7-18 illustrates a convolution process in which the response function is assumed to be rectangular with a dynamically varying time base equal to the time of concentration as defined by equation [7.41].

$$[7.41] \qquad t_c = k\left(\frac{L.n}{\sqrt{S}}\right)^{0.6} \frac{1}{i_{eff}^{0.4}}$$

where L = flow length (m or feet)

n = Manning's roughness coefficient

S = slope of overland flow (m/m or ft/ft)

ieff = effective rainfall (mm/h or inch/h)

k = 6.989 for metric units

= 0.939 for Imperial or US customary units

The ordinate of the response function is given by *umax = A/tc* so that the evaluation of a discretized form of the convolution integral is relatively straightforward. If the effective rainfall is also a simple rectangular function the method reduces to the rational method. There is some evidence (Smith & Lee, 1984 see references) that this method is appropriate when the overland flow is dominated by runoff from relatively smooth, impervious surfaces.

In using the Rectangular Response option it is possible to define an artificially short flowlength (e.g. 1.0 m) thus making the time of concentration of negligible duration. This is equivalent to employing a Dirac δ- function as the response function and may be of interest in simulating other models.

7.3.7 The SCS Triangular Response

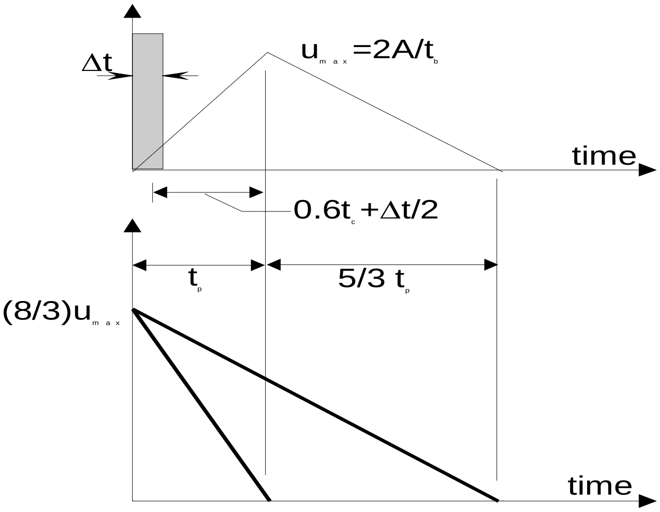


Figure 7-19 - Representation of the triangular response function.

A very common and popular technique proposed by the Soil Conservation Service uses a triangular response function as shown in the upper part of Figure 7-19 above. The time to peak *tp* of the triangular IUH and the time base *tb* are given by equations [7-42] and [7-43] respectively.

[7.42] $$t_p = 0.6t_c + \frac{\Delta t}{2}$$

[7.43] $$t_b = \left(\frac{8}{3}\right)t_p$$

The ‘Triangular SCS’ option in MIDUSS represents a modification of this method in that the value of *tc* is obtained by [7-41] and is assumed to vary in a nonlinear fashion in much the same way as for the rectangular response function. For each time step the effective rainfall intensity is known and the triangular response function for the corresponding time of concentration *tc* is discretized and multiplied by the effective rainfall. The resulting contributions to the overland flow hydrograph are lagged and accumulated. The computation is made more efficient by representing the triangular response as the difference between two right- angled triangles as indicated in the lower part of Figure 7-19.

It will be found that with both the “Rectangular’ and ‘Triangular SCS’ methods the time of concentration of the impervious fraction is much shorter than that for the pervious area due to smaller values of Manning's 'n' and higher effective rainfall intensities. In cases where the contribution from each fraction is of the same order of magnitude the total runoff hydrograph exhibits a double peak because of this feature.

7.3.8 The Linear Reservoir Response

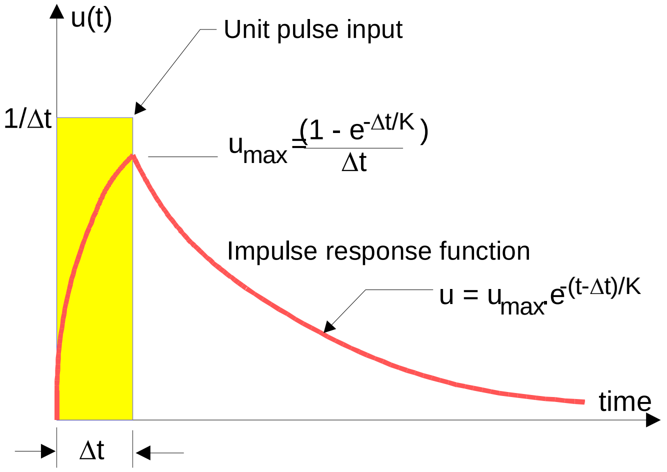


Figure 7-20 - The Single Linear Reservoir IUH

A more complex response function was suggested by Pederson (see references) and is currently in use in the URBHYD routine of the OTTHYMO model. The shape of the Instantaneous Unit Hydrograph (IUH) is obtained as the response of a single linear reservoir to a rectangular pulse of rainfall of unit volume and duration Δt. The storage coefficient K of the linear reservoir is taken to be $0.5\ tc$ where tc is computed by equation [7-41] in which the maximum rainfall intensity is used since this intensity tends to dominate the subsequent convolution process. The resulting IUH is illustrated in Figure 7-20 and comprises a steeply rising limb over the time step Δt followed by an exponential decay. Most applications of this method have used a procedure in which the IUH is discretized at intervals of Δt and then convoluted with the effective rainfall.

Because tc is assumed to be constant in this method, both the response of the linear reservoir and the convolution are linear processes and it is therefore immaterial in what order they are carried out. The essential equivalence of the alternate methods is illustrated in Figure 7-21.

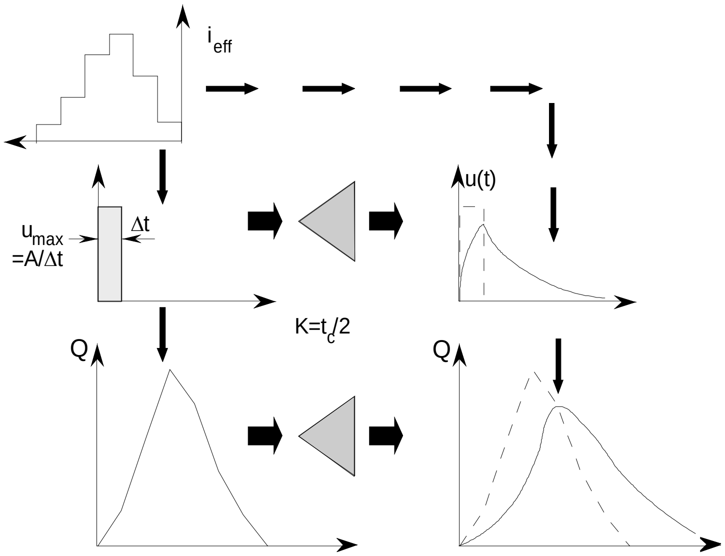


Figure 7-21 - Alternative implementations of a linear reservoir response.

MIDUSS uses the alternate approach of convoluting the effective rainfall with a simple rectangular response of duration Δt and height $umax = A/\Delta t$. The resulting 'instantaneous' runoff hydrograph is then routed through the linear reservoir. This approach reduces the computational time by at least an order of magnitude and improves the accuracy.

The routing process is carried out using a time step of $\Delta t/2$ in order to improve the accuracy in the vicinity of the peak runoff but the results are presented only at intervals of Δt.

7.3.9 The SWMM - RUNOFF Algorithm

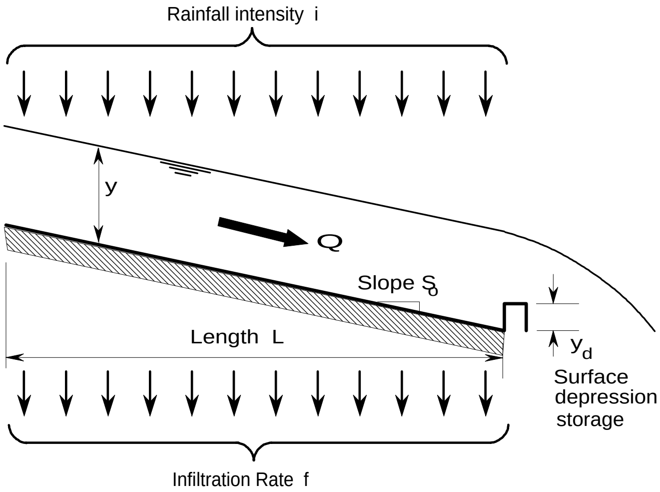


Figure 7-22 - Representation of the SWMM/RUNOFF algorithm.

The U.S. EPA SWMM model is made up of a number of large program modules. One of these - the RUNOFF block - is used to generate the runoff hydrograph from a sub-catchment. In MIDUSS, the ‘SWMM Method’ option uses a similar algorithm with the limitation that only the Horton or Green and Ampt infiltration equations are supported.

The method employs the surface water budget approach and may be visualized as shown in Figure 7-22. The incident rainfall intensity is the input to the control volume on the surface of the plane; the output is a combination of the runoff Q and the infiltration f. Considering a unit breadth of the catchment the continuity and dynamic equations which have to be solved are as shown in equations [7-44] and [7-45].

$$iL = \left(fL + \frac{Q}{B} \right) + L\frac{\Delta y}{\Delta t} \quad [7.44]$$

$$Q = B\frac{C_M}{n} S^{1/2} \left(y - y_d \right)^{5/3} \quad [7.45]$$

where L = overland flow length

B = catchment breadth

CM = 1.0 for metric units

1.49 for Imperial or US customary units

n = Manning roughness coefficient

y_d = surface depression storage depth

Rewriting [7-44] with $q = Q/B$ and then substituting for Q by means of [7-45] yields the single equation:

$$[7.46] \qquad \Delta y = \Delta t\left(i - f - \frac{q}{L}\right)$$

or

$$[7.46a] \qquad \Delta y = i \quad \Delta t - f \quad \Delta t - \Delta t \quad \frac{C_M}{n} S^{1\!/\!2} \frac{\left(y - y_d\right)^{5\!/\!3}}{L}$$

If the depth on the plane at the start and finish of the time step Δt is represented by y_1 and y_2 respectively an equation for y_2 can be developed using the following approximations.

$$[7.47] \qquad \Delta y = y_2 - y_1$$

$$[7.48] \qquad \left(y - y_d\right)^{5\!/\!3} = \frac{\left(y_1 - y_d\right)^{5\!/\!3} + \left(y_2 - y_d\right)^{5\!/\!3}}{2}$$

$$[7.49] \qquad f \quad \Delta t = f_c \quad \Delta t + K \quad \left(f_0 - f_c\right)\left(1 - e^{-\Delta t\!/\!K}\right) \qquad \text{for } y>0$$

Equations [7-47] - [7-49] are solved using a Newton Raphson method to yield a solution for $y2$ which is then used to obtain a value for Q.

It can be shown (Smith, 1986a, see references) that the algorithm developed above is equivalent to convoluting the storm rainfall with a Dirac δ–function and then routing the resulting 'instantaneous' runoff through a nonlinear reservoir with storage characteristics given by:

$$[7.50] \qquad S = C \quad Q^{0.6}$$

where

$$C = C_M^{\;0.6}\left(\frac{L\,n}{S^{0.5}}\right)^{0.6} A^{0.4}$$

In equation [7-50] '***A***' is the catchment drainage area and other terms are as defined previously (see equations [7-41] and [7-45]).

Three points of some significance arise with respect to the ‘SWMM Method’ option.

(1) Considering the method to be equivalent to routing the instantaneous runoff through a nonlinear reservoir, it follows that the peak of the outflow must lie on the recession limb of the inflow. Consequently the time to peak for pervious and

impervious fractions will not differ significantly and the total runoff will not exhibit the double peaked hydrographs which are sometimes encountered with the 'Rectangular' or 'Triangular SCS' options.

(2) The form of equations [7-44] and [7-45] implicitly assumes that the depth of flow over the plane is quasi-uniform. This over-estimates the volume on the plane and will usually result in over-attenuation of the peak runoff.

(3) Since infiltration is assumed to continue over the entire surface after cessation of rainfall as long as the average depth is finite, the recession limb of the runoff hydrograph will generally be much steeper than for the 'Rectangular' or 'Triangular SCS' options. In practice, after cessation of rainfall, the surface water tends to concentrate in pools and rivulets so that the area over which the infiltration continues is likely to be much less than the total area ***A***. A more realistic representation of the infiltration after the storm is likely to be intermediate between the two extreme cases represented by the 'SWMM Method' method on one hand and the 'Triangular SCS' or 'Rectangular' method which employs the concept of effective rainfall. This feature is sufficiently important that a detailed example is presented in the following section in order to illustrate the fundamental difference between the methods.

7.3.10 An Example of the SWMM Runoff Algorithm

The object of this example is to compare the overland flow that is generated by the 'SWMM Method' option with that which would be obtained using an effective rainfall approach. For simplicity we shall assume a catchment of 5.0 ha with no impervious area and no depression surface storage. The storm used is a 3rd quartile Huff storm with a total rainfall depth of 30 mm occurring in 60 minutes. Infiltration will be modelled by the Horton method with the following parameter values:

- n = 0.25
- $f0$ = 40 mm/hour
- fc = 20 mm/hour
- K = 0.25 hours
- y_d = 0

To simulate the SWMM algorithm using an effective rainfall approach we shall make use of equation [7.50] to define a nonlinear reservoir through which the instantaneous runoff is routed. This hydrograph can be created by convoluting the effective rainfall with an impulse (also known as a Dirac δ-function) which can be simulated by specifying a very short overland flow length.

The steps are summarized as follows. You may find it instructive to run this example on your own computer as you read through the steps.

- In the Time Parameters use 2 minute timesteps and a storm duration of 60 minutes.
- Define the Huff storm; use 30 mm rainfall; 60 minutes duration; 3rd quartile.
- The impervious characteristics are not important but we must use the Horton method. Set n = 0.015 and the other parameters to zero.

- The first catchment 101 is used to represent the Dirac δ-function so use the following parameters.

Area = 5.0 ha

Length = 0.1 m

Slope = 2.0 %

Percent impervious = 0

For the infiltration parameters use the Horton method with:

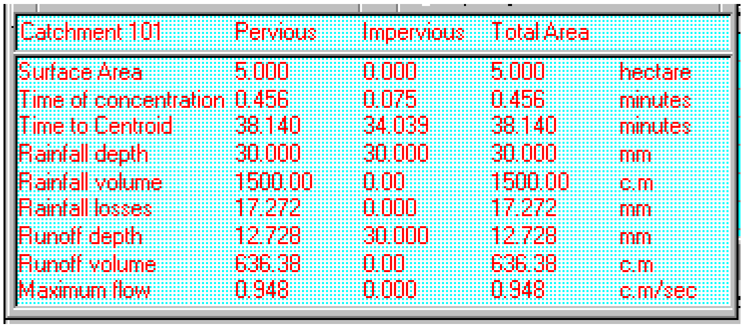
fo = 40 mm/hour

fc = 20 mm/hour

K = 0.25 hour

yd = 0

The peak effective rainfall intensity is found to be 68.279 mm/h. Use the ‘Rectangular’ option since this most closely approximates an impulse. The peak runoff is 0.948 c.m/s. A few seconds with a calculator will confirm that for an area of 5 hectares this is equivalent to 68.279 mm/h.

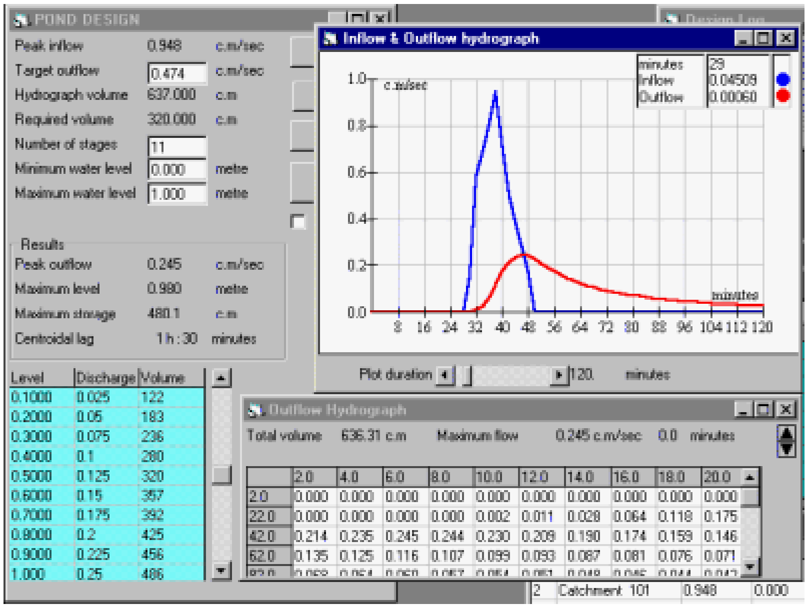


Catchment 101	Pervious	Impervious	Total Area	
Surface Area	5.000	0.000	5.000	hectare
Time of concentration	0.456	0.075	0.456	minutes
Time to Centroid	38.140	34.039	38.140	minutes
Rainfall depth	30.000	30.000	30.000	mm
Rainfall volume	1500.00	0.00	1500.00	c.m
Rainfall losses	17.272	0.000	17.272	mm
Runoff depth	12.728	30.000	12.728	mm
Runoff volume	636.38	0.00	636.38	c.m
Maximum flow	0.948	0.000	0.948	c.m/sec

Figure 7-23 – Statistics of the Dirac-δ response hydrograph

- We shall want to route this runoff through an imaginary pond with stage discharge characteristics as given by [7- 50]. Use the **Hydrology/Add Runoff** command to define the inflow to the pond to be used in step (7).

- The final step to simulate the SWMM hydrograph is by routing the instantaneous runoff through a nonlinear reservoir. For the data used in this example, the value of C in equation [7-50] works out to be 1115.39. We now define a pond with discharges ranging from 0.0 to 0.25 in increments of 0.025 - i.e. 11 stages. Figure 7-25 shows the result of a pond design. The value of each storage volume is given by 1115.39 x $Q^{0.6}$. The peak outflow is found to 0.246 c.m/s In Figure 7-25, the hydrograph has been extended to 130 minutes.



Peak inflow	0.948	c.m/sec
Target outflow	0.474	c.m/sec
Hydrograph volume	637.000	c.m
Required volume	320.000	c.m
Number of stages	11	
Minimum water level	0.000	metre
Maximum water level	1.000	metre

Results

Peak outflow	0.245	c.m/sec
Maximum level	0.980	metre
Maximum storage	480.1	c.m
Centroidal lag	1 h : 30	minutes

Level	Discharge	Volume
0.1000	0.025	122
0.2000	0.05	183
0.3000	0.075	236
0.4000	0.1	280
0.5000	0.125	320
0.6000	0.15	357
0.7000	0.175	392
0.8000	0.2	425
0.9000	0.225	456
1.000	0.25	486

minutes	29
Inflow	0.04509
Outflow	0.00060

Plot duration 120. minutes

Total volume 636.31 c.m Maximum flow 0.245 c.m/sec 0.0 minutes

	2.0	4.0	6.0	8.0	10.0	12.0	14.0	16.0	18.0	20.0
2.0	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
22.0	0.000	0.000	0.000	0.000	0.002	0.011	0.028	0.064	0.118	0.175
42.0	0.214	0.235	0.245	0.244	0.230	0.209	0.190	0.174	0.159	0.146
62.0	0.135	0.125	0.116	0.107	0.099	0.093	0.087	0.081	0.076	0.071

2	Catchment 101	0.948	0.000

Figure 7-24 - Design of a hypothetical pond.

- The next step is to generate the 'SWMM method' hydrograph, so define another catchment with the same parameters as in step (4) but with a flow length of 50 m. The infiltration options are the same as before. The peak runoff is found to be 0.248 c.m/s.

The similarity in peak flows is promising, but the true test is to compare the plotted hydrographs. Figure 7-24 shows the 'SWMM Method' and simulated SWMM hydrographs

The rising limbs of the two hydrographs are in good agreement apart from a slight lag of about 2 minutes which is the shortest 'impulse' that MIDUSS can create when Δt is 2 minutes. However, immediately following the cessation of the effective rainfall the 'SWMM Method' recession limb drops more steeply. This is due to the fact that the surface water budget method assumes that infiltration continues as long as there is excess water on the pervious surface whereas the effective rainfall approach - which produced the longer curve in Figure 7-25 - assumes that infiltration stops at the end of the effective rainfall. The two recession limbs start to diverge at t = 50 min. which marks the end of the effective rainfall hyetograph.

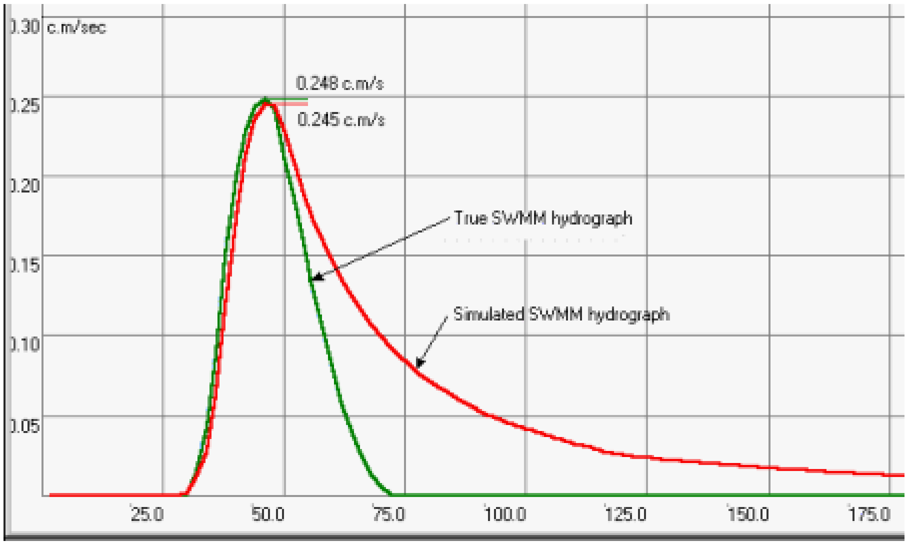
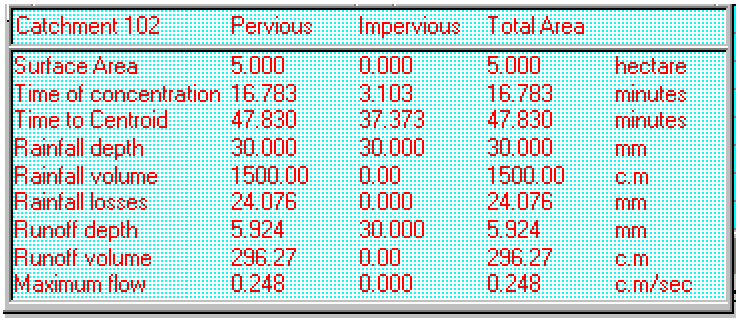


Figure 7-25 - Comparing the SWM HYD and simulated SWMM hydrographs.



Catchment 102	Pervious	Impervious	Total Area	
Surface Area	5.000	0.000	5.000	hectare
Time of concentration	16.783	3.103	16.783	minutes
Time to Centroid	47.830	37.373	47.830	minutes
Rainfall depth	30.000	30.000	30.000	mm
Rainfall volume	1500.00	0.00	1500.00	c.m
Rainfall losses	24.076	0.000	24.076	mm
Runoff depth	5.924	30.000	5.924	mm
Runoff volume	296.27	0.00	296.27	c.m
Maximum flow	0.248	0.000	0.248	c.m/sec

Figure 7-26 - Statistics of the 'SWMM Method' hydrograph.

This difference serves also to explain the anomaly that appears in the hydrograph statistics screen when using 'SWMM Method' option. Figure 7-26 shows the summary statistics obtained at the end of step (7) and you will note that the runoff volume (296.2 c.m) is much less than that for the effective rainfall volume of 636.38 c.m. (from Figure 7-23).

This may not always be the case and you should repeat this experiment with a finite value for depression surface storage - say 2 mm or 100 c.m over the 5 hectares of area. You will find that the effective rainfall volume is reduced by exactly 100 c.m. The infiltration and runoff volume are also reduced by amounts which add up to 100.0 c.m. less the volume still trapped in surface depressions when the calculation was ended. If continued long enough, this too would have infiltrated thus balancing the books properly. Hence the name 'surface water budget'.

To assist you in trying some experiments, a full listing of the output file is included in the Miduss98\Samples\ folder when you install MIDUSS..

7.4 Simulation of Large Catchments

Rainfall-runoff simulation requires certain assumptions with respect to the level of discretization to be employed and the parameter values to be used for the sub-areas. When modelling very large watersheds a compromise is necessary between using sub-areas that are too small or too large. Small sub-catchments impose cost penalties in data preparation and computation effort. Large areas present problems in assigning values to parameters – such as overland flow length - that are a reasonable representation of the physical system.

The ratio of channel travel time to sub-area response time varies widely for areas that are close to or distant from the outflow point. This variation results in a diffusion of the flow peaks from individual areas and accounts for a significant part of the basin lag. In large catchments this basin lag can equal or exceed the overland flow travel time and attempts to represent this by distorting overland flow parameters are subjective, unrealistic and storm specific. The process can be better represented by convoluting the overland flow response function with the derivative of the time-area diagram for the total watershed. The resulting modified response function can then be convoluted with the effective rainfall to yield a good approximation to the runoff hydrograph for the total area.

Numerical experiments suggest that the unwieldy process of double convolution can be approximated by using a single convolution of overland flow and rainfall and then routing the resulting hydrograph through a hypothetical linear channel and linear reservoir. The latter have lag times that are related to the maximum conduit travel time through the drainage network. MIDUSS uses some preliminary guidelines to estimate the lags and partially automate the process.

This section describes the process used and compares a typical example with a fully discretized simulation. It must be emphasized that the suggested method is preliminary and would benefit from further testing of either real or idealized cases to verify or improve the guidelines.

7.4.1 Example of a Large Catchment

The process is described with reference to the catchment area in Figure 7-27. The runoff obtained from the discretized version will be compared to the approximate 'lumped-parameter’ version. In Figure 7-27 the area in hectares of the sub-areas is shown in italics and in parenthesis in the lower-right corner of each rectangular area.

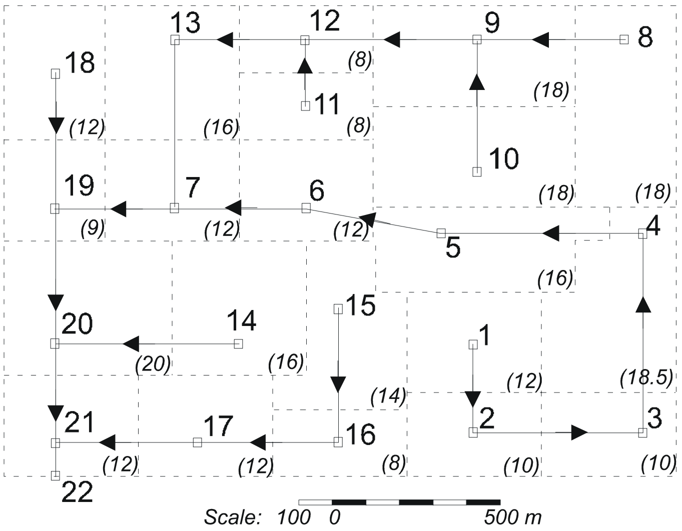


Figure 7-27 – Discretized version of a large catchment

(Areas shown in hectares as *(12)* in lower right corners)

The system is subjected to a 5-year storm represented by a 360 minute Chicago hyetograph with a total depth of 50.45 mm (the MIDUSS default values). All the sub-areas are assumed to have the same overland flow characteristics with the exception of area, i.e.

Overland flow length = 45 m

Overland slope = 2.0 %

Percent impervious = 30 %

Pervious roughness n = 0.25

Impervious roughness n = 0.013

With this simplifying assumption, the response functions of the sub-areas will have the same time parameters and vary only with respect to the area. Thus, for a triangular response function:

[7.51] $$t_c = k\left(\frac{L\,n}{\sqrt{S_0}}\right)^{0.6} i_{eff}^{\;-0.4}$$

[7.52] $$t_p = 0.6t_c + {}^{\Delta t}\!/_{2}$$

[7.53] $$t_b = \left({}^{8}\!/_{3}\right)t_p$$

[7.54] $$u_{\max} = {}^{2A}\!/_{t_b}$$

The drainage network is composed of pipes with a gradient of approximately 0.4%. The output file from the discretized simulation is called ‘Large1.out’ and can be found in the ..\Samples\ folder of the Miduss98 directory.

The distribution of areas relative to the outflow point can be represented by a time-area diagram as illustrated in Figure 7- 28. From Figure 7-27 you will note the rather circuitous (and unrealistic) drainage path of areas 1, 2 and 3. This gives rise to the late contribution of the furthest upstream 32 ha which in turn makes the Time-Area diagram depart from the reasonably linear shape which is apparent over the first 15 minutes. This feature makes the approximation more challenging.

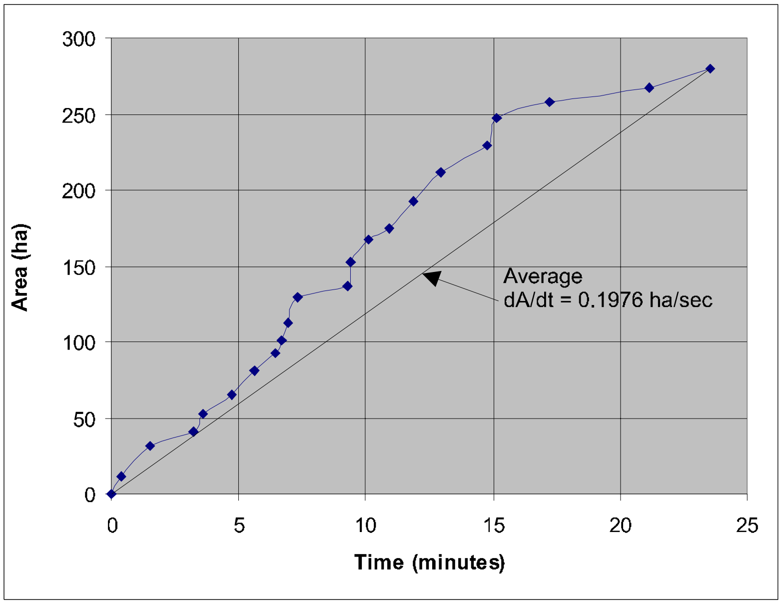


Figure 7-28 – Time-Area diagram for the catchment of Figure 7-27

The time to equilibrium *Te* is 23.21 minutes from node #1 to the outflow point at node #22. This is the time at which the entire catchment is contributing and is a function of the drainage network. The time of concentration *tc* and therefore the timebase *tb* of the response function, is a characteristic of the overland flow. Both quantities are also dependent on the magnitude of the storm.

The relative magnitude of *Te* and *tc* is an important parameter in determining the limit for lumped representation of a catchment. If $Te / tc << 1.0$ then it is likely that overland flow dominates the runoff process and thus the overland flow response function is a reasonable approximation for the entire catchment. In large catchments, Te / tc is larger (although still probably less than 1.0) and channel/pipe routing will play an important role in determining the shape and peak of the runoff hydrograph.

7.4.2 Combining Overland Flow and Drainage Network Routing

The combined effect of overland flow routing and drainage network routing can be obtained by convoluting one response function with the other. The right side of Figure 7-29 shows a triangular response function being convoluted with the derivative of the time-area diagram to produce a modified response function. This is then convoluted with the hyetograph of effective rainfall to produce the modified runoff hydrograph in the lower right corner of the figure.

The unwieldy process of double convolution can be approximated by the process shown on the left side of the Figure 7-29. The normal overland flow response function is convoluted with the effective rainfall to produce a ‘lumped’ runoff hydrograph. This assumes that all the sub-areas contribute to runoff simultaneously. This is then routed through a linear channel and linear reservoir. If appropriate values can be set for *Kch* and *Kres* the resulting hydrograph should be a close approximation of the modified runoff hydrograph.

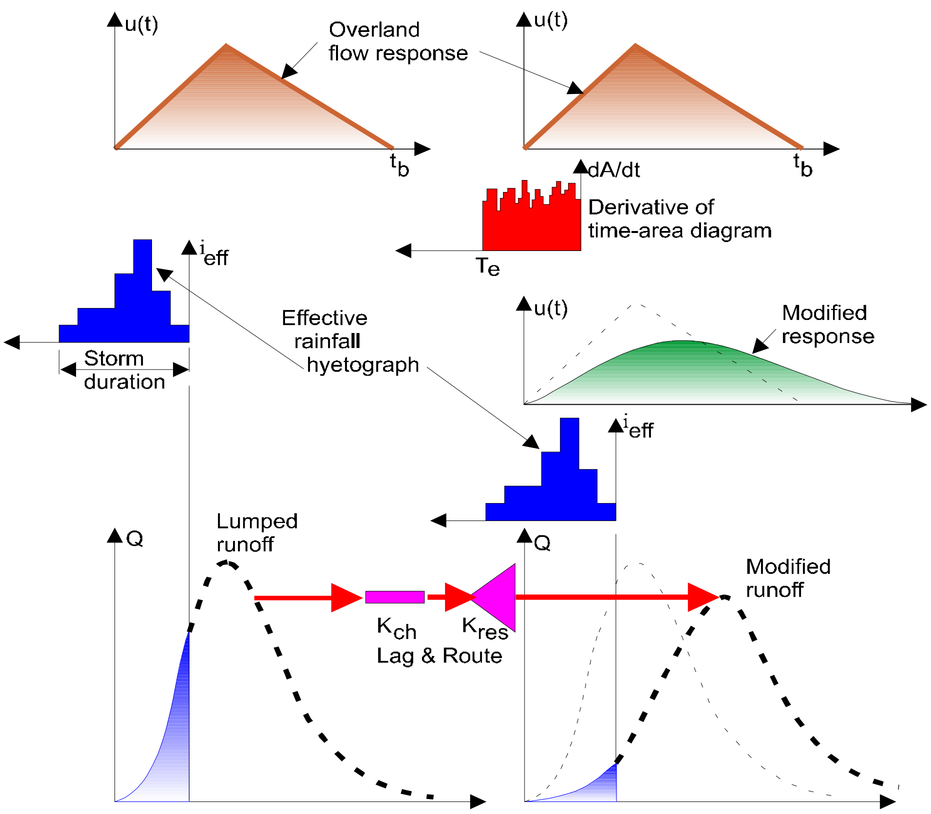


Figure 7-29 – Representation of the Lag and Route method

The total lag *Ktot* is defined as the sum of the two components *Kch* and *Kres*, i.e.

[7.54] $$K_{tot} = K_{ch} + K_{res}$$

The distribution of the total lag between the two constituent parts is defined by a fraction r ($0.0 < r < 1.0$) as follows.

[7.55] $$K_{ch} = \left(1 - r\right) K_{tot}$$

and

[7.56] $$K_{res} = r\,K_{tot}$$

7.4.3 Estimating the Lag Values

Results from a limited number of numerical experiments suggest that some correlation exists between:

- The total lag *Ktot* and the ratio of time to equilibrium to time of concentration (*Te/tc*), and

- The fraction *r = Kres/Ktot* and the basin time to equilibrium *Te*

As preliminary guidelines the following relationships are used in MIDUSS.

$$[7.57] \qquad K_{tot} = T_e\left(0.4 + 0.005\frac{T_e}{t_c}\right)$$

Values for the fraction *r = Kres/Ktot* are based on a curve which – for the data analyzed – approaches an asymptotic value of about 0.55 as shown in Figure 7-30.

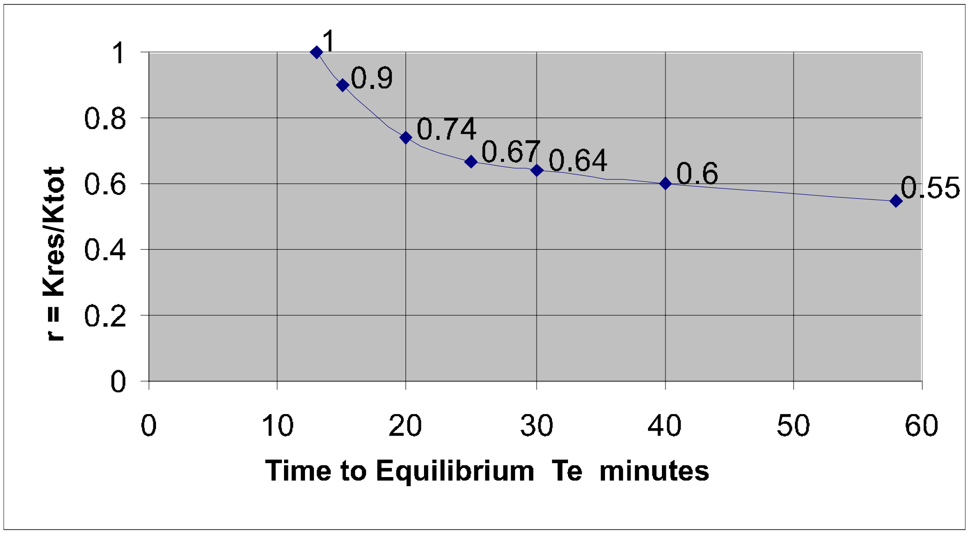


Figure 7-30 – Empirical curve defining r = Kres/Ktot = f(Te)

The data of Figure 7-30 is contained in a small data file called 'LagRout1.dat' which resides in the Miduss98 folder. This file can be edited or updated as and when further numerical experimental results are available. The trends suggested are interesting but inconclusive. More experiments are required to test the sensitivity of the identified parameters to factors such as:

- Shape and duration of storm
- Size and shape of the catchment
- Choice of overland routing model
- The rainfall loss model employed.

Until such time as further test results are available you should use this feature with caution. When possible, it is useful to carry out a comparison between the Lag and Route approximation and a typical discretized simulation to provide a measure of confidence in the method. The next section describes the results obtained for the catchment of Figure 7-27.

7.4.4 Comparison of Discretized and Approximate Results

Refer to the output file …\MIDUSS\Samples\Large1.out for details of the test described here.

The fully discretized simulation produced a peak runoff of 19.136 c.m/s.

The lumped catchment runoff is found to have a peak of 26.488 c.m/s

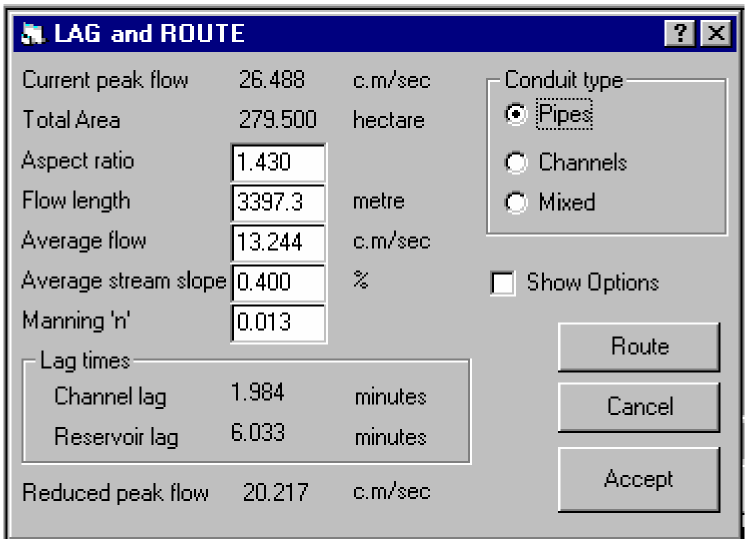
The Lag and Route command is then used with MIDUSS default values for all quantities with the exception of the catchment area aspect ratio which is set at 2000m/1400 m or 1.43 and the average pipe slope of 0.4%. The form is shown in Figure 7-31 below. The longest drainage path estimated by MIDUSS is 3397 m whereas scaling the reach from node #1 the length is 3900 m. The underestimate is close to 13% and is due to the circuitous route from node #1 to node #5. The error will result in a slightly higher peak flow for the reduced peak flow that is shown as 20.217 c.m/s. A graphical comparison of the results is shown in Figure 7-32. Apart from the over-estimated peak flow the approximation is reasonable.

If the Stream length is entered as 3900 m as a result of scaling the drawing (Figure 7_27) the result is improved. The peak of the approximate runoff is reduced to 19.745 c.m/s with no measurable change in the general agreement between the discretized and approximate runoff hydrographs. The effect of the change in stream length is summarized in the Table below.

Stream length (m)	**Kch (min)**	**Kres (min)**	**Ktot (min)**	**r**	**Qpeak (c.m/s)**
3397	1.984	6.033	8.017	0.753	20.217
3900	2.725	6.508	9.233	0.705	19.745
Discretized simulation					19.136

By checking the output file you will also see that continuity is respected and the total runoff volume is given as 6.2605 ha-m in all cases.

You can experiment with this example by running the file ‘Large1.out’ in automatic mode. After generating the database Miduss.Mdb, navigate to the **Hydrograph/Start new tributary** command following completion of the discretized simulation. Change the command from ‘40’ to ‘- 40’ and then use the [RUN] button in the Automatic Control Panel to run up to that point. You can then step through the ‘lumped’ runoff calculation and the Lag and Route approximation using the [EDIT] command button or in Manual mode



LAG and ROUTE

Current peak flow 26.488 c.m/sec

Total Area 279.500 hectare

Aspect ratio 1.430

Flow length 3397.3 metre

Average flow 13.244 c.m/sec

Average stream slope 0.400 %

Manning 'n' 0.013

Lag times

Channel lag 1.984 minutes

Reservoir lag 6.033 minutes

Reduced peak flow 20.217 c.m/sec

Conduit type

● Pipes

○ Channels

○ Mixed

☐ Show Options

Figure 7-31 – Using the Lag and Route command

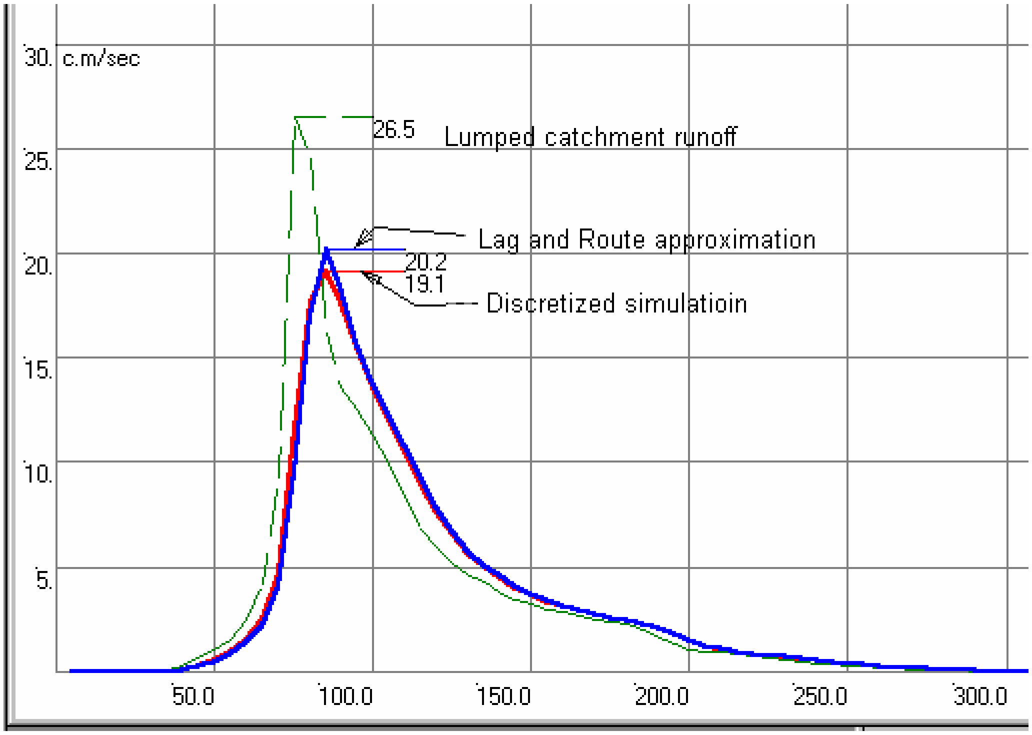


Figure 7-32 – Comparison of the Lag and Route approximation with the discretized runoff.

Chapter 8 Theory of Hydraulics

This chapter explains some of the principles of hydraulics which are used in the Design commands available in MIDUSS. It is not intended to be a general treatment of hydraulics and you should use a standard text on the subject to obtain information not covered in this Help System.

The chapter is subdivided into six sections corresponding to 6 commands available in the Design menu.

- Pipe design
- Channel design
- Flood Routing
- Detention Pond design
- Exfiltration Trench design
- Diversion Structure design

8.1 Theory of Pipe Design

8.1.1 Theory of Pipe Design

This section summarizes the hydraulic principles which are used in MIDUSS for the analysis and design of pipes. Flow is assumed to be uniform within each reach of pipe, so that the depth and other cross-sectional properties are constant along the length of the pipe. It follows that the bed slope $S0$, the water surface and the slope of the energy line Sf are all parallel. The resistance is assumed to be represented by the Manning equation:

$$[8.1] \qquad Q = \frac{M}{n} A R^{2/3} S_0^{1/2}$$

where Q = normal discharge (c.m/s or c.ft/s)

M = 1.0 for metric units

1.49 for imperial or US customary units

n = Manning's roughness coefficient

A = cross-sectional area

R = hydraulic radius = Area/Wetted perimeter

$S0$ = bed slope (m/m or ft/ft)

No allowance is made for any apparent variation of 'n' with the relative depth of flow in the pipe.

8.1.2 Normal Depth in Pipes

For a part-full circular section the cross-sectional properties are expressed in terms of the angle ϕ subtended at the centre by the free surface as shown in Figure 8.1.

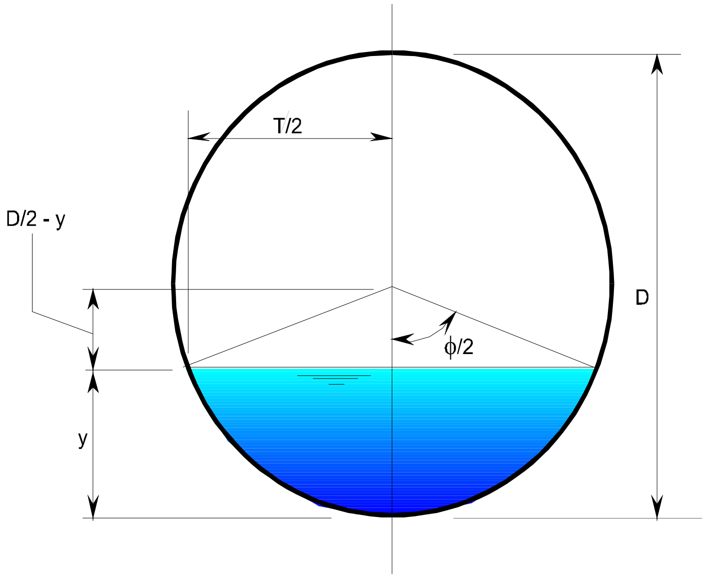


Figure 8.1 Definition sketch of a part-full pipe.

The following equations can be obtained by considering the geometry of the triangle subtending the half-angle $\phi/2$ at the centre of the pipe.

[8.2] $$y = \frac{D}{2}\left(1 - \cos\frac{\phi}{2}\right)$$

[8.3] $$A = \frac{D^2}{8}(\phi - \sin\phi)$$

[8.4] $$P = D\frac{\phi}{2}$$

The value of ϕ can be found in terms of the ratio of the discharge Q to the full-bore pipe capacity $Qfull$ by an iterative solution of the implicit equation [8.5].

[8.5] $$f(\phi) = \phi - \sin\phi - C_2 = 0$$

where

$$C_2 = C_1\phi^{2/5} = \left(\frac{2\pi Q}{Q_{full}}\right)^{3/5}\phi^{2/5}$$

Equation [8.5] is solved by a Newton-Raphson procedure, thus:

[8.6] $$\phi_{k+1} = \phi_k - \Delta\phi_k = \phi_k - \frac{f(\phi_k)}{f'(\phi_k)}$$

where

$$f(\phi) = \phi - \sin(\phi) - C_2$$

and

$$f'(\phi) = 1 - \cos(\phi) - 0.4\frac{C_2}{\phi}$$

Equation [8.6] is applied until $\Delta\phi < 0.001$ radians; the depth is then determined from equation [8.7].

[8.7] $$y = \frac{D}{2}\left(1 - \cos\frac{\phi}{2}\right)$$

For a cross-section with a closed top it is usual to find that maximum normal discharge occurs at a depth slightly below full-bore flow. For a circular pipe this occurs at a relative depth of y/D = 0.93818. It follows that there must be a smaller depth which produces a discharge equal to the full-bore flow. In a part-full pipe this occurs when y/D = 0.81963.

The root finding procedure in MIDUSS will always find a solution within the relative depth range 0.0 < (y/D) < 0.81963 as long as the discharge is less than the full-bore flow. If the discharge is greater than this then MIDUSS reports that the pipe will be surcharged and the slope of the hydraulic grade line is reported. (See Chapter 4 *Design Options Available,* Surcharged Pipe Design)

It is not possible, therefore, to take advantage of the slightly higher carrying capacity in the range 0.81963<(y/D)<1.0. It is not normally good practice to design pipes for uniform flow in this range of depth because the slightest surface disturbance will cause the free surface to 'snap through' abruptly to a condition of pressurized flow.

8.1.3 Critical Depth in Pipes

When a pipe is designed it is often important to know if the normal flow depth $y0$ is less than or greater than the critical depth ycr. If $y0 < ycr$ then the flow is supercritical and there is a high probability that a hydraulic jump will occur at some point downstream. This is usually to be avoided.

The calculation of critical depth in a circular pipe is based on the critical flow condition of minimum specific energy which leads to the criterion of equation [8.8].

[8.8] $$\frac{Q^2 T}{g A^3} = 1$$

This is solved by an interval halving procedure using a function of the form:-

[8.9] $$f(y) = \frac{A^3}{T} - \frac{Q^2}{g} = 0$$

in which A is obtained by combining equation [8.3] with equations [8.10] and [8.11] below.

[8.10] $$T = 2\sqrt{\left(Dy - y^2\right)}$$

[8.11] $$\phi = 2\tan^{-1}\left(\frac{T/2}{\left(D/2 - y\right)}\right) = 2\tan^{-1}\left(\frac{T}{D - 2y}\right)$$

Convergence is assumed when $\Delta y/y < 0.00001$.

Equation [8.9] cannot be solved if the free-surface width T is zero. A test is therefore made to ensure that the specified discharge is not greater than the critical discharge corresponding to a depth of $ycr = 0.999\,D$. If this condition is violated MIDUSS assumes the critical depth to be equal to the diameter. For further information on uniform or critical flow in pipes see a text on Open Channel Flow such as Henderson (References).

8.2 Theory of Channel Design

This section summarizes the methods used to analyze the channel for uniform and critical flow depth. As with the **Pipe** command, each reach of channel is assumed to be prismatic, that is, of constant cross-section and slope. As long as the channel flow has a free surface, the flow in each reach is assumed to be quasi-uniform, neglecting the variation of flow with time. For this condition the friction slope Sf and the water surface are assumed to be parallel to the bed slope $S0$. The resistance is assumed to be represented by the Manning equation [8.12].

MIDUSS lets you define the cross-sectional shape of the channel either as a trapezoidal shape as shown in Figure 8.2 or as an arbitrary cross-section defined by the coordinates of up to 50 points.

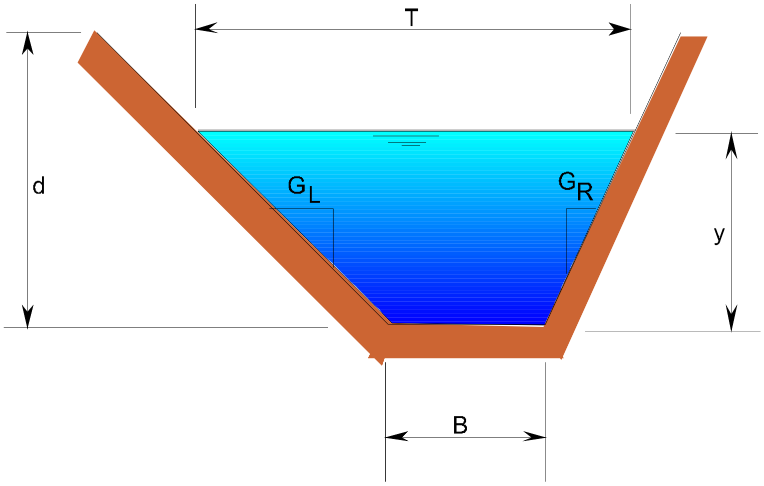


Figure 8.2 - Definition of a trapezoidal section.

Figure 8.3 shows a cross-section defined by 9 points. For the purpose of illustration, the remainder of this section assumes that the cross-section is trapezoidal in shape. MIDUSS uses a simple routine to process the coordinates of a more complex section with a specified water surface elevation to yield the same cross-sectional properties.

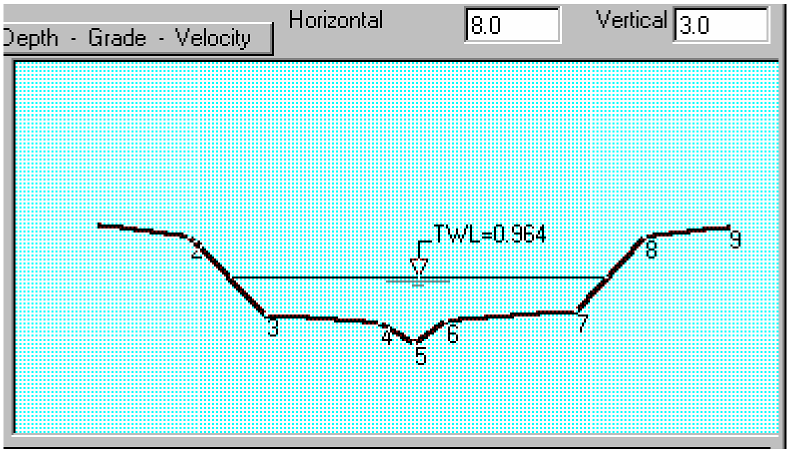

Horizontal 8.0
Vertical 3.0

Figure 8.3 - An arbitrary cross-section using 9 points.

8.2.1 Normal Depth in Channels

Figure 8.2 shows a cross-section of arbitrary trapezoidal shape, of total depth d with a flow depth y. Using the Manning equation the normal discharge Q for any given depth y is given as follows.

[8.12] $$Q = \frac{M}{n} A R^{2/3} S_0^{1/2}$$

where Q = normal discharge (c.m/s or c.ft/s)

M = 1.0 for metric units

1.49 for imperial or US customary units (3.28 ft/m)^(1/3)

n = Manning's roughness coefficient

A = cross-sectional area

R = hydraulic radius = Area/Wetted perimeter

$S0$ = bed slope (m/m or ft/ft)

Evaluation of the cross-section properties depends on whether a simple trapezoidal cross-section or a more complex cross-section is defined. For a general trapezoidal shape the following equations are used.

[8.13a] $$T = B + (G_L + G_R)y$$

[8.13b] $$A = y\frac{(T+B)}{2} = \frac{y}{2}\left(2B + y(G_L + G_R)\right)$$

[8.13c] $$P = B + y\left(\sqrt{1+G_L^2} + \sqrt{1+G_R^2}\right)$$

[8.13d] $$R = A/P$$

where B = base width

T = top width

P = wetted perimeter

y = depth of flow.

GL = slope of the left bank (GL horiz : 1 vert)

GR = slope of the right bank (GR horiz : 1 vert)

and other terms are as previously defined.

The maximum carrying capacity Qfull for flow with a free surface is found from equation [8.12] setting $y = d$. If the peak discharge Q is less than $Qfull$ the depth of uniform flow is found by an interval halving technique. Convergence is assumed when $\Delta y/y < 0.000001$.

The hydraulic gradient is then computed by equation [8.14].

$$[8.14] \qquad S_f = \left(\frac{Vn}{M}\right)^2 \frac{1}{R^{4/3}}$$

8.2.2 Critical Depth in Channels

The calculation of critical depth in a channel assumes that a free surface exists. MIDUSS does not check to see if the critical depth is less than the specified total depth d.

If the base-width is finite but the side slopes are vertical the cross-section is rectangular and the critical depth can be calculated explicitly by equation [8.15].

$$[8.15] \qquad y_{cr} = \left(\frac{Q^2}{gB^2}\right)^{1/3}$$

If the basewidth is zero and at least one of the side slopes are finite the cross-section is triangular and again an explicit solution for *ycr* can be found from equation [8.16].

$$[8.16] \qquad y_{cr} = \left(\frac{8Q^2}{g(G_L + G_R)^2}\right)$$

For the case of a general trapezoidal cross-section an iterative solution is required to solve the critical flow criterion of equation [8.8]. This involves the application of the Newton-Raphson method (equation [8.6]) in which the function and its derivative are defined by equations [8.17] and [8.18] respectively.

$$[8.17] \qquad f(y) = \frac{A^3}{T} - \frac{Q^2}{g}$$

$$[8.18] \qquad f'(y) = \frac{3A^2}{T}\left(\frac{dA}{dy}\right) - \frac{A^3}{T^2}\left(\frac{dT}{dy}\right)$$

$$= 3A^2 - \frac{A^3}{T^2}(G_L + G_R)$$

Convergence is assumed when $\Delta y/y < 0.0001$. More information on the hydraulics of open channels can be found in many standard texts (See Reference)

8.3 Theory of Kinematic Flood Routing

Flood routing methods can be classified as hydraulic - in which both continuity and dynamic equations are used - or hydrologic, which generally uses the continuity equation alone. MIDUSS uses a method based on a kinematic wave equation and therefore falls into the second category. For the type of conduits used in storm drainage systems, kinematic routing yields results of very acceptable accuracy.

The continuity equation is simply a statement that the difference between inflow and outflow must equal the rate of change of storage in the reach being considered. Equation [8.20] is a general continuity equation in which lateral inflow is ignored.

[8.20] $$\frac{\partial Q}{\partial x}+\frac{\partial A}{\partial t}=0$$

If the flow in the channel can be assumed to be quasi-uniform then discharge is uniquely defined by the depth or stage, i.e.

[8.20] $$Q = f\left(WL\right)$$

so that

[8.21] $$\frac{\partial Q}{\partial A}=\frac{dQ}{dA}$$

Now separating variables in [8.20] yields:

[8.22] $$\frac{\partial Q}{\partial x}=-\frac{dA}{dQ}\left(\frac{\partial Q}{\partial t}\right)=-\frac{1}{c}\frac{\partial Q}{\partial t}$$

Equation [8.22] is seen to be a wave equation in which a function $Q(x)$ is propagated with a celerity c which is given by $c = dQ/dA$. The variation of Q with respect to both time t and space x can be best described using a space-time coordinate system which defines changes in discharge Q in an elementary reach Δx and over a timestep Δt.

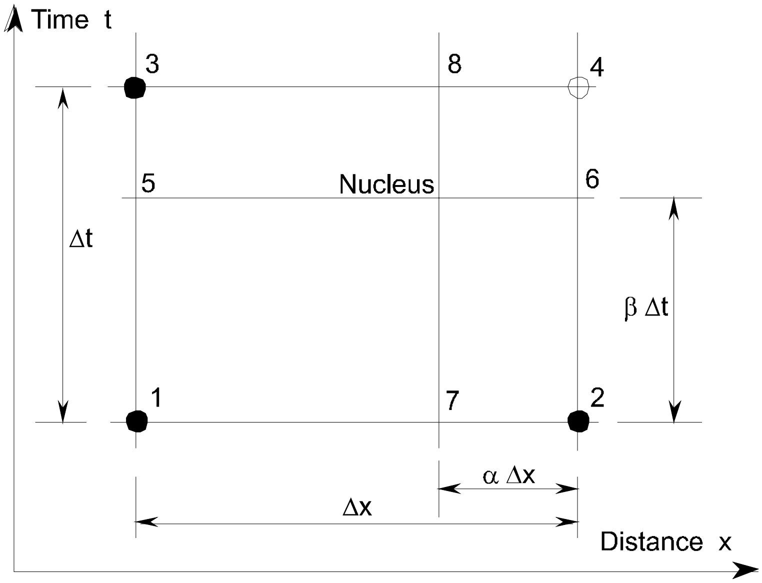


Figure 8.4 - An element of a space-time coordinate system.

Figure 8.4 shows such a situation. For this element of the space-time system equation [8.22] can be expressed in terms of finite differences as follows.

$$\frac{\Delta Q}{\Delta x}+\frac{1}{c}\frac{\Delta Q}{\Delta t} \tag{8.23}$$

This equation is applied around a 'nucleus' of the space-time element which is off-centre and defined by the weighting factors α and β which are applied to the x and t dimensions respectively as shown in Figure 8.4.

The finite difference quotients of equation [8.23] are expanded around the 'nucleus' point of Figure 8.4 in terms of the values of Q at points 5, 6, 7 & 8. These in turn can be expressed as weighted averages of the values at points 1, 2, 3 & 4 and the weighting coefficients α and β. Thus:

$$\frac{\Delta Q}{\Delta x}=\frac{Q_6-Q_5}{\Delta x} \quad \text{and} \quad \frac{\Delta Q}{\Delta t}=\frac{Q_8-Q_7}{\Delta t} \tag{8.24}$$

where

$$Q_5=\beta Q_3+(1-\beta)Q_1$$

$$Q_6=\beta Q_4+(1-\beta)Q_2$$

$$Q_7=\alpha Q_1+(1-\alpha)Q_2$$

$$Q_8=\alpha Q_3+(1-\alpha)Q_4$$

Substituting equations [8.24] in [8.23] yields:

[8.25] $$\left(\frac{c\ \ \Delta t}{\Delta x}\right)(Q_6 - Q_5) + Q_8 - Q_7 = 0$$

The quantity ($c.\Delta t/\Delta x$) is a dimensionless time ratio which is equivalent to the Courant criterion for numerical stability. Denoting this by τ and substituting for $Q5$ etc. results in equation [8.26].

[8.26] $$\tau\beta Q_4 + \tau(1-\beta)Q_2 - \tau\beta Q_3 - \tau(1-\beta)Q_1 + \\ \alpha Q_3 + (1-\alpha)Q_4 - \alpha Q_1 - (1-\alpha)Q_2 = 0$$

Now the process of flood routing usually involves a 'marching' solution in which the initial conditions are known at time t and it is required to predict conditions at time ($t+\Delta t$). An upstream boundary condition is provided by the time-history of the inflow hydrograph at $x=0$. In the case of kinematic wave routing it is possible to advance the solution for all values of time *t* so that the solution advances over the whole time domain for each reach of channel.

In either case, the solution for the element of Figure 8.4 involves an estimate of $Q4$ in terms of the other three known values. Collecting terms and casting $Q4$ as the dependant variable yields:

[8.27] $$C_4Q_4 = C_1Q_1 + C_2Q_2 + C_3Q_3$$

where

$$C_1 = \alpha + (1-\beta)\tau$$

$$C_2 = (1-\alpha) - (1-\beta)\tau$$

$$C_3 = -\alpha + \beta\tau$$

$$C_4 = (1-\alpha) + \beta\tau$$

Equation [8.27] is a generalized form of the Muskingum flood routing method but for the special case of β=0.5 this reduces to the more familiar form shown below.

$$C_1 = \alpha + 0.5\tau = X + \frac{\Delta t}{2K}$$

[8.28] $$C_2 = (1-\alpha) - 0.5\tau = 1 - X - \frac{\Delta t}{2K}$$

$$C_3 = -\alpha + 0.5\tau = -X + \frac{\Delta t}{2K}$$

$$C_4 = (1-\alpha) + 0.5\tau = 1 - X + \frac{\Delta t}{2K}$$

Before equation [8.27] can be applied two additional pieces of information are required:

(i) What values of α and β should be used to best represent the attenuation for a specific hydraulic condition?

(ii) What conditions must apply for the computation to be numerically stable?

8.3.1 Evaluation of the Weighting Coefficients

Convergence is the condition in which the solution of a finite-difference equation for a finite grid size approximates the true solution of the partial differential equation which it represents. It can be shown (Biesenthal (1975) and Smith (1980)) for the non-centred scheme of Figure 8.4 that as the coefficients α and β depart from a value of 0.5, truncation errors of the order of $O(\Delta x)$ and $O(\Delta t)$ increase respectively and independently. This property is fundamental to the use and apparent success of kinematic routing methods in modelling attenuation. It should be emphasized, however, that this attenuation results from truncation error and is a property of the numerical finite difference scheme and not of the physical system. The trick is to find a way to make the numerical truncation error a close approximation to the attenuation which the flood wave will experience.

For a non-centred finite-difference scheme the error may be included in the partial differential equation as ε in equation [8.29].

[8.29] $$\frac{\partial Q}{\partial x}+\frac{1}{c}\frac{\partial Q}{\partial t}+\varepsilon=0$$

If only first order terms are included in the error term this becomes:

[8.30] $$\frac{\partial Q}{\partial x}+\frac{1}{c}\frac{\partial Q}{\partial t}-\frac{\Delta x}{2}\left((1-2\alpha)+(2\beta-1)\tau\right)\frac{\partial^2 Q}{\partial x^2}=0$$

or

[8.31] $$\frac{\partial Q}{\partial x}+\frac{1}{c}\frac{\partial Q}{\partial t}=D\frac{\partial^2 Q}{\partial x^2}$$

Equation [8.31] is in the form of a diffusion equation where $\boldsymbol{D}$ is the coefficient of diffusion. In order to relate $\boldsymbol{D}$ to the physical characteristics of the channel an alternate diffusion equation can be developed using the continuity equation [8.20] with a simplified form of momentum equation in which the convective and temporal accelerative terms are assumed to be negligible, i.e.

[8.32] $$\frac{\partial h}{\partial x}=S_f$$

where h = water surface elevation

Sf = friction gradient

This can be developed (Smith(1980) see References) to yield the diffusion equation of [8.33].

[8.33] $$\frac{\partial Q}{\partial x}+\frac{1}{c}\frac{\partial Q}{\partial t}=\left(\frac{K^3}{2Q^2\, dK/dh}\right)\frac{\partial^2 Q}{\partial x^2}$$

where K = channel conveyance defined by $Sf = Q^2/K^2$

Now from the definition of the conveyance K the total derivative is obtained as:

[8.34] $$\frac{dK}{dh} = \frac{dQ}{dh}\frac{1}{S_f^{1/2}}$$

Comparison of the terms in equations [8.31] and [8.33] provides a means of evaluating the diffusion coefficient $\boldsymbol{D}$ and thus the weighting coefficients α and β in terms of the hydraulic characteristics of the channel. Thus:

[8.35] $$D = \frac{Q}{2S_f \, dQ/dh} = \frac{\Delta x}{2}\left((1-2\alpha)+(2\beta-1)\tau\right)$$

[8.36] $$(1-2\alpha)+(2\beta-1)\tau = \frac{Q}{h_f \, dQ/dh}$$

where the friction head loss over the reach being considered is given by

$$h_f = S_f \Delta x \qquad .$$

Equation [8.36] can be simplified by assuming an initial value of $\beta = 0.5$ so that:-

[8.37] $$\alpha = 1 - \frac{Q}{2h_f \, dQ/dh}$$

If equation [8.37] yields a value for α which is less than zero, MIDUSS sets $\alpha = 0.0$ and solves for β from [8.36]. This value will generally be in the range $0.5 \le \beta \le 1.0$. The generalized Muskingum coefficients of equation [8.27] can then be evaluated and a solution obtained for *Q4.*

In MIDUSS, evaluation of the diffusion coefficient differs depending on whether the conduit is a pipe or channel. The Manning equation [8.12] can be differentiated to obtain an expression for *dQ/dy* as follows:

[8.38] $$\frac{dQ}{dy} = \left(\frac{M}{n}S_0^{1/2}\right)\left(\frac{5}{3}TR^{2/3} - \frac{2}{3}R^{5/3}\frac{dP}{dy}\right)$$

Substituting in the 2nd part of [8.35] yields an expression for D which can be evaluated in terms of the channel cross-section parameters and the channel gradient, thus:

[8.39] $$D = \frac{1.5A}{S_0\left(5T - 2R\dfrac{dP}{dy}\right)}$$

For pipes, an alternative procedure is used in which a fitted polynomial represents the ratio *Q/(dQ/dyr)* as a function of the proportional discharge *Qr* which is the ratio of actual discharge to full pipe capacity. With very acceptable accuracy this can be represented as follows:

[8.40] $$\frac{Q}{dQ/dy_r} = ((1.06Q_r - 1.16)Q_r + 0.8336)Q_r + 0.034743$$

from which the diffusion coefficient D can be found using equation [8.35].

8.3.2 Criteria for Numerical Stability in Flood Routing

Once values have been determined for the weighting coefficients α and β it is possible to carry out a check on the numerical stability of the process. This involves the calculation of limiting values for the grid dimensions Δx and Δt.

Biesenthal (1975) (see References) obtained the following condition for numerical stability.

[8.41] $$\frac{\alpha}{\beta} \le \left(\frac{c\,\Delta t}{\Delta x} \right) \le \frac{1-\alpha}{1-\beta}$$

In general, this means that the nucleus of Figure 8.4 must lie above and to the right of a diagonal through the centre of the space-time element which has a slope of *-(1/c).*

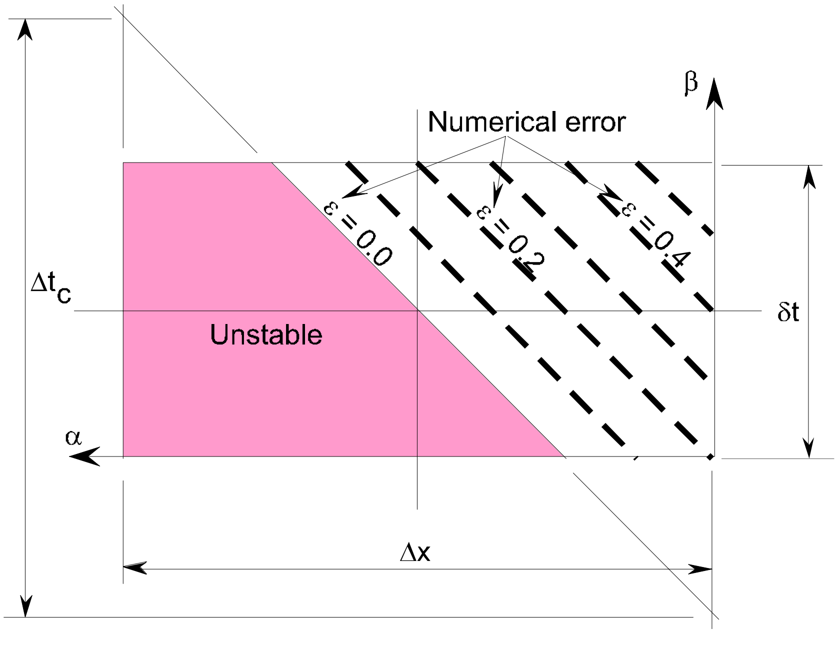


Figure 8.5 - Stability and numerical error characteristics of a space-time element.

Figure 8.5 shows a typical case in which the time step δt is approximately half of the Courant time step given by $\Delta tc = \Delta x/c$. The heavy dashed lines parallel to the main diagonal form a family of lines each of which comprises the locus of points for which the nucleus will generate a numerical error ε of a specific value. It is significant that the same numerical attenuation can be produced by a set of (α, β) coordinates and MIDUSS makes use of this feature.

The shaded area of Figure 8.5 indicates locations of the nucleus which are numerically unstable. Because the values of α and β are constrained to provide a required numerical error, the criterion for stability given by equation [8.38] must be satisfied by manipulating either the routing timestep δt or the reach length Δx. The greater the coarseness of the space-time grid the greater the chance of numerical instability. Thus we need to determine upper limits for both Δx and the routing time-step δt

Equation [8.35] shows a relationship between the diffusion coefficient D and the weighting coefficients α and β. If we assume initially that $\beta = 0.5$ this reduces to:

$$[8.42] \qquad 2\alpha = 1 - \frac{2D}{\Delta x}$$

Similarly if $\beta = 0.5$ the inequality [8.41] becomes:

$$[8.43] \qquad 2\alpha \le \frac{c\Delta t}{\Delta x} \le 2(1-\alpha)$$

Taking the 1st and 2nd parts of [8.43] and substituting for α from [8.42] we obtain:

$$[8.44] \qquad 1 - \frac{2D}{\Delta x} \le \frac{c\Delta t}{\Delta x}$$

or

$$\Delta x \le 2\,D + c\,\Delta t$$

To get an upper bound for the routing time-step we use the 2nd and 3rd parts of [8.43] and obtain:

$$[8.45] \qquad \frac{c\Delta t}{\Delta x} \le 2 - 1 + \frac{2D}{\Delta x}$$

or

$$\Delta t \le \frac{\Delta x + 2\,D}{c}$$

MIDUSS uses the limiting criteria of [8.44] and [8.45] to divide either the reach length L or the hydrograph time-step *?t* into sub-multiples which satisfy the conditions for stability.

In the case of very long reaches the time step remains unchanged but routing is carried out over two or more subreaches. The final outflow hydrograph is the only one presented to the user.

In the case of very short reaches only a single reach length is used but the routing time-step is set at $\delta t = \Delta t/n \ (n = 2,3,4...)$. Only flow values at intervals of Δt are presented in the final outflow hydrograph.

8.4 Theory of Reservoir Routing

Reservoir routing involves the application of the continuity equation to a storage facility in which the storage volume for a particular geometry is a dependant only on the outflow. This can be viewed as a special case of the more general kinematic wave routing procedure described in section 8.3 *Kinematic Flood Routing* in which the weighting coefficients are assigned values of $\alpha = 0.0$ and $\beta = 0.5$ (see Figure 8.5).

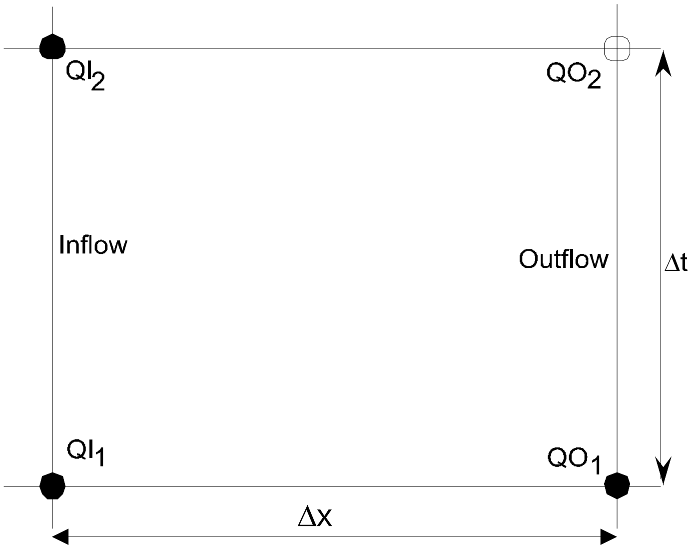


Figure 8.6 A space-time element for reservoir routing.

With reference to Figure 8.6 the continuity equation can be averaged over a time-step Δt as follows.

$$\frac{QI_1 + QI_2}{2} = \frac{QO_1 + QO_2}{2} + \frac{S_2 - S_1}{\Delta t} \tag{8.46}$$

where QI = time series of inflow values

QO = time series of outflow values

S = storage volume

$1,2$ = subscripts corresponding to times t and $t+\Delta t$ respectively.

Equation [8.46] can be expanded as follows to yield an indirect solution for the outflow $QO2$.

$$QI_1 + QI_2 = \left(\frac{2S_2}{\Delta t} + QO_2\right) - \left(\frac{2S_1}{\Delta t} + QO_1\right) + 2 \quad QO_1 \tag{8.47}$$

or

[8.48] $$f(QO_2) = f(QO_1) + QI_1 + QI_2 - 2QO_1$$

where

$$f(QO) = \frac{2S}{\Delta t} + QO$$

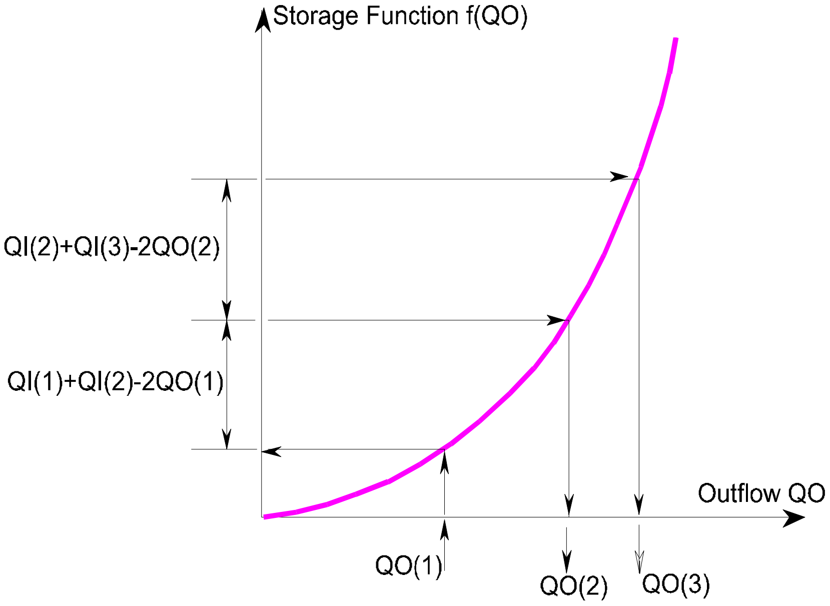


Figure 8.7 Graphical illustration of equation [8.48]

The application of equation [8.48] is illustrated graphically in Figure 8.7. Starting with some initial, known value of $QO1 = QI1$ the corresponding value of $f(QO1)$ is found by interpolation or otherwise. The inflow hydrograph $QI(t)$ provides an upstream boundary condition from which $QI2$ can be found. Then from equation [8.48] a value is obtained for $f(QO2)$ and finally by back interpolation $QO2$ is calculated and the process continues for other time increments.

8.4.1 Estimating the Required Pond Storage

At the start of the **Pond** command MIDUSS estimates the required volume by making the assumption that the reservoir is linear. This means that the storage volume S is a linear function of the outflow QO and defined in terms of a lag coefficient K. Thus:

[8.49] $$S = KQO$$

For this special case the storage terms can be eliminated from equation [8.47] and an explicit solution is obtained for $QO2$ as follows.

[8.50] $$QI_1 + QI_2 = \left(\frac{2K}{\Delta t} + 1\right)QO_2 - \left(\frac{2K}{\Delta t} + 1\right)QO_1 + 2QO_1$$

[8.51] $$QO_2 = QO_1 + \frac{QI_1 + QI_2 - 2QO_1}{\left(\frac{2K}{\Delta t} + 1\right)}$$

With reference to Figure 8.8, MIDUSS initially assumes that the lag coefficient *K2* has a value of *0.2tb* (where *tb* is the time base of the inflow hydrograph) and the corresponding peak outflow *QO2* is obtained by applying equation [8.51] to the inflow hydrograph. Arbitrary assumptions are also made for the lag which will attenuate the peak outflow to 1/100th of the peak inflow. Using a secant method as illustrated in Figure 8.8 the estimate of *K* is successively improved. The relation is shown in equation [8.52].

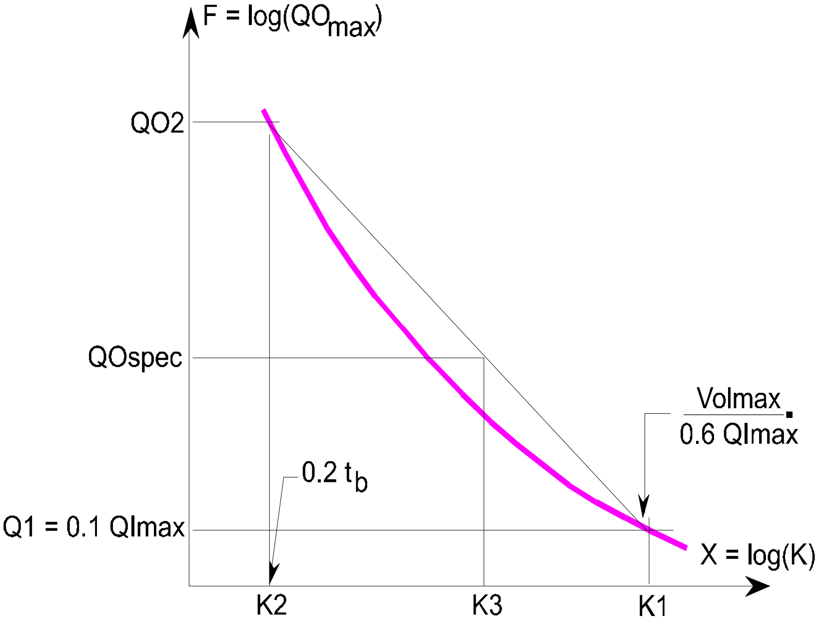


Figure 8.8 **An iterative solution for K.**

[8.52] $$X_3 = X_2 - \left(\frac{F_2 - F_{spec}}{F_2 - F_1}\right)(X_2 - X_1)$$

where *X* = log(*K*)

F = log(*QOmax)*

This procedure converges very rapidly on the necessary lag *Kopt* to produce the desired peak outflow *QOspec*. The corresponding storage volume is given as *Sopt = Kopt* x *QOspec.*

8.4.2 Numerical Stability in Reservoir Routing

The storage indication method is traditionally assumed to be inherently stable. However, this complacency is not justified in situations where the hydrograph is sharply peaked and the discharge-volume functions are poorly conditioned or exhibit pronounced discontinuities or points of contraflexure. In such circumstances, use of an arbitrary time-step can result in a computed outflow peak that is larger than the peak inflow - a condition that is physically impossible.

The search for a criterion to avoid this anomaly can start with the assumption that:

[8.54] $$QO_2 = QO_{\max} = QI_2 - \varepsilon \quad \text{where } \varepsilon > 0$$

Now by substituting [8.54], equation [8.49] can be written as:

[8.55] $$QI_1 + QI_2 = QO_1 + QI_2 - \varepsilon + \frac{2}{\Delta t}\left(S_2 - S_1\right)$$

or

[8.56] $$\left(QI_1 - QO_1\right) < \frac{2}{\Delta t}\left(S_2 - S_1\right)$$

This provides an upper limit on the routing time-step to be used which is shown in equation [8.57].

[8.57] $$\Delta t \le \left(\frac{2\left(S_2 - S_1\right)}{QI_1 - QO_1}\right)$$

The peak outflow must lie on the recession limb of the inflow hydrograph, so that:

[8.58] $$QI_1 = QI_2 - \Delta QI = QO_2 + \varepsilon - \Delta QI \approx QO_2$$

The routing time-step is then defined approximately as:

[8.59] $$\Delta t \le 2\left(\frac{\Delta S}{\Delta QO}\right)_{\min}$$

To implement this check, MIDUSS scans the storage-discharge function to determine the flattest part of the curve and uses this to determine an appropriate sub-multiple of the time-step to be used. Figure 8.9 shows a typical situation which can give rise to problems of this type.

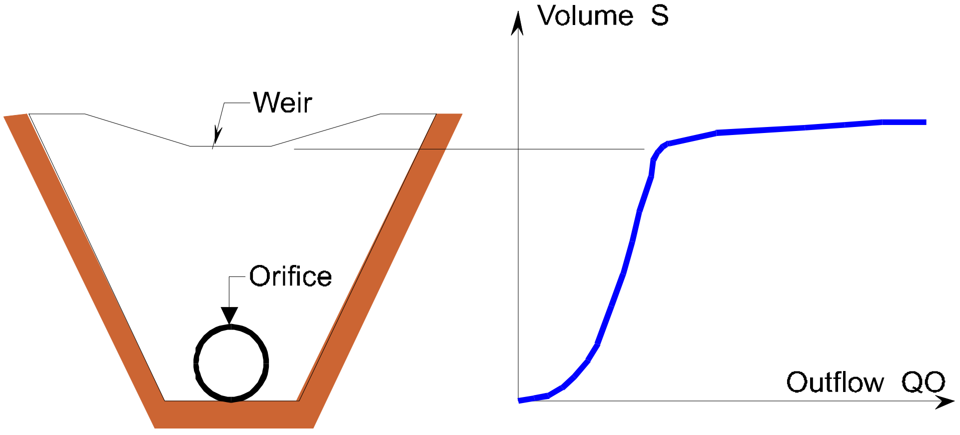


Figure 8.9 Storage-discharge function for a typical outlet control device.

8.4.3 Outflow Control Devices in Ponds

MIDUSS provides a number of tools to assist in the creation of the necessary table of stage, discharge and storage values which form the basis for evaluating the function *f(QO)* of [8.48]. This section describes how these flow estimates are made for two basic types of outflow control device.

- Orifices
- Weirs
- Outflow pipes
- Horizontal orifices

Figure 8.9 shows a simple but typical device which incorporates an orifice for low flow control and a weir for less frequent flood events. Up to 10 weirs and 10 orifices can be defined. In addition, MIDUSS has a special tool to assist in the design of Rooftop Flow Control and Storage for on-site control.

8.4.4 Orifice Flow for Pond Control

The stage discharge equation for the orifice is calculated for two cases which depend on the relative value of the specific energy H relative to the invert of the orifice and the diameter of the orifice D.

In Case 1, $H > D$ and the orifice is fully submerged.

[8.60] $$Q = C_c \frac{\pi}{4} D^2 \sqrt{2g(H - \frac{2}{3}D)}$$

where H = head relative to the invert of the orifice

D = orifice diameter

g = gravitational acceleration

Cc = coefficient of contraction

In Case 2, $H \le D$ and the orifice acts as a broad-crested weir of circular shape. The critical discharge can be approximated by equation [8.61]

[8.61] $$Q = f\left(\frac{H}{D}\right) C_c \sqrt{g} D^{5/2}$$

where

$$f\left(\frac{H}{D}\right) = 0.494\left(\frac{H}{D}\right)^{1.57} - 0.04\left(\frac{H}{D}\right)^{0.5}$$

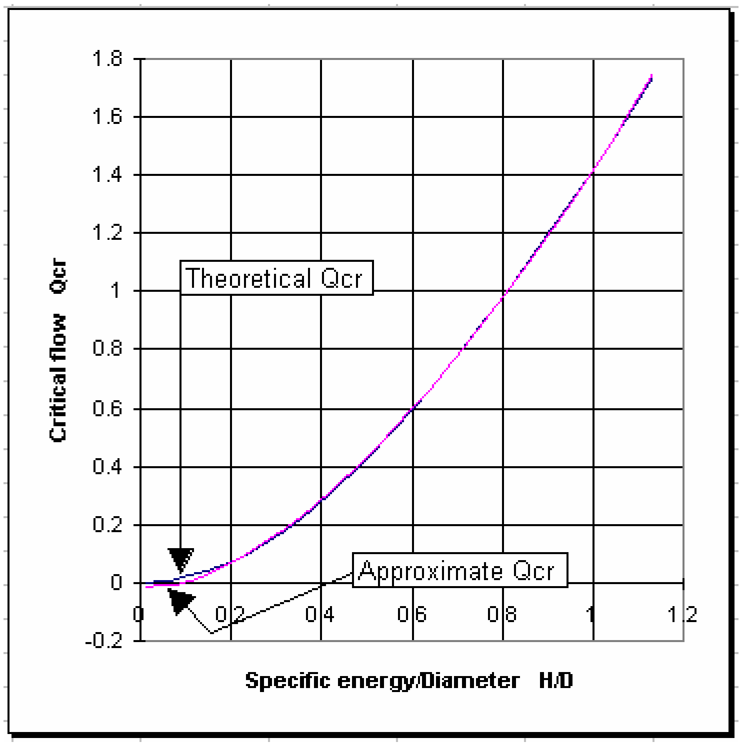


Figure 8.10 **Critical flow through a segment of a circle.**

As shown by the comparative plot of Figure 8-10, equation [8.61] is a very reasonable approximation to the critical discharge through a segment of a circle.

8.4.5 Weir Flow for Pond Control

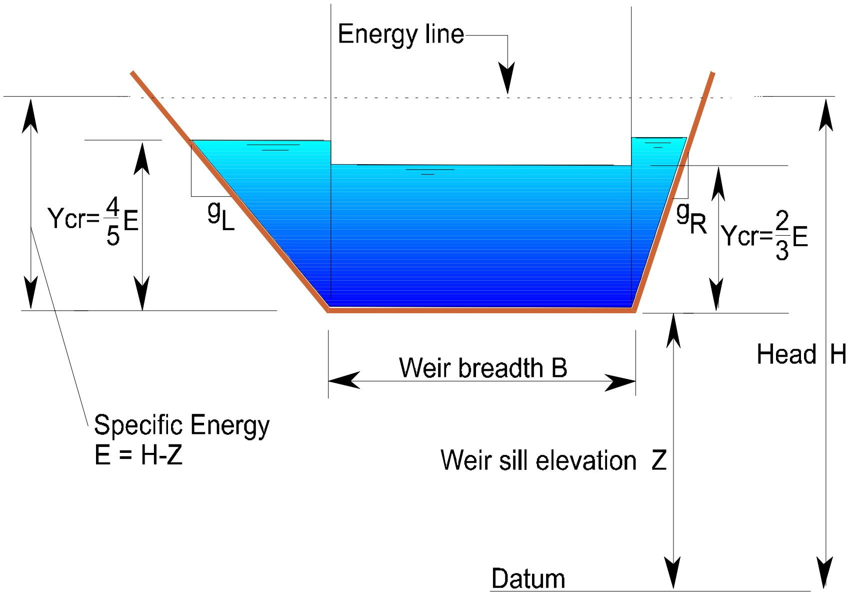


Figure 8.11 Definition sketch of a trapezoidal weir.

The weir control is assumed to have a general trapezoidal shape as illustrated above. The critical discharge is calculated for the central rectangular section and two triangular sections. For any value of head H greater than the weir sill elevation Z the critical discharge can be calculated using the general criterion for critical flow of equation [8.8]

$$\frac{Q^2T}{gA^3} = 1 \tag{8.8}$$

and calculating the cross-section properties A and T in terms of the parameters shown in Figure 8.11.

For a rectangular section:

$$Q_{cr} = B\sqrt{g}\,y_{cr}^{3/2} \tag{8.62}$$

where

$$y_{cr} = \frac{2}{3}(H - Z)$$

For a triangular section:

$$Q_{cr} = \sqrt{g/2}\,\frac{(S_L + S_R)}{2}\,y_{cr}^{5/2} \tag{8.63}$$

where

$$y_{cr} = \frac{4}{5}(H - Z)$$

8.4.6 Typical Storage Components for Detention Ponds

In addition to control flow estimation tools, MIDUSS provides a few methods for estimating the available volume in various standard storage facilities. These assist you in setting up the stage, discharge and storage values which form the basis for evaluating the function *f(QO)* of [8.48]. This section describes how these storage estimates are made for three basic types of storage facility.

(1) Rectangular ponds

(2) Super Pipes

(3) Wedge (or Inverted Cone) ponding

In addition, MIDUSS has a special tool to assist in the design of Rooftop Flow Control and Storage for on-site control.

8.4.7 Rectangular Pond Storage

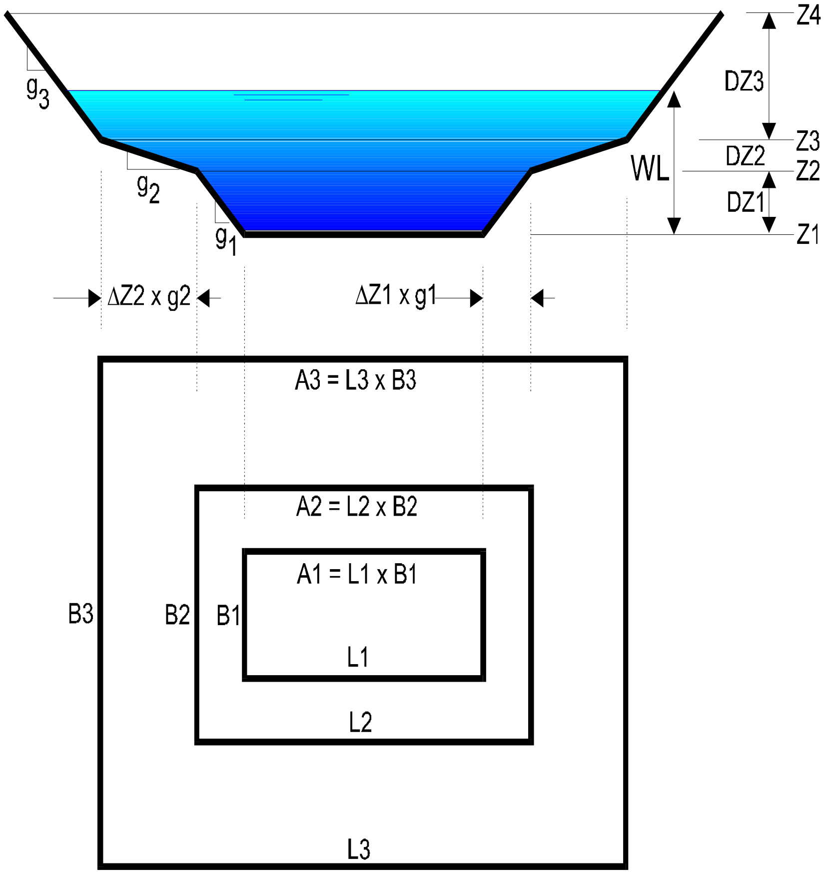


Figure 8.12 Schematic of a 3-stage rectangular pond

Detention ponds are usually constructed with side slopes which are dictated by consideration of maintenance (e.g. grass cutting) and safety. It is common for the side slope to be different at different water surface elevations. If the pond has a permanent storage component (e.g. for quality) it may be desirable to maintain a flat slope of 4:1 or 5:1 for 3m/10ft both below and above the permanent water surface elevation. Even if the pond is a "dry" pond it may be necessary to have a flatter slope at higher depths in order to get a suitably nonlinear stage-storage curve.

Figure 8.12 shows an idealized pond with three stages. The shape in plan is approximated by a series of rectangles corresponding to different elevations and which have an aspect ratio *L/B* which reduces with increasing height.

In practice it is most unlikely that the pond geometry correspond closely to this idealized shape but the rectangular pond method provides a useful design tool to estimate the general dimensions (volumes, land area etc.) required to achieve a required level of flow peak reduction.

The volume is calculated using Simpson's rule so that;

[8.64] $$V = \frac{H}{6}\left(A_1 + 4A_m + A_2\right)$$

where

$$\begin{aligned} A_1 &= B_1 L_1 \\ A_m &= \left(B_1 + g_1 H\right)\left(L_1 + g_1 H\right) \\ A_2 &= \left(B_1 + 2g_1 H\right)\left(L_1 + 2g_1 H\right) \\ H &= Min(WL, Z_2) - Z_1 \end{aligned}$$

MIDUSS provides an approximate estimate for the base area $A1$. This is calculated from the estimate of required storage volume and assumes that only a single stage is used and that the depth is 2/3 of the depth range specified, the base aspect ratio is 2:1 and the side slope is 4H:1V.

8.4.8 Super-Pipes for Pond Storage

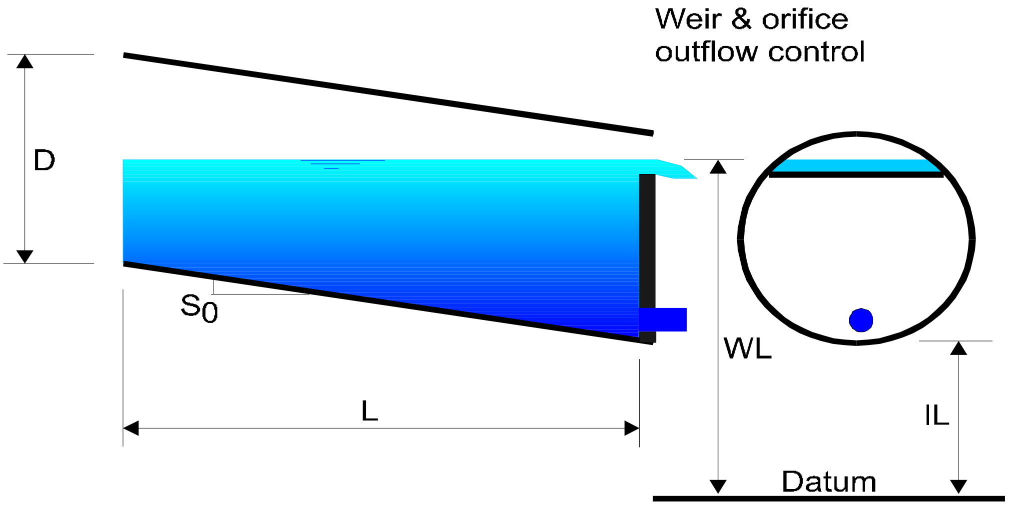


Figure 8.13 - Schematic of Super Pipe Storage

Figure 8.13 shows a typical arrangement of a single super-pipe with a simple outflow control device installed at the downstream end. The control can be installed either in the pipe barrel or in a manhole structure. The latter is convenient if more than one super pipe converges at a junction node.

You should remember to avoid using too steep a gradient as this can seriously limit the available storage volume since the water surface is likely to be nearly horizontal.

In MIDUSS the volume is obtained by calculating the cross-section of the storage at 21 equally spaced sections along the length L and then using Simpson's Rule. The section area is given as a function of the relative depth y/D from the following equations.

$$A = \frac{D^2}{8}\left(\phi - \sin\phi\right) \tag{8.3}$$

in which ϕ is obtained by [8.11]

$$\phi = 2\tan^{-1}\left(\frac{T/2}{\left(D/2 - y\right)}\right) = 2\tan^{-1}\left(\frac{T}{D - 2y}\right) \tag{8.11}$$

where

$$T = 2\sqrt{\left(Dy - y^2\right)} \tag{8.10}$$

The volume is then obtained as:

$$V = \frac{L}{20}\left(A_1 + 4A_2 + 2A_3 + \cdots + 2A_{19} + 4A_{20} + A_{21}\right) \tag{8.65}$$

MIDUSS provides an initial default length for a single super pipe assuming that (1) the diameter is approximately half the depth range, (2) the slope is zero (3) the pipe is full and the volume is equal to the estimated required storage.

8.4.9 Wedges (or inverted Cones) for Pond Storage

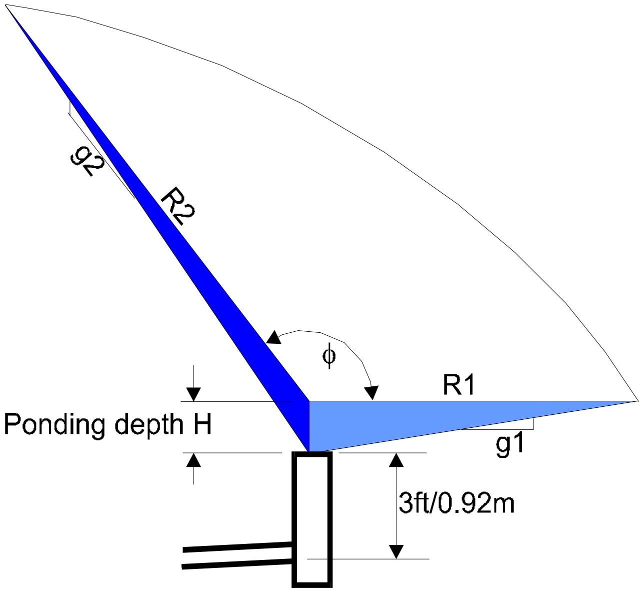


Figure 8.14 Schematic of wedge storage

To assist in estimating the available storage on parking lots, MIDUSS provides a wedge storage procedure that calculates the volume of a sector of a flat, inverted cone as illustrated in Figure 8.14. The angle subtended by the segment is defined as an angle ϕ (in radians) for generality. In practice, this will often be 90 degrees with four such segments describing the storage around a catchbasin draining the parking lot with grades *g1* and *g2* mutually at right angles.

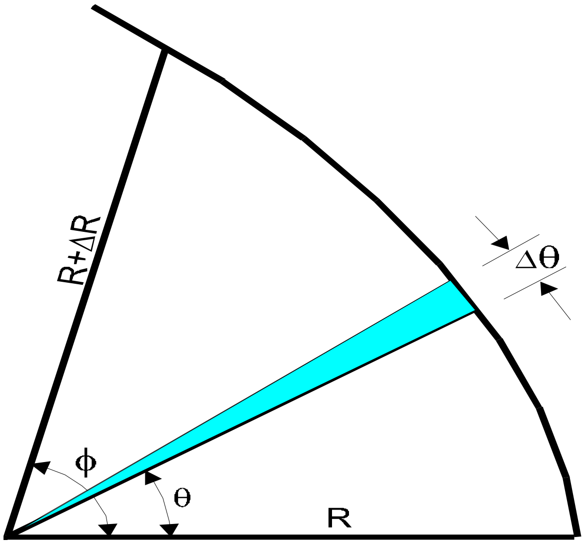


Figure 8.15 Calculation of surface area of a segment of inverted, ovoid cone.

The radius R and grade g are assumed to vary linearly with the angle as shown in Figure 8.15. Then a small element of the surface area is described as:

$$[8.66] \qquad dA = \frac{1}{2}\left(R_1 + \frac{\Delta R}{\phi}\theta\right)^2 d\theta$$

:

Integrating between the limits 0 and ϕ gives the surface area as

$$[8.67] \qquad A = \frac{\phi}{6}\left(R_1^2 + R_1R_2 + R_2^2\right)$$

The volume V is then calculated as:

$$[8.68] \qquad V = A\frac{H}{3} = \frac{\phi}{18}\left(g_1^2 + g_1g_2 + g_2^2\right)H^3$$

MIDUSS assumes that the invert of the tail pipe or the Inflow Control Device (ICD) in the catch basin is approximately 3 ft (0.92 m) below the rim elevation and that the maximum depth of ponding will probably be less than 1 ft (0.3m) above rim elevation as illustrated in Figure 8.14. To provide an initial estimate for design purposes MIDUSS assumes that the last defined impervious area has a catch basin density of 1 per 2500 sq.m or 2989 sq.yd. It is further assumed that each catch basin has a drainage area with orthogonal grades in a ratio of 2:1 (e.g. 40H:1V in one direction and 80H:1V at right angles). Based on a depth of 1 ft (0.3m) above rim elevation, the necessary grades to provide the estimated required volume are calculated and displayed together with the total number of elliptical quadrants (four such quadrants per catch basin).

In setting up parking lot storage the depth range should be slightly more than 4 ft (1.22m) to comply with the assumptions made above.

8.4.10 Rooftop Flow Control for Pond Storage

For developments involving large commercial buildings with flat roofs, on-site storage can be provided by installing roof drain controls. Typically these devices contain one or more "notches" which take the form of a linear proportional weir in which discharge is directly proportional to the head or depth of storage, for example 24 litres per minute per 25 mm of head or 6 US gallons per minute per inch of head. The actual value is defined along with other relevant parameters.

If the roof is dead level then the volume of storage is calculated simply as roof area times head where the roof area available for storage is smaller (e.g. 75%) than the building footprint to allow for service structures (access, elevator, HVAC) on the roof.

When a finite grade is used to promote drainage the calculation of available storage depends on whether the head H is less than or greater than the fall or difference in elevation between ridge and valley in the roof profile.

$$[8.69] \qquad V = A\left(\frac{H}{\Delta Z}\right)^2\frac{H}{3} \quad \text{for} \quad H < \Delta Z$$

$$[8.70] \qquad V = A\left(H - \frac{2\Delta Z}{3}\right) \quad \text{for} \quad H \geq \Delta Z$$

where

$$\Delta Z = L\,S_0$$

and L = flow length from ridge to drain

$S0$ = roof grade

8.5 Theory of Exfiltration Trench Design

An exfiltration trench is a facility that encourages the return of runoff to the ground water. It may be a very simple "soak-away" and comprise only a trench filled with clear stone (i.e. single sized gravel) into which runoff is directed. A more complex facility might be incorporated in-line with a conventional storm sewer and include one or more perforated pipes along the length of the trench to provide more uniform distribution of the inflow over the length of the trench. It is this latter type of facility which is described in the MIDUSS **Trench** command.

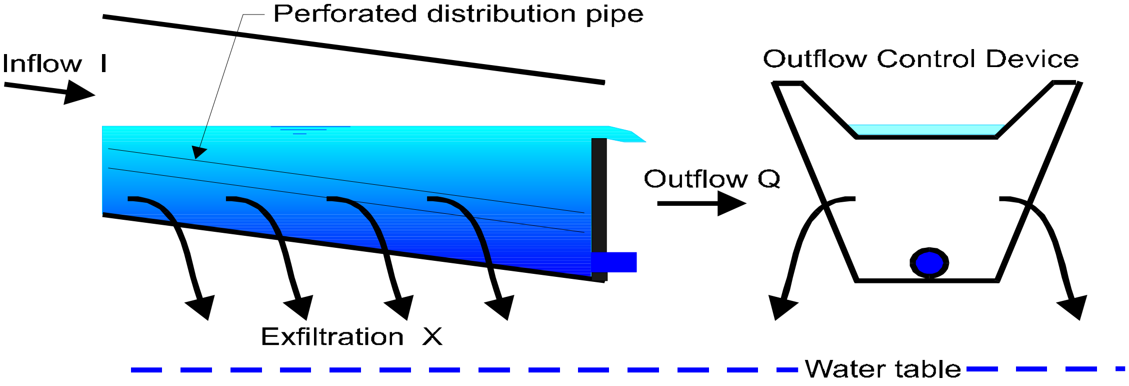


Figure 8.16 A typical exfiltration trench.

Figure 8.16 illustrates a typical arrangement of an exfiltration trench which splits the inflow hydrograph into two components. One fraction is transmitted downstream as an outflow hydrograph that is attenuated by the storage within the voids of the clear stone fill. The balance of the flow is transmitted to the ground water through the pervious walls of the trench. The Trench form has an option to include or exclude the base of the trench in estimating the area contributing to exfiltration.

It is usual to provide some form of outflow control device at the downstream end of the trench to force the free surface in the trench to rise. This causes (1) the volume of voids available for storage to be increased and (2) the surface area along the walls of the trench is increased to allow increased exfiltration. Figure 8.16 shows a typical outflow control device with a small orifice at or near the downstream invert of the trench to allow drainage of accumulated flow in the trench plus an overflow weir to produce high water levels during the maximum inflow rate. The trench may be thought of as a variation of the "super-pipe" facility with a permeable pipe wall.

Analysis of the facility is based on a form of the continuity equation which takes account of the outflow control, the rate of exfiltration and the rate of change of storage within the trench. Thus

Inflow = Outflow + Exfiltration + Rate of change of Storage

or

$$\frac{I_1 + I_2}{2} = \frac{Q_1 + Q_2}{2} + \frac{X_1 + X_2}{2} + \frac{V_2 - V_1}{\Delta t} \qquad [8.71]$$

where I = Inflow rate

Q = Outflow rate

X = Exfiltration rate

V = Volume stored

and the subscripts 1 and 2 define values at times t and $(t+\Delta t)$ respectively.

Equation [8.71] can be expanded as:

$$[8.72] \qquad I_1 + I_2 = \left(\frac{2V_2}{\Delta t} + Q_2 + X_2 \right) - \left(\frac{2V_1}{\Delta t} + Q_1 + X_1 \right) + 2Q_1 + 2X_1$$

or

$$[8.73] \qquad I_1 + I_2 = f\left(V_2, Q_2, X_2\right) - f\left(V_1, Q_1, X_1\right) + 2Q_1 + 2X_1$$

For any specified outflow control device, the water surface elevation in the trench is dependent on the outflow Q. Both storage volume V and exfiltration X are therefore dependent on Q and a solution for the unknown outflow at time $(t+\Delta t)$ can be obtained from:

$$[8.74] \qquad f\left(Q_2\right) = f\left(Q_1\right) - 2Q_1 - 2X_1 + I_1 + I_2$$

The method is similar to the graphical solution described in Figure 8.7 *Graphical illustration of equation [8.48]* in topic Theory of Reservoir Routing. One difference is that it is convenient to construct curves (or tables) of both *f(V,Q,X)* and X as functions of the water surface elevation. In order to do this we must first provide a method of predicting the rate of exfiltration from the trench.

8.5.1 Trench Exfiltration Rate

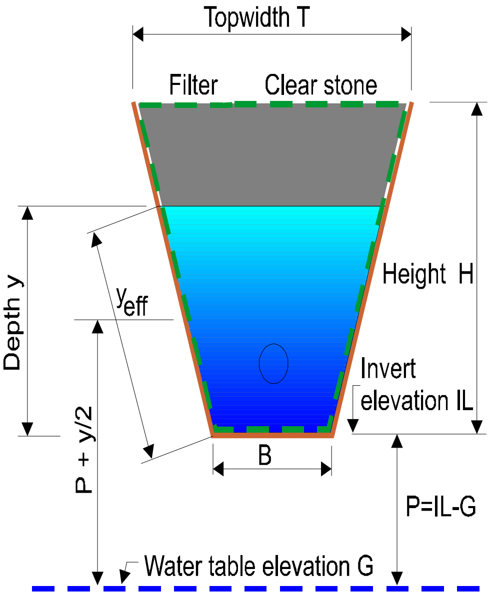


Figure 8.17 Exfiltration Trench Cross-section

Figure 8.17 shows the cross-section assumed in MIDUSS. The shape is a trapezium of height H and top width T tapering symmetrically to a bottom width B. The water table is assumed to be horizontal and located at a depth P=(IL-G) below the downstream invert level of the trench. If the depth of water in the trench voids is y the wetted surface of the trench wall has a length αy where α is given by:

$$[8.75] \qquad \alpha = \sqrt{1 + \left(\frac{T-B}{2H}\right)^2}$$

Flow through the porous soil is assumed to be laminar and can be estimated using Darcy's Law

$$[8.76] \qquad \frac{Q}{A} = q = KS_f$$

where K = hydraulic conductivity of the soil

Sf = friction gradient

Q/A = volumetric flux.

Note that the volumetric flux is much smaller than the actual velocity through the voids since only a fraction of area A is available for flow.

The average driving head between the water in the trench and the water table is P + y/2 and the path length is P so that the available gradient is given by [8.77].

[8.77] $$S_f = \frac{IL - G + y/2}{IL - G} = 1 + \frac{y/2}{IL - G}$$

The exfiltration flow through a unit length of trench can then be estimated as:

[8.78] $$dX = (2\alpha y + \beta B) K S_f$$

where β = 1 or 0 depending on whether the 'Include base width' check box is checked or unchecked. Checked is the default condition.

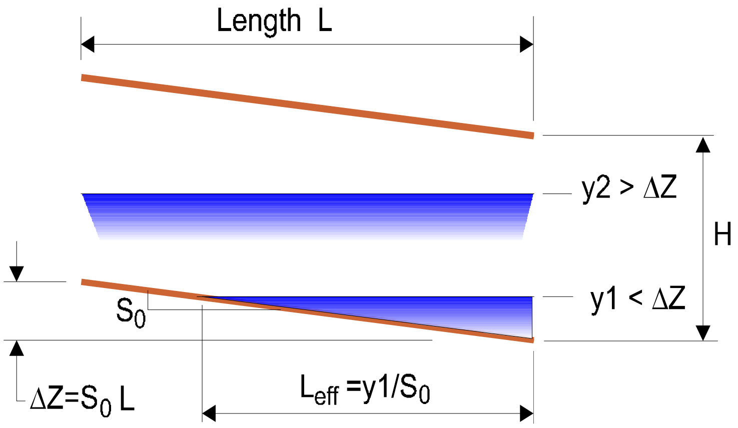


Figure 8.18 **Idealized Longitudinal section on an Exfiltration Trench.**

If the trench invert has a finite slope it is possible that for low flows which can be transmitted by the orifice in the outflow control device, the horizontal free surface does not extend over the full length of the trench. Figure 8.18 shows this situation. Even if the downstream depth is greater than the invert drop ΔZ the available surface for exfiltration must be corrected to allow for the reduced depth at the upstream end. This assumes that the hydraulic gradient along the trench is negligible and that the surface is essentially horizontal. The available wall surface through which exfiltration can occur is therefore given by:[8.79] and [8.80].

[8.79] $$A_X = \alpha\left(y - \frac{\Delta Z}{2}\right)L \quad \text{for} \quad y \geq \Delta Z$$

[8.80] $$A_X = \alpha \frac{y^2}{2S_f} \quad \text{for} \quad y < \Delta Z$$

8.5.2 Estimating the Required Trench Volume

When the Trench command is invoked MIDUSS tries to estimate the required trench volume (i.e. voids plus stone) which is required to achieve the currently defined target peak outflow. The process is similar to that described in the topic Theory of Reservoir Routing; Estimating the

Required Pond Storage. However, an additional level of iteration is required as for each estimate of storage volume the corresponding exfiltration must be computed and the target outflow reduced by this amount.

As with the Pond procedure, the iteration uses the secant method to solve a relationship between Q and K to yield the required value of Q and thus estimate the storage from the corresponding lag K.

The algorithm is summarized as follows.

1. Assume maximum exfiltration rate Xmax = 0
2. Set desired Qout = TargetQout - Xmax
3. Initialize values of K and Q for two points on the curve, i.e.

   K1 = Hydrograph Volume/(0.6*Imax)

   K2 = 0.2 Inflow hydrograph timebase

   Q1 = 0.1 Imax
4. Route inflow through a linear reservoir of lag K2 to get maximum outflow Q2
5. Interpolate between points (K1,Q1) and (K2,Q2) to get K3 for required Qout
6. For next iteration set K1 = K2

   K2 = K3

   Q1 = Q2
7. If change in Q2 > ε go to step 4.
8. Solution found for Q2. Estimate storage S = K2.Q2 and convert to trench volume.
9. From trench volume estimate maximum water level Wlmax.
10. For WL calculate exfiltration Xmax.
11. For 5 iterations go to step 2.

Because of the many other quantities which can affect the routing operation the estimate is only an approximate guide and trial and error is normally required.

Chapter 9 Displaying your Results

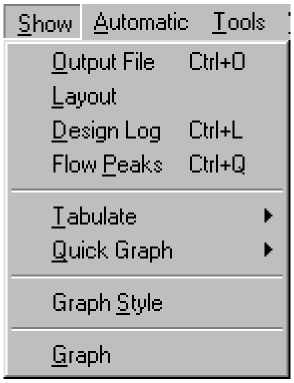


Figure 9-1 – The Show menu lets you display results in various ways

This chapter describes the commands available to show you the results of your MIDUSS session. As indicated in the menu shown in Figure 9-1, the main topics are as follows:

Output File	Show the current content of the Output File
Layout	Automatically plots the network elements to help you visualize the drainage system you are designing.
Design Log	Show the current contents of the Design Log
Flow Peaks	Show the summary of Peak Flows
Tabulate	Display a table of Hyetograph or Hydrograph values
Quick Graph	Display one or more graphs of a Hyetograph or Hydrograph
Graph Styles	Review or alter currently selected Colors, Patterns and Line properties
Graph	Draw and store a customized graph containing hydrographs and/or hyetographs

9.1 The Show Menu

9.1.1 Showing the Output File

Each use of this command causes an instance of a standard Windows editor (Notepad or WordPad) to be created with the current Output file as the subject of the command line. If the current output is less than 50,000 bytes in size MIDUSS tries to use the Microsoft Notepad text editor. If the output file is larger than 50,000 bytes the Microsoft WordPad editor is used.

After reviewing the output file you should close the editor explicitly either by using the File/Exit command in the Notepad or WordPad menu or by clicking on the Close Window icon **[x]** in the top-right corner of the window. If you simply click in the MIDUSS window, the editor window will disappear behind the MIDUSS window and the next time you use the Show/Output file command you will create a second instance of the editor.

9.1.2 Showing the Layout

This command provides a visual display of the elements added to the drainage network. The procedure is carried out automatically in MIDUSS and Catchments, Pipes, Channels and other stormwater control devices are drawn on the Layout as they are designed.

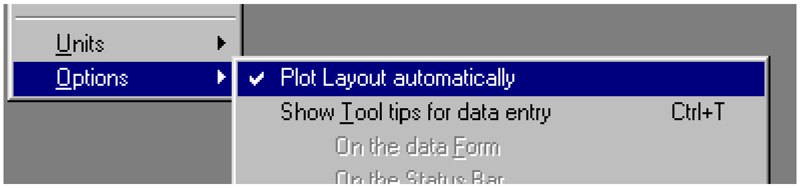
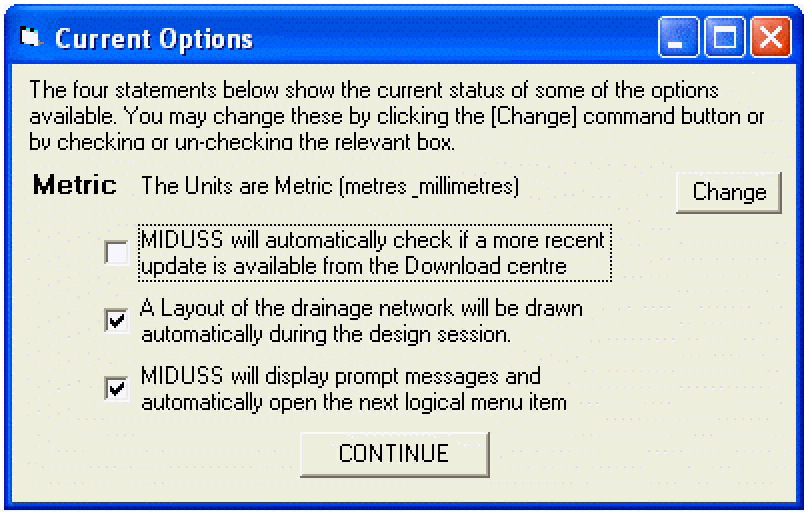


Figure 9-2 – Options shows the layout status

You can enquire about the status of the Plot Layout command by looking at the top of the Options list as displayed in Figure 9-2. In this case it is checked indicating that the feature is ON.

When MIDUSS starts you will be presented with a window as displayed in Figure 9-3 below.



The four statements below show the current status of some of the options available. You may change these by clicking the [Change] command button or by checking or un-checking the relevant box.

Metric The Units are Metric (metres _millimetres)

☐ MIDUSS will automatically check if a more recent update is available from the Download centre

☑ A Layout of the drainage network will be drawn automatically during the design session.

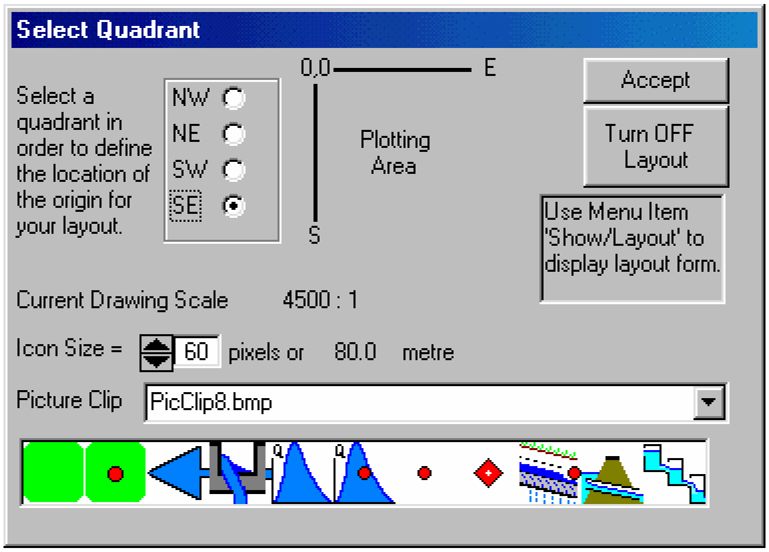
☑ MIDUSS will display prompt messages and automatically open the next logical menu item

Figure 9-3 – Layout is one of the important options at startup.

In this window you can tell MIDUSS that you want to have the drainage network automatically drawn as you design it.

You cannot stop and start the automatic drawing of the layout after you have started a MIDUSS session.

On the Layout form you can select and drag one or more objects to more closely represent the topology of the drainage network that you are designing. To assist in this customization of the Layout you can initially select whether the origin (i.e. where X = 0, Y = 0) implies plotting on a specific quadrant – NE, SE, NW or SW. For example, if the default South-East quadrant is selected the origin will be at the top-left corner of the drawing area. When MIDUSS is ready to start drawing the drainage layout you will be asked to select the origin and direction of the layout. Figure 9-4 below shows this dialog window.



Select a quadrant in order to define the location of the origin for your layout.

SE ●

Use Menu Item 'Show/Layout' to display layout form.

Current Drawing Scale 4500 : 1

Icon Size = 60 pixels or 80.0 metre

Picture Clip PicClip8.bmp

Figure 9-4 – Setting up the layout drawing direction.

Note in this window that you can Turn OFF the layout feature at this point. However, you will not be able to turn it back on again until a new MIDUSS session is started.

In this example the drainage elements will start plotting at the top left of the layout and grow the network in a general south-east plotting direction. This is a starting point for your layout since you can easily move the drainage elements around to match your real life network characteristics. You can change the drawing direction at any point by selecting the appropriate orientation button on the layout menu.

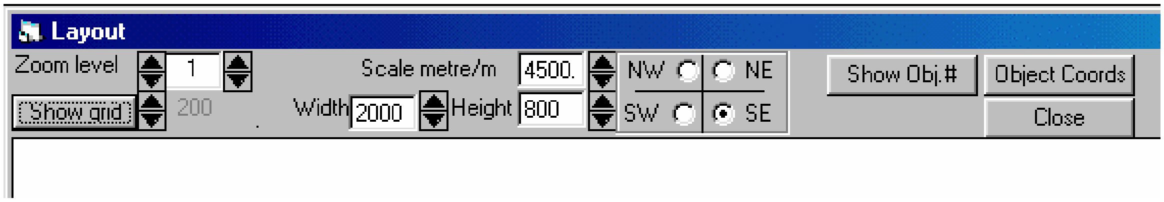
The form shows you the Current Drawing Scale and you can change this to a more convenient number once the layout menu appears.

The Icon size dialog allow you to scale the size of the drainage element icons to a size that you like. Beside this pixel size is the approximate equivalent real size scaling.

The Picture Clip dialog shows the BMP file used to hold the Icon gallery of drainage elements. Future MIDUSS features will allow you to select your own custom gallery of drainage element icons.

9.1.3 The Layout Menu

When the Layout window appears you will see the layout menu at the top. Fig 9-5 provides an example of this menu.



Zoom level 1

Scale metre/m 4500.

Width 2000

Height 800

● SE

Figure 9-5 – The layout menu

The Zoom level selection lets you zoom in and out of the plot to enlarge or shrink all the icons and connections. There are two spin buttons in the Zoom selector. The left spin button increases or decreased the zoom adjustment by 0.4 increments. The right zoom button provides a finer increment of 0.05 per click.

The Show grid button shown in Figure 9-6 will turn the grid lines on or off. The spin button will changes the grid spacing increment by 50 units at a time to a minimum of 50.


Zoom level 1

200

Figure 9-6 – The Zoom and Grid selection buttons

The Scale units will change depending on the type of units you selected at the beginning of MIDUSS. In the example in Figure 9-7 the scale is 7000 metre / m. For example, if the layout window on your computer screen window is 0.1 m wide (about 4 inches) wide then the layout is displaying 700 metres of the drainage network.

The Width and Height buttons change the layout drawing size in pixels. Figure 9-7 shows the layout to have dimension of 2000 metres wide and 800 metres high.

For both Scale and drawing dimensions you adjust the values using the spin buttons or you can manually enter a desired number.

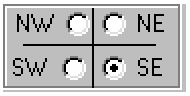

Scale metre/m 7000.

Width 2000

Height 1000

Figure 9-7 – Setting the layout scale and dimensions.

Figure 9-8 show you the currently selected drawing direction of the plot. In this example the drawing is in a South-East direction meaning the origin (0,0) is at the top left. You can change the drawing direction at any time and MIDUSS will immediately redraw the plot accordingly.


NW ○ ○ NE

SW ○ ● SE

Figure 9-8 – The drawing direction.

Fig 9-9 shows the [Show Obj #] and the [Object Coords] buttons in addition to the [Close] button.

The [Show Obj #] button will redraw the layout screen exactly as presented but will add object numbers to each of the elements that are displayed. These object numbers are assigned sequentially within MIDUSS and are not related to any node numbers that you may be assigning during the engineering design of the drainage system.

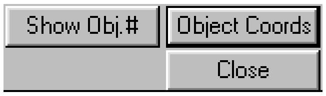
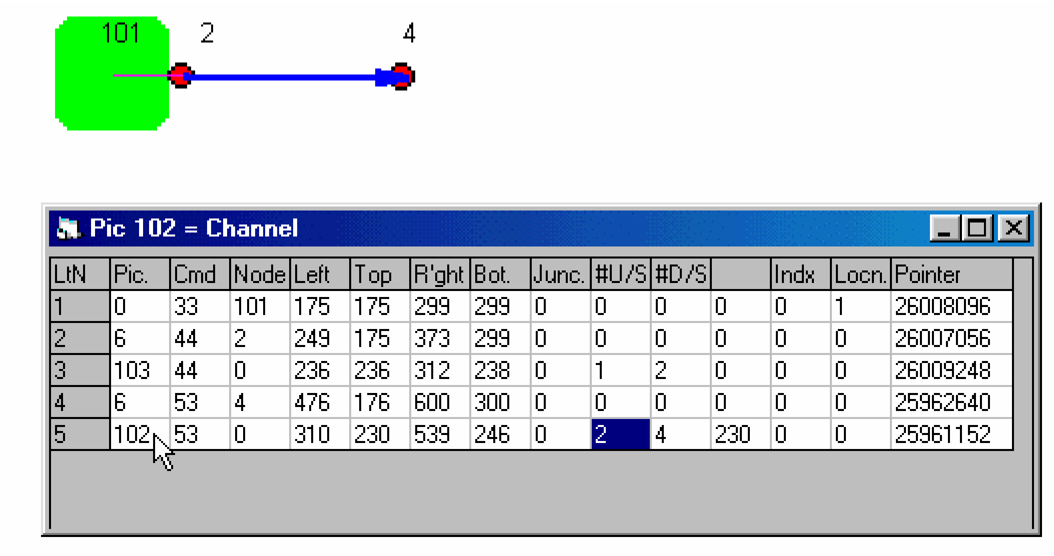


Figure 9-9 – Object coordinate buttons.

The [Object Coords] button presents a grid window at the bottom left of the screen which includes display and coordinate information for each of the objects on the layout. Fig 9-10 shows an example of this layout data table.

In this table the first column titled LtN is the actual object number. The mouse arrow pointing to object 5 in the Pic column will display what that object is known as in the blue banner of the window. In this case the pic 102 is a channel icon. On the row for object 5 (the channel) you can see in the #U/S and #D/S columns that the catchment is linked to object 2 upstream and object 4 downstream. Also, from this table you can see in the Cmd column that the numbers are identical to the command numbers used in the MIDUSS output files and are sequentially in the same order of operation.



Pic 102 = Channel

LtN	Pic.	Cmd	Node	Left	Top	R'ght	Bot.	Junc.	#U/S	#D/S		Indx	Locn.	Pointer
1	0	33	101	175	175	299	299	0	0	0	0	0	1	26008096
2	6	44	2	249	175	373	299	0	0	0	0	0	0	26007056
3	103	44	0	236	236	312	238	0	1	2	0	0	0	26009248
4	6	53	4	476	176	600	300	0	0	0	0	0	0	25962640
5	102	53	0	310	230	539	246	0	2	4	230	0	0	25961152

Figure 9-10 – Layout object details table.

The ability to view layout object information using the [Show Obj #] and the [Object Coords] buttons can help you if there are any questions or doubts about the accuracy of the way MIDUSS automatically linked the icons.

9.1.4 The Layout Popup Menu

When the Layout window is displayed on your computer screen you can click your mouse right button to open up a menu used just for interaction with the layout information.

Select mode – lets you move the icons and connections around to better match your real drainage network.

View mode – lets you hover over a layout element to reveal data about that element and its inflow and outflow data.

Print Area – use this to print the full layout or a specific rectangular section.

Select Printer – lets you setup your target computer for layout printing.

Background – with this feature you can import bitmap or vector-based graphics as a backdrop for your layout.

Save Picture – the layout and background can be exported to certain graphic file formats

9.1.4.1 The Layout Popup Menu - Select

If you right click your mouse while hovering over any part of the layout window the layout menu will appear. A check mark indicated in Fig 9-11 that Select mode is active.

Select mode lets you click on an icon or a group of icons and manipulate the size or location of the selection.

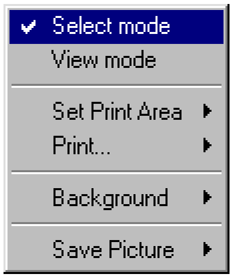


Figure 9-11 – The mouse is in Select mode.

Clicking on a single icon such as the diversion shown in Fig 9-12 below will present white boxes sometimes referred to as 'handles.' With your mouse pointer you can drag the handles to re-size the icon or you can click in the middle of the handle area and move the icon to a new location. As you move the icon all links connected to the icon will stretch and follow the icon movement.

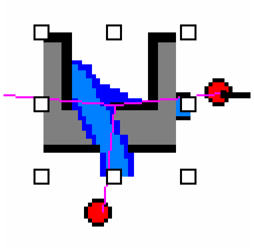


Figure 9-12 – The diversion icon has been selected.

In Select mode you can draw a rectangular area around many icons and a large number of gray handles will appear on the icons inside that rectangular space. You can then scale or move this group of icons just as you can with the single icon. Fig 9-13 below shows you a grouping of icons and connectors.

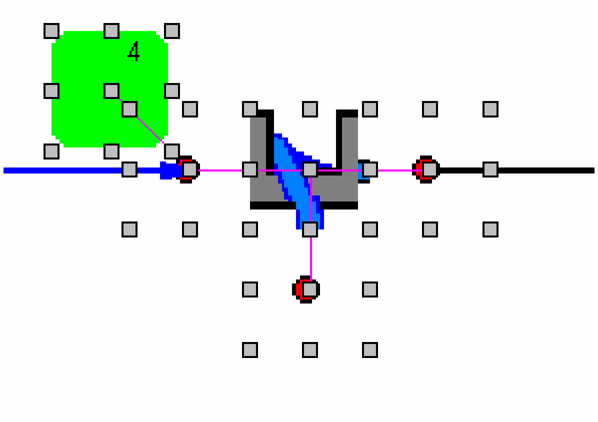


Figure 9-13 – Selecting a group of icons.

9.1.4.2 The Layout Popup Menu - View

The layout popup menu includes a View mode as shown in Figure 9-14. When checked it will display icon and connector information when you hover over a network element.

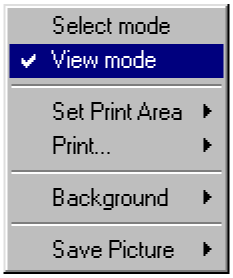
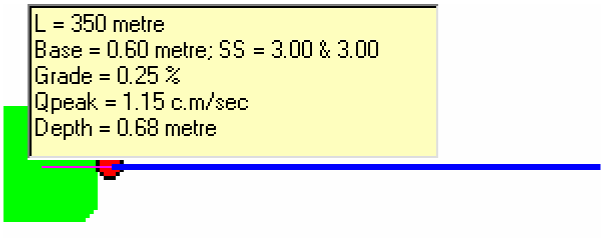


Figure 9-14 – Setting the mouse to use View mode.

In Fig 9-15 the mouse is hovering over a channel element and a yellow box appears with design information about the channel. This information includes basic design parameters (length, base, side slopes, grade) and flow information (peak flow, flow depth).

The View information is customized for each type of design element so that the data show both design and performance values.



L = 350 metre
Base = 0.60 metre; SS = 3.00 & 3.00
Grade = 0.25 %
Qpeak = 1.15 c.m/sec
Depth = 0.68 metre

Figure 9-15 – Channel information is revealed while in view mode.

9.1.4.3 The Layout Popup Menu - Print

The Print Area item has a sub-menu which includes Rectangular or Full Window choices as depicted in Fig 9-16.

If you select the Rectangular choice you can then sweep out a rectangular area anywhere on the layout and once you release your mouse button, this area will be printed immediately.

If you select Full Windows then once you click on this item the full layout window will be printed immediately.

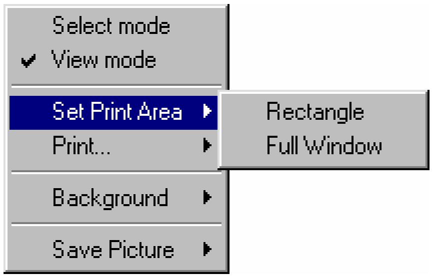


Figure 9-16 – Setting the area to be printed.

The Select Printer item also has a sub-menu (see Fig 9-17) which lets you change the printer used for layout printing. When you click on either item the normal Windows system of selecting a printer will appear. At that point you can decide on print quality, colour use, orientation etc. or any option that the printer driver has made available to you.

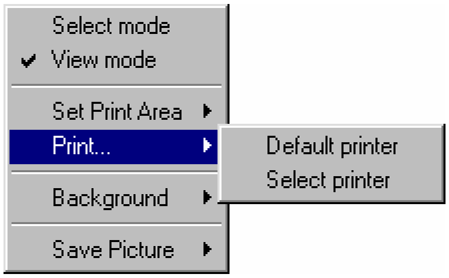


Figure 9-17 – Selecting the printer.

9.1.4.4 The Layout Popup Menu - Background

The Background menu item (Fig 9-18) lets you import or clear a graphical image on to a background layer of the layout window.

The Load Background image can be either a:

- BMP – Windows bitmap file, pixel by pixel drawing, usually large in size
- WMF – Windows Meta file, vector based drawing
- EMF – Enhanced Meta file, vector based drawing

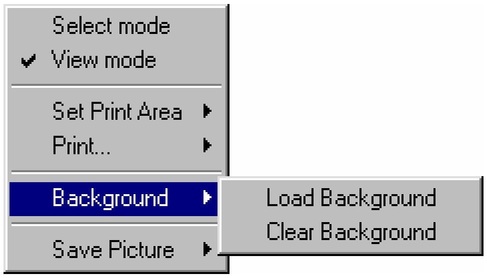


Figure 9-18 – Setting the background layer of the layout window.

On the layout banner you can adjust the size of the layout dimensions in the current length unit by adjusting the Width and Height fields. In Fig 9-19 below, the dimensions are 2000 metres wide by

1000 metres high. Therefore, if you are importing a background drawing file you should make sure that the dimensions of the imported graphic has the same aspect ratio as the layout drawing area.

Scale metre/m 7000.

Width 2000 Height 1000

Figure 9-19 – Specifying the layout dimensions to match the imported background.

If the dimensions of the layout and the imported graphic are different you will see distortion in the form of stretching or compression either vertically or horizontally. If this occurs you can adjust the Width and Height data until the distortion is reduced or eliminated.

The Clear Background selection will delete the background image that you have imported.

9.1.4.5 The Layout Popup Menu - Save Picture

The MIDUSS layout feature lets you export your layout (plus background if used) to a standard graphic file.

This export is done by selecting the Save Picture item and then deciding on the file format for the exported file. There is a choice of three formats as displayed in Fig 9-20 below. Exporting to a BMP format means you are writing a bitmap file on a pixel by pixel basis.

The other export options are to WMF and EMF formats which are vector based graphic files. Once used in another software package, these file types can be stretched in any manner without any distortion. This is because the lines in the drawing use coordinates as the starting and ending points for each line. If you stretch the graphic all the coordinates simply move in a coordinated way thus eliminating the distortion.

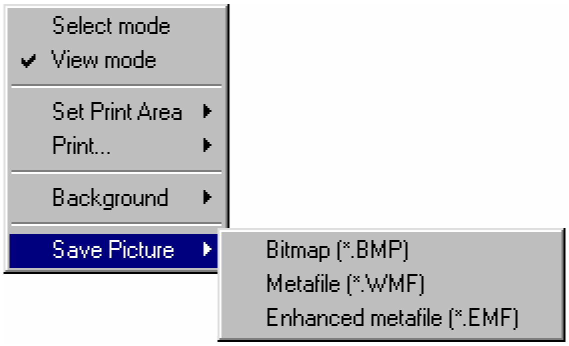


Figure 9-20 – Saving the layout and background to an external file.

The exported layout file a standard file name and folder location. These are:

- The filename is always either ‘**layout.bmp**’, or ‘**layout.wmf**’, or ‘**layout.emf**’ depending on the type of file format choice.
- The file is stored in the working folder for the current MIDUSS session.

Future updates of MIDUSS will provide a more flexible naming system and the ability to store in any folder.

9.1.5 Layout Objects

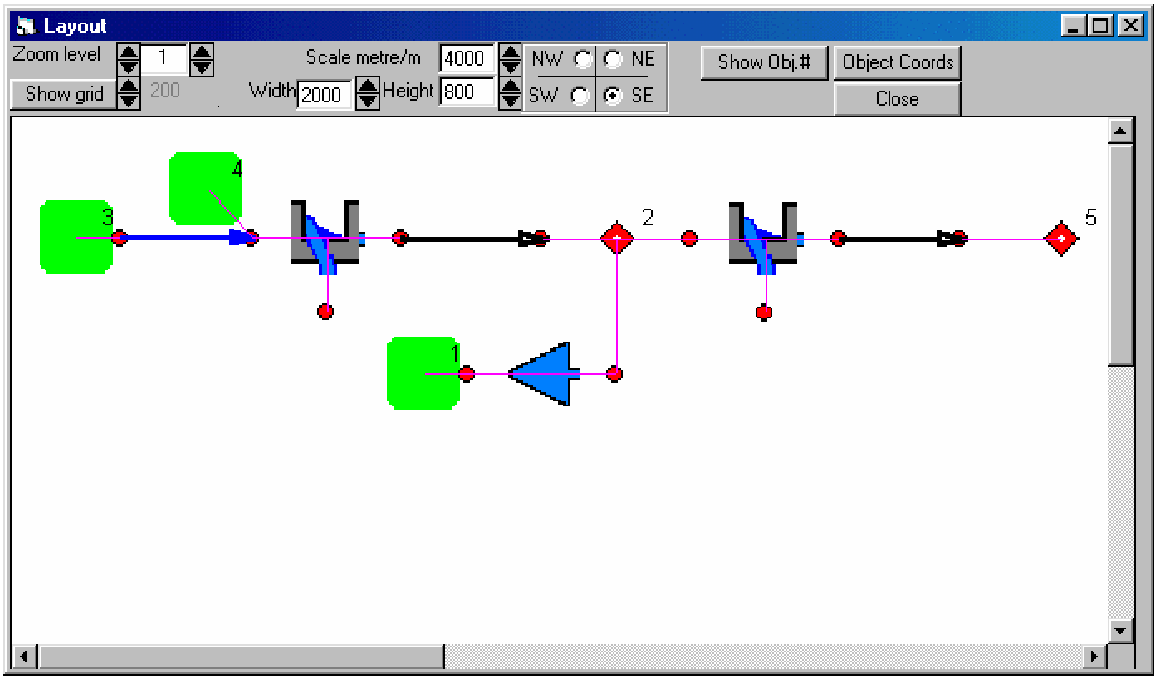
The presentation of your drainage network in the layout window is done using icons which represent the various elements of your design.

Fig 9-21 show a small network with various icons. Notice how they are connected together using dots and directional lines of different colour and thickness. These links represent pipes, channels that convey flow and also dummy links that merely indicate connectivity between objects.

The process of linking the icons together is done automatically by MIDUSS. As you design your system MIDUSS will join the element together in the proper manner. You can move the elements around, scale them and hover over each to see their status. The layout menu at the top of the window is supplemented with a popup menu that is displayed by pressing the secondary (right) mouse button.

It is important to understand the concept behind the layout system provided in MIDUSS. You cannot remove or add icons directly on the layout. Adding (and occasional deleting) of icons representing design elements is done using the hydrology and hydraulic design commands of the MIDUSS main menu. The MIDUSS design philosophy proposes that it is best to design and optimize one drainage element at a time as you progress downstream. If the graphical depiction of the network was to be performed first, with element optimization done after, then you would be faced with trying to optimize many different elements at the same time thus reducing your productivity and the quality of the design.

The design procedure used by MIDUSS requires that you start from the furthest upstream point in any branch of the drainage network and work downstream generating flows and designing drainage elements to manage the flows. Design decisions are therefore made interactively for only one element at a time. The Layout display maintains a graphical history of the sequence and connectivity of the various elements of your design. This is done whether the Layout window is open or closed. At any time you can modify the actual location of elements on the drawing area with a simple drag-and-drop procedure without changing the sequence and connectivity of the elements.



Zoom level 1

Scale metre/m 4000

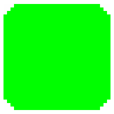
Width 2000

Height 800

○ NW ○ NE ○ SW ● SE

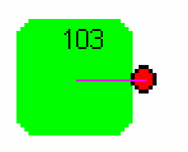
Figure 9-21 – Example of a small drainage network.

Fig 9-22 presents a list of icons that can appear on your layout.



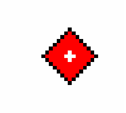
Catchment

Overland flow calculated.

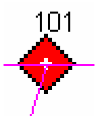


Catchment

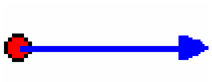
Overland flow added to the drainage network.



Junction

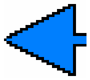


Junction with identifying number and links.

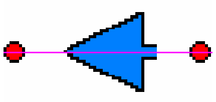


Channel

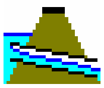
Pipe



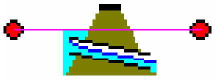
Pond



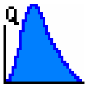
Pond with links



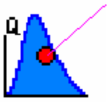
Culvert



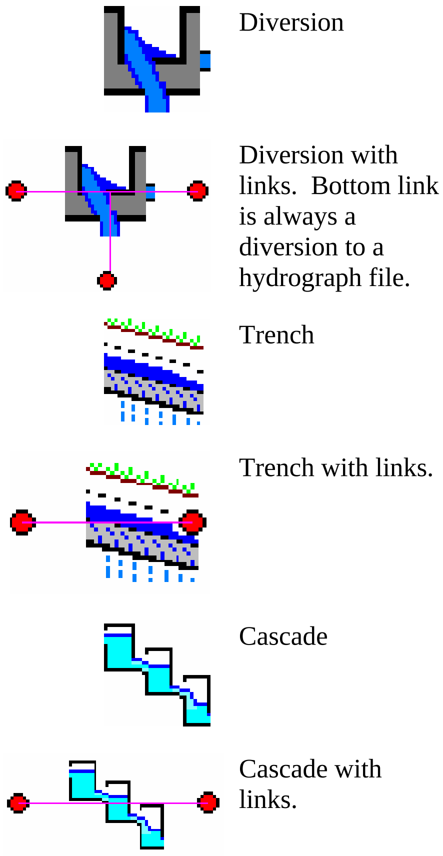
Culvert with links



Hydrograph



Hydrograph with link



Diversion

Diversion with links. Bottom link is always a diversion to a hydrograph file.

Trench

Trench with links.

Cascade

Cascade with links.

Figure 9-22 – The layout icons.

9.1.6 Showing the Design log

This command opens the Notepad text editor and displays the current contents of the Design log. This contains the accumulated files created during each of the Design functions which generate a log of changes and results.

The file can be saved under another name or printed at any point during the MIDUSS session.

9.1.7 Showing the Flow Peaks File

The Peak flows summary table is normally displayed in the lower, right corner of the screen. To allow you to save, print or otherwise manipulate this information a simple text file Qpeaks.txt is maintained in the MIDUSS directory. Both the summary table and the text file are renewed at the start of each MIDUSS session.

The **Show/Flow Peaks** command displays the current contents of the Qpeaks.txt file using the Notepad editor. The file can be saved under another name or in another directory or printed.

9.1.8 The Show/Tabulate command

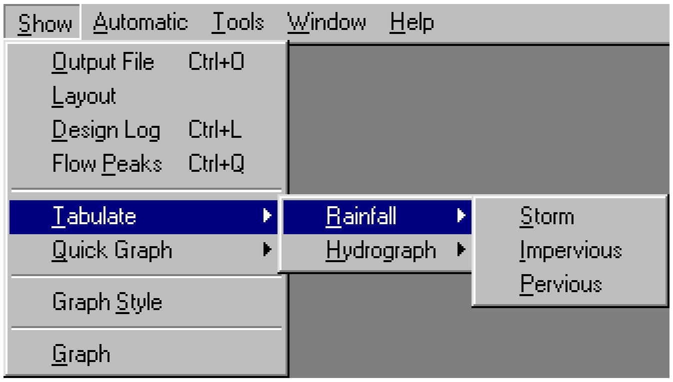


Figure 9-23 – You can display a table of hyetograph or hydrograph at any time

With all of the Hydrology or Design commands a table is displayed showing the affected rainfall hyetograph or flow hydrograph. This command causes a similar tabular display to be opened showing the contents of any hyetograph or hydrograph that is currently available.

The menu fragment shown above (Fig 9-23) indicates the available selection from the three rainfall or effective rainfall hyetographs. A similar display is shown for available hydrographs. In both cases the name of any array which is not currently available is shown in gray and cannot be selected.

Only one array can be tabulated at a time.

9.1.9 The Show/Quick Graph command

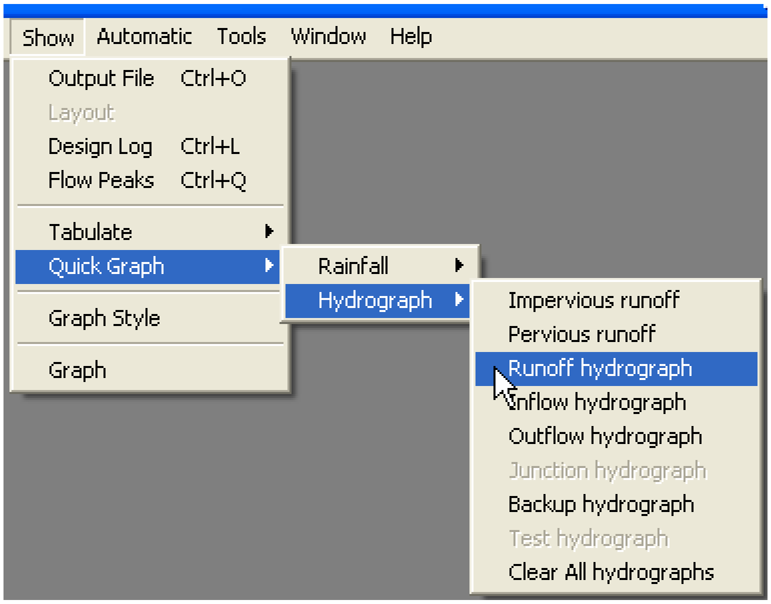


Figure 9-24 – Quick Graph shows a plot of any hyetograph or hydrograph

This command causes a graph to be displayed of any one of the currently available hyetographs or hydrographs as indicated in the menu fragment shown above in Fig 9-24. The names of hydrograph arrays which are not currently available are shown in gray and cannot be selected.

To display two or more hydrographs you simply use the Show/Quick Graph command as many times as required without closing the graph window. With each added hydrograph, a checkmark is added to the menu to indicate the displayed graphs.

An example is shown in figure 9-25 below. Note that it is not necessary to plot the largest hydrograph first as the scale is adjusted as each hydrograph is added.

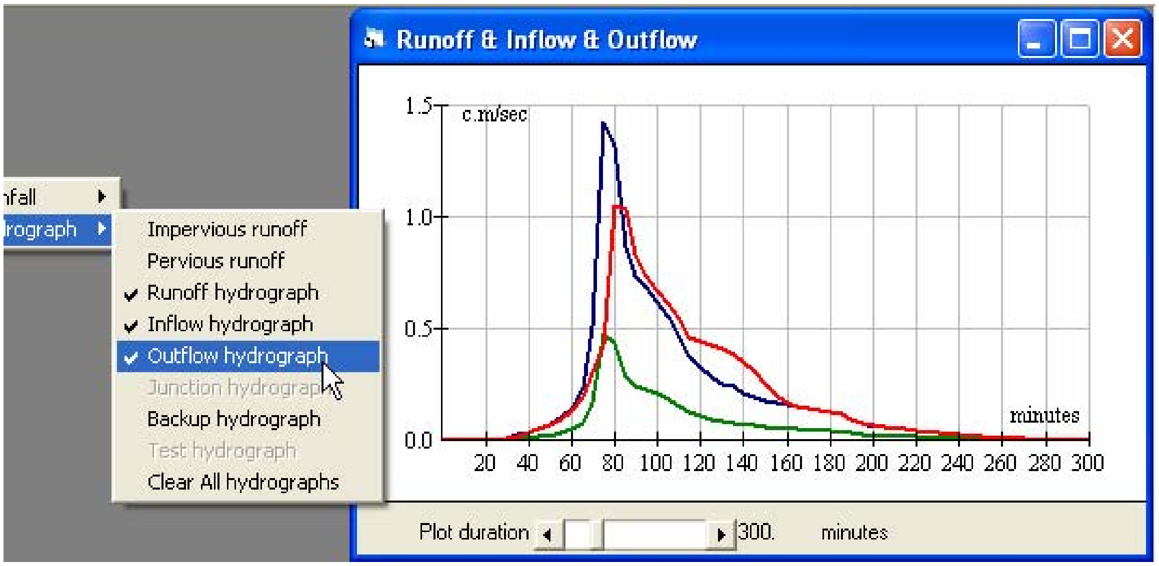


Figure 9-25 – Multiple plots using Quick Graph

9.2 Show Graph

Figure 9-26 – The Show/Graph command has a special menu

The purpose of this command is to let you build a composite graph containing one or more hydrographs and hyetographs together with custom annotations and simple graphic shapes. As illustrated above in Fig 9-26, the Main Menu bar is replaced with a customized Graph Menu.

The options displayed on the Graph Menu are summarized below and are discussed in more detail in the sections which follow.

Graph/File	Lets you save and load graph bitmap files, print hardcopy, minimize the Graph form or exit the Graph command
Graph/Edit	Contains options to enter text with different fonts and colours, draw simple graphical shapes such as lines, arrows, rectangles and circles and selectively erase rectangular areas of the graph
Graph/Plot	Select the rainfall hyetograph or flow hydrograph to be added to the graph, preview the graph and then add it to the current graph.
Graph/Scale	Allows vertical and horizontal scales to be set, select whether hyetographs are on the top or bottom edge, set the fraction of the plot height for hydrographs and hyetographs and toggle the grid and crosshairs off and on.
Graph/Graph Styles	Define or change the colours, patterns and line thickness to be used for different hyetographs and hydrographs.
Graph/Help	Opens the normal MIDUSS Help System.

9.2.1 Show/Graph/File Menu Options

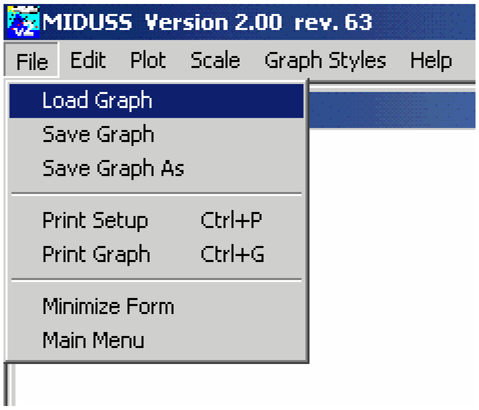


Figure 9-27 – Options available in the Graph/Files menu

The Graph menu shown in Figure 9-27 offers the following options:

Load Graph	Load a previously saved bitmap file (*.bmp) into the Graph window
Save Graph	Save the current contents of the Graph window as a bitmap file. The default file is called DefaultGraphFile.bmp and is stored in the job folder. It is approximately 0.5 MB in size.
Save Graph As	Save the current contents of the Graph window with a special name in the currently defined job folder.
Print Setup	Select a printer or printer parameters
Print Graph	Produce a hardcopy of the Graph window
Minimize Form	Minimize the Graph form to an icon. The Graph window can be restored by re-invoking the **Show/Graph** command.
Main Menu	Close the graph form (erasing the contents) and return to Main Menu

9.2.2 Show/Graph/Edit Menu Options

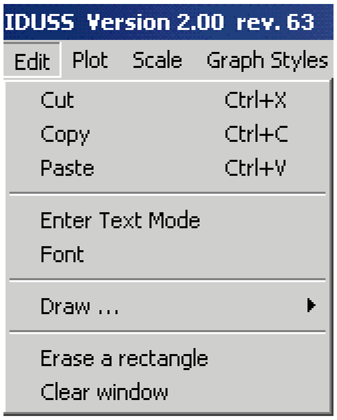


Figure 9-28 – Choices in the Graph/Edit menu let you annotate the plot.

This Graph menu item (Fig 9-28) offers the options listed below. Most of these are sufficiently complex that more detail is provided in the appropriate sub-sections which follow.

Enter Text Mode	Prepare to enter text on the Graph window
Font	Select a font (style, size, weight and colour) for text entry
Draw	Draw one of a number of simple shapes
Erase a Rectangle	Define a rectangle to be blanked out
Clear Window	Clear the entire Graph window and all Scale settings

9.2.3 Show/Graph/Edit/Enter Text Mode

Selecting this menu item changes the mouse pointer to a 'writing hand' and disables all of the other items in this menu with the exception of Font.

Click the primary (left) mouse button to change the ‘hand’ to a cross. Position the cross at the top left corner of the intended location of the text and click the primary (left) button to ‘set’ the starting point for the text.

Text can be entered from the keyboard but editing by means of the Backspace, Arrow and Delete keys is not available. The Enter key causes a new line to be started aligned vertically with the previous one.

Click the cross cursor at a new location to enter text with same attributes at a new location.

Pressing the Escape key or the End key restores the 'writing hand' pointer and you can move to another position to enter text.

If you wish, you can access the Font item in the Show/Graph/Edit menu and select size, font, colour and attributes from a standard dialogue box.

When you have finished entering text, click the menu item 'Text Entry mode' to de-select it (i.e. remove the check mark). The other Graph menu items will be enabled and the default mouse pointer will be restored.

9.2.4 Show/Graph/Edit/Font

This option lets you select the Font (e.g. Times Roman), Font Style (e.g. Bold), Point size and Font Colour from a Font dialogue box.

When you are in the process of entering text (i.e. when the ‘writing hand’ icon is displayed) this Graph menu item is the only one which is not disabled so that you can enter text items in a variety of styles.

9.2.5 Show Graph/Edit/Draw

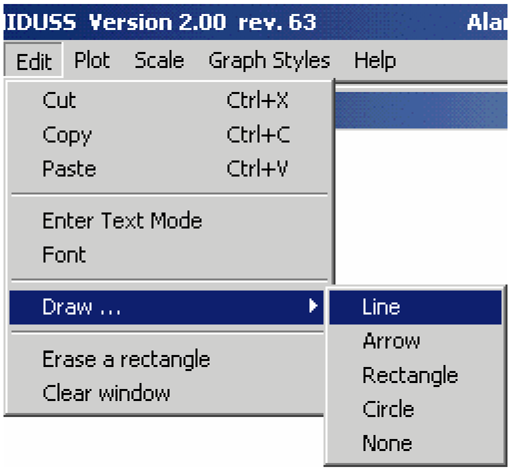


Figure 9-29 – Several simple shapes can be added to the graph

The menu item in Figure 9-29 shows the simple shapes that can be added to the Graph window by using the **Show/Graph/Edit/Draw** command. Shapes are drawn by clicking and holding down the primary (left) mouse button and dragging the pointer to the final position. A dynamic grayed image is displayed to let you decide on the desired size and shape of the object. When drawing an arrow the ‘arrow-point’ is at the start of the ‘drag’ operation, i.e. the arrow direction is the reverse of the drawing direction.

Once a shape is selected it is shown against the 'Draw' item as a reminder.

The 'None' option should be used to avoid drawing unwanted shapes accidentally.

9.2.6 Show/Graph/Edit/Erase a Rectangle

This option lets you erase a rectangular area from the current graph either to create a space for text entry or to correct a previous mistake.

To define the rectangle, click and hold down the primary mouse button and drag out the dotted rectangular frame.

9.2.7 Show/Graph/Plot Menu Items

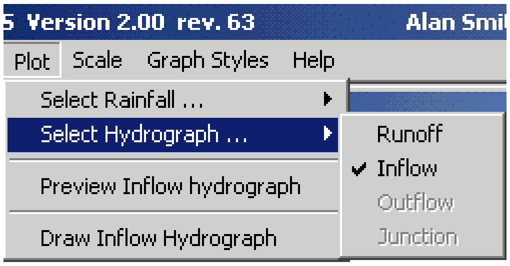


Figure 9-30 – The Graph/Plot menu lets you select, preview and draw flow data

These four menu items (Fig 9-30) let you select, preview and add a hydrograph or hyetograph to the current contents of the Graph window. The list opened by either of the **Select…** items will allow you to select only from hydrographs and hyetographs which contain significant data.

The checkmark against a list item indicates the last selected item. The **Preview…** and **Draw…** menu items are modified to show the last selected item.

Select Rainfall	Select one of the three rainfall hyetographs to plot
Select Hydrograph	Select one of the four flow hydrographs to plot
Preview Selection	Display a 'Quick Graph' of the selected hyetograph or hydrograph
Draw Selection	Add the selected item to the current contents of the Graph window

9.2.8 Show/Graph/Scale Menu Items

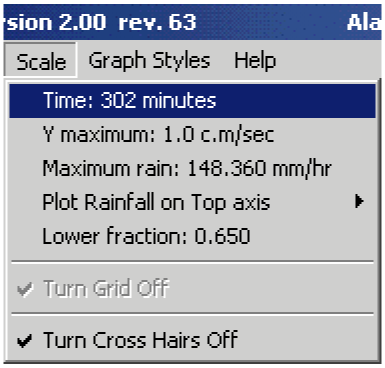


Figure 9-31 – The hyetographs can be added at top or bottom of the graph

These menu items (Fig 9-31) allow you to control the limits for scaling the graph. In many cases, these values must be set before the first hyetograph or hydrograph is plotted since subsequent objects must use the same scales to be consistent.

The graph window can be split horizontally to avoid hyetographs plotted on the top edge from overlapping with hydrographs on the bottom axis.

A brief description of each of the options is given below.

Maximum time	Set the time scale on the X-axis. Once the first item has been plotted the time scale cannot be changed.
Maximum flow	Set the highest hydrograph flow value which sets the scale for the lower fraction of the window which is used for hydrographs. Use this if you want to make room for a subsequent hydrograph which is larger than the first one plotted. Once this has been set by plotting the first hydrograph it cannot be changed.
Maximum rain	Set the maximum intensity with which a rainfall hyetograph can be plotted. Once a hyetograph has been plotted this value cannot be changed. If the first hyetograph is the total storm this will accommodate the effective rainfall on both pervious and impervious surfaces.
Plot Rainfall On	Lets you choose whether to plot the rainfall hyetographs on the bottom or top axis. Once this has been set it cannot be changed without clearing the window.
Lower Fraction	Allows you to split the window into upper and lower fractions to hold rainfall and hydrograph plots respectively. Once an object has been plotted, you cannot change this except by clearing the window. The default fraction is 0.65.
Turn Grid Off/On	Toggles the grid off or on.
Turn Cross Hairs Off/On	Toggles the cross-hairs which move with the mouse pointer when the primary mouse button is held down. This is useful for drawing simple shapes with reasonable precision. When the cross-hairs are enabled the X and Y coordinates are also displayed at the left end of the Title bar

9.2.9 The Show/Graph Styles Command

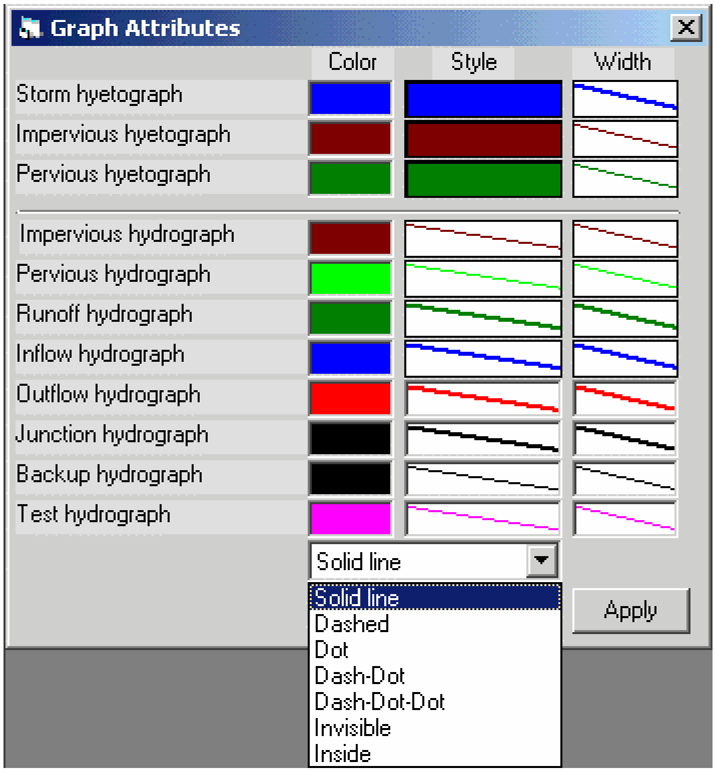


Figure 9-32 – You can customize the colour and fill attributes of the graph

This command (Fig 9-32) lets you customize the colours, patterns and line styles used in all of the graphic displays of hyetographs and hydrographs including the **Show/Graph** command. As shown, three columns let you select Colour, Style or Width to be used when graphing any of the three hyetographs and four hydrographs.

By clicking on a Colour choice a dialogue box is opened to reveal the choices available. However, if your screen display can show only a limited number of colours your choice may not always be displayed as expected. Some experimentation may be necessary.

For the three hyetographs, Style means a fill pattern. By clicking on one of the three boxes a drop down list is opened (as illustrated in Figure 9-32) which lets you choose from a small number of options.

Style for the four hydrographs means line pattern - continuous, dashed etc. This choice is also offered from a list box which is shown when you click on any of the Hydrograph/Style boxes. However, note that a line pattern other than continuous is available only if the line width is 1 pixel.

Width describes the thickness of the line in pixels. A restriction of Windows is that widths of more than 1 pixel will automatically default to a continuous line pattern. For the 3 hyetographs, the line width is used for the line enclosing the bar graph.

Notes:

Chapter 10 Running MIDUSS in Automatic Mode

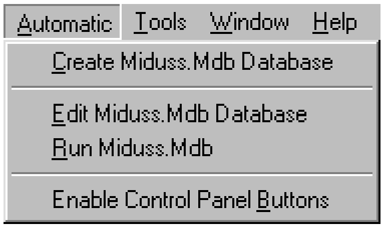


Figure 10-1 – The Automatic Menu

This chapter describes the commands that let you use MIDUSS in automatic mode. This is a rather unique feature of MIDUSS and it is worth learning these time-saving operations if you are (or plan to be) a regular user of MIDUSS.

The chapter begins with a number of general topics listed below.

- Reasons for Using Automatic Mode
- Files Used in Automatic Mode
- Structure of the Database File
- Advantages of Using a Database.

Following this general introduction a detailed description is given of the three steps available in the menu commands and the commands available within each step. The steps illustrated in the Automatic menu are introduced in the topic Steps to Run MIDUSS in Automatic Mode. More detail is given in the following sections.

- Create the MIDUSS.Mdb Database
- Edit the MIDUSS.Mdb Database
- Using the Automatic Control Panel

10.1.1 Reasons for Using Automatic Mode.

While MIDUSS is being run in manual mode, all commands and all relevant data are input from the keyboard and the results are displayed on the screen. If you have defined an output file a log is maintained on this file of all commands, input data and some of the results. The file contains all of the data necessary to duplicate the MIDUSS session. This file not only serves as a detailed record of the session but can be used to create an input file for use during a subsequent MIDUSS session, allowing all the recorded commands and data to be read in automatic mode, thus relieving you of the need to re- enter this information. This form of automatic processing will be found useful in a number of situations of which the ones described below are typical.

(a) A design session can be completed in several runs. The commands, data and design decisions of the first session are recorded and used during the second session to quickly get to the point where the previous session stopped. The design can then be continued in manual mode. The output file will then contain the commands and data for both sessions.

(b) A design may be completed using a storm hyetograph with (say) a five- year return period. Later, it may be desired to test the design under a more severe storm. Using the output file created in the previous session, this can be done by running through the previous design in automatic mode and redefining a new storm at the appropriate point.. An example is described in Chapter 11 *A Detailed Example*. – An Automatic Design for a Historic Storm.

(c) You may wish to revise one or more components in a previously completed design. This can be done by running the design in automatic mode and revising the design of the specific component(s). After a design decision is read from the input file - e.g. the diameter and gradient of a pipe - you can either accept or revise the design. The output file then contains any altered design parameters.

(d) You may want to add one or more commands in manual mode to those already captured in the output file. These commands may represent a change in the hydrologic modelling, the design of a new component in the drainage network or simply an opportunity to generate and print graphics for use in a final report.

The output file is a sequential, formatted file and can be displayed on the screen using a text editor such as Notepad or WordPad or copied to your printer.

If you have used an earlier version of MIDUSS you will notice that there very many differences between the method used in MIDUSS and that used previously. However, the general principle remains the same - that output from a manual session can be used to provide input for a subsequent run in Automatic mode. There is no currently available program to convert an output file created with an earlier DOS version of MIDUSS for use with MIDUSS.

10.1.2 File Structure Used for Automatic Mode

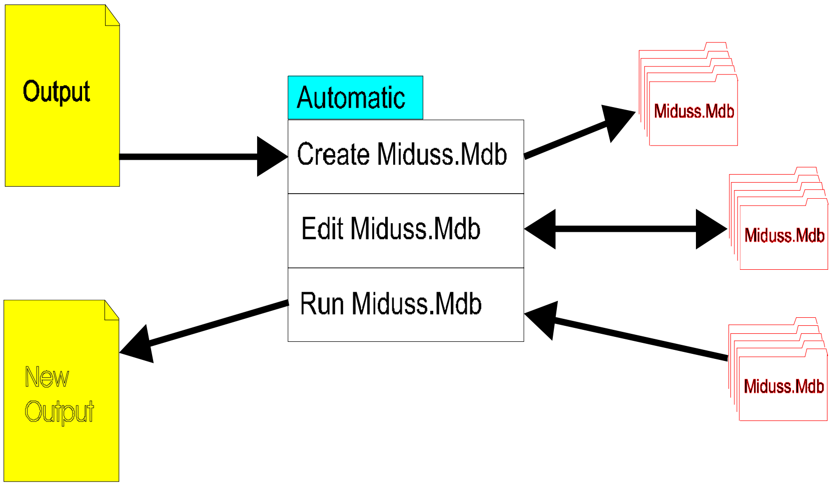


Figure 10-2 - Schematic of Automatic File Operations

Figure 10.2 above illustrates the three operations which are available under the **Automatic** command in the Main Menu. These commands operate on two types of file.

Output The Output file is an ASCII text file which has been produced by MIDUSS and which contains lines of text containing commands, data and results. The Output file can be read using any text editor such as Notepad or WordPad. These are included with your Windows 95/98/NT operating system.

The icon labeled 'New Output' represents a file which contains the results of a Run in Automatic mode and includes any changes (such as a larger storm event) made during the run.

MIDUSS.Mdb	This is a database file comprising a number of records (rows) each of which corresponds to a line in the Output file. The contents of the database can be viewed only with special software designed to handle databases such as Microsoft Access ®, dBase® or Paradox ®. MIDUSS contains procedures for viewing, editing and reading the contents of MIDUSS.Mdb. The default file extension for a database file is 'Mdb' and you should not change this. It is important to note that in order to be available as an input source, MIDUSS.Mdb must reside in the MIDUSS directory (e.g. C:\Program Files\MIDUSS98\ ). Obviously only one copy of MIDUSS.Mdb can exist in this directory. However, you can create a copy of the file in another directory should you wish to save it for future use. The four commands in the Automatic menu do the following.

Create MIDUSS.Mdb	Reads the contents of your Output file line by line and creates corresponding rows in the database MIDUSS.Mdb. The same operation is carried out automatically when you use the Files/Open Input File command in the **Files** menu.
Edit MIDUSS.Mdb	Displays the current contents of the database MIDUSS.Mdb and allows you to edit the data in any of the fields of any row. The Edit process cannot be used to insert new records (rows) into the database. You can do this by entering additional commands in Manual mode while running in Automatic mode.
Run MIDUSS.Mdb	Start reading and executing the commands in MIDUSS.Mdb. There are three modes in which you can run MIDUSS.Mdb which are controlled by the command buttons on the Control Panel.
Enable Control Panel Buttons	Sometimes you may find that the command button on the Control Panel that you want to use has been disabled. This command is designed for these situations and enables all of the command buttons.

10.1.3 Structure of the Database File

The database file can be visualized as a table made up of many rows and several columns. The rows are referred to as **Records**. Each record contains a number of **Fields** that correspond to the cell at the intersection of a row and column. The number of Fields or Columns is fixed and cannot be changed by the user.

The database file used by MIDUSS is always called MIDUSS.Mdb and it always resides in the MIDUSS folder. MIDUSS.Mdb contains 4 columns; it follows that every Record has four Fields. These Fields contain the following data:

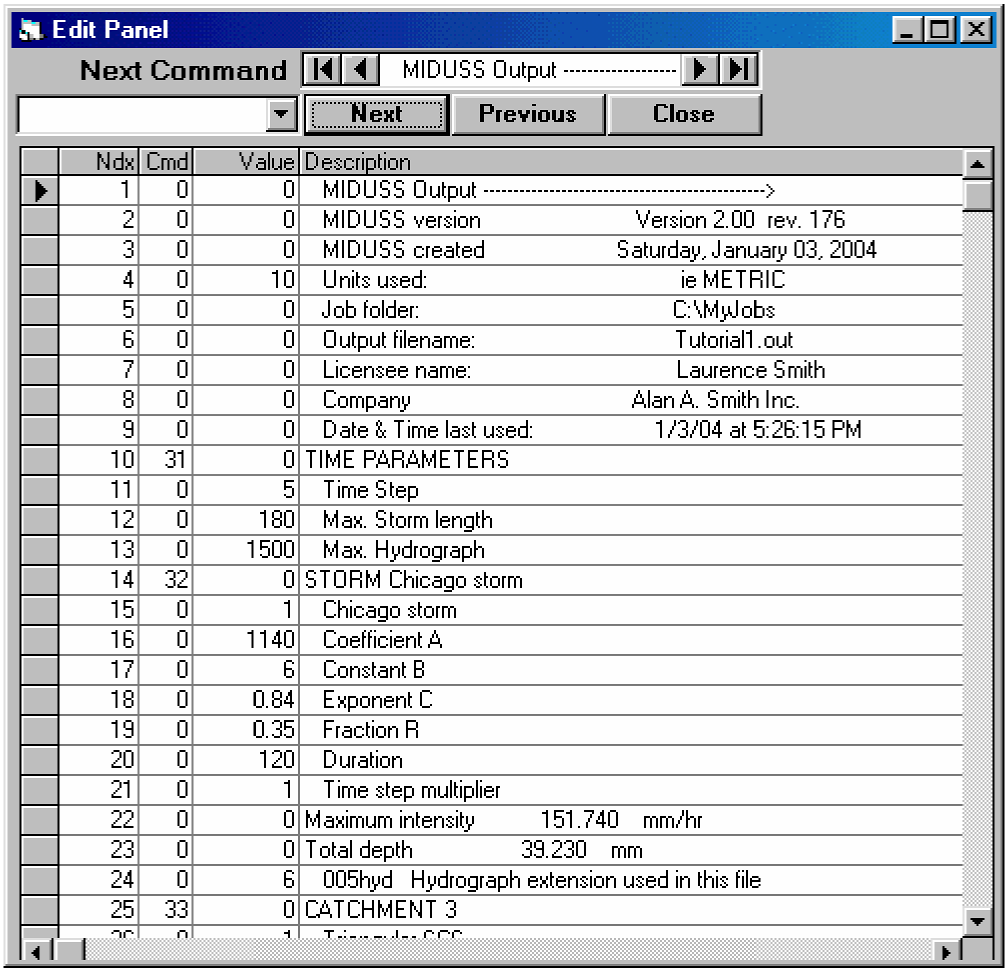
1) An index counter starting from 1 to the maximum number of lines in the Output file which was used to create MIDUSS.Mdb.

2) An integer which is either zero or an integer corresponding to one of the commands in the MIDUSS Main Menu. For example, the Hydrology/Time Parameters command is command 31 since it is in the third column of the Main Menu and is the first item in the Hydrology menu.

3) A value of a parameter required for input, such as a maximum storm duration, catchment area or pipe roughness.

4) A text or string variable which may contain a command name (e.g. STORM), descriptive text, a series of required numerical values such as historical rainfall or channel cross-section coordinates, or results for information.

The illustration below shows the top of a typical database.



Ndx	Cmd	Value	Description
1	0	0	MIDUSS Output --->
2	0	0	MIDUSS version Version 2.00 rev. 176
3	0	0	MIDUSS created Saturday, January 03, 2004
4	0	10	Units used: ie METRIC
5	0	0	Job folder: C:\MyJobs
6	0	0	Output filename: Tutorial1.out
7	0	0	Licensee name: Laurence Smith
8	0	0	Company Alan A. Smith Inc.
9	0	0	Date & Time last used: 1/3/04 at 5:26:15 PM
10	31	0	TIME PARAMETERS
11	0	5	Time Step
12	0	180	Max. Storm length
13	0	1500	Max. Hydrograph
14	32	0	STORM Chicago storm
15	0	1	Chicago storm
16	0	1140	Coefficient A
17	0	6	Constant B
18	0	0.84	Exponent C
19	0	0.35	Fraction R
20	0	120	Duration
21	0	1	Time step multiplier
22	0	0	Maximum intensity 151.740 mm/hr
23	0	0	Total depth 39.230 mm
24	0	6	005hyd Hydrograph extension used in this file
25	33	0	CATCHMENT 3

Figure 10-3 - Layout of MIDUSS.Mdb in the Edit Panel

10.1.4 Advantages of Using a Database File

Previous versions of MIDUSS used a simple ASCII text file for both Input and Output. Using a database instead of a text file for input has a number of advantages.

1) The database is structured so that any record can be accessed directly instead of having to read records sequentially from the beginning of the file. This is much more efficient and allows large database files to be manipulated very rapidly.

2) Due to the direct access described above it is possible to link or 'bind' the database to a number of different controls or objects on the screen. The most convenient is a grid which can have rows and columns corresponding to the records and fields of the database. This is the grid that you see in the Edit Panel (Figure 10-3) and in the Control Panel which is displayed when running MIDUSS in automatic mode.

3) Linking the database to a visible grid allows the database file to be used as an input source for MIDUSS while still making it visible to the user.

4) The visible records allow you to keep track of the progress of the run and anticipate commands where you may wish to take some special action.

5) The grid displayed allows you to make changes to the data during the run and any changes entered into the grid are immediately reflected in the Input database.

6) MIDUSS has a special data editing feature which lets you change a command number to the equivalent negative value. This causes a continuous run to stop and revert to the step-by-step Edit mode. This lets you run at speed up to a point in the input data where you want to take control or perhaps revert to Manual mode.

10.2 Steps to Run MIDUSS in Automatic Mode

There are three steps in using an existing Output file to run MIDUSS in Automatic mode

(1) Create the Input database MIDUSS.Mdb

(2) Review and/or Edit the Input Database MIDUSS.Mdb

(3) Use MIDUSS.Mdb to run MIDUSS in Automatic mode

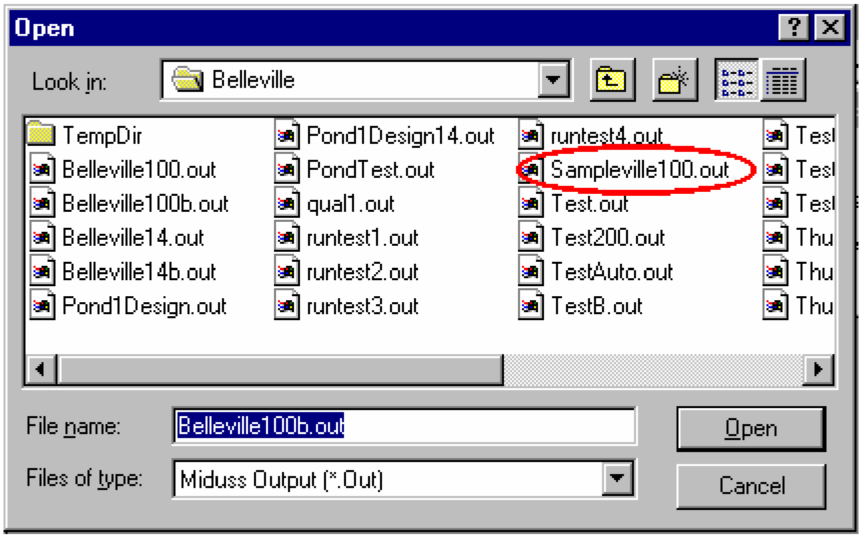
These steps are described in more detail in the sections which follow.

10.2.1 Creating the Input Database MIDUSS.Mdb

This first step assumes that you have already completed a session of MIDUSS in manual mode and can access the output file. If you didn't explicitly name an Output file, the output will be contained in the file DEFAULT.OUT which can be found in the MIDUSS working directory (default is typically C:\MIDUSSData\).

If this is the case it is probably wise to make a copy of DEFAULT.OUT under a different name and preferably in a different Job Directory. It's also a good idea to look at the output using Notepad or WordPad to re-assure yourself that you are working with the correct file.

You may also find it useful to have a printout of the file for your first use of Automatic mode or for a long and complex file.

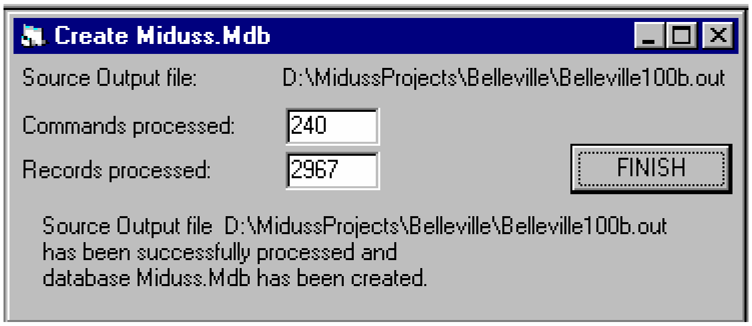

Look in: Belleville

File name: Belleville100b.out

Files of type: Miduss Output (*.Out)

Figure 10-4 - Selecting an Output file from a Dialog Box

When you select the **Automatic / Create MIDUSS.Mdb** command a dialog box is opened as shown above and you can navigate to your Job directory and select an existing output file as shown circled in red. Pressing the [Open] button causes the dialog box to close and a small progress form with the title 'Create MIDUSS.Mdb' is opened displaying the number of commands and the total number of records converted from the output file to the database MIDUSS.Mdb.



Source Output file: D:\MidussProjects\Belleville\Belleville100b.out

Commands processed: 240

Records processed: 2967

Source Output file D:\MidussProjects\Belleville\Belleville100b.out
has been successfully processed and
database Miduss.Mdb has been created.

Figure 10-5 - The Create MIDUSS.Mdb window.

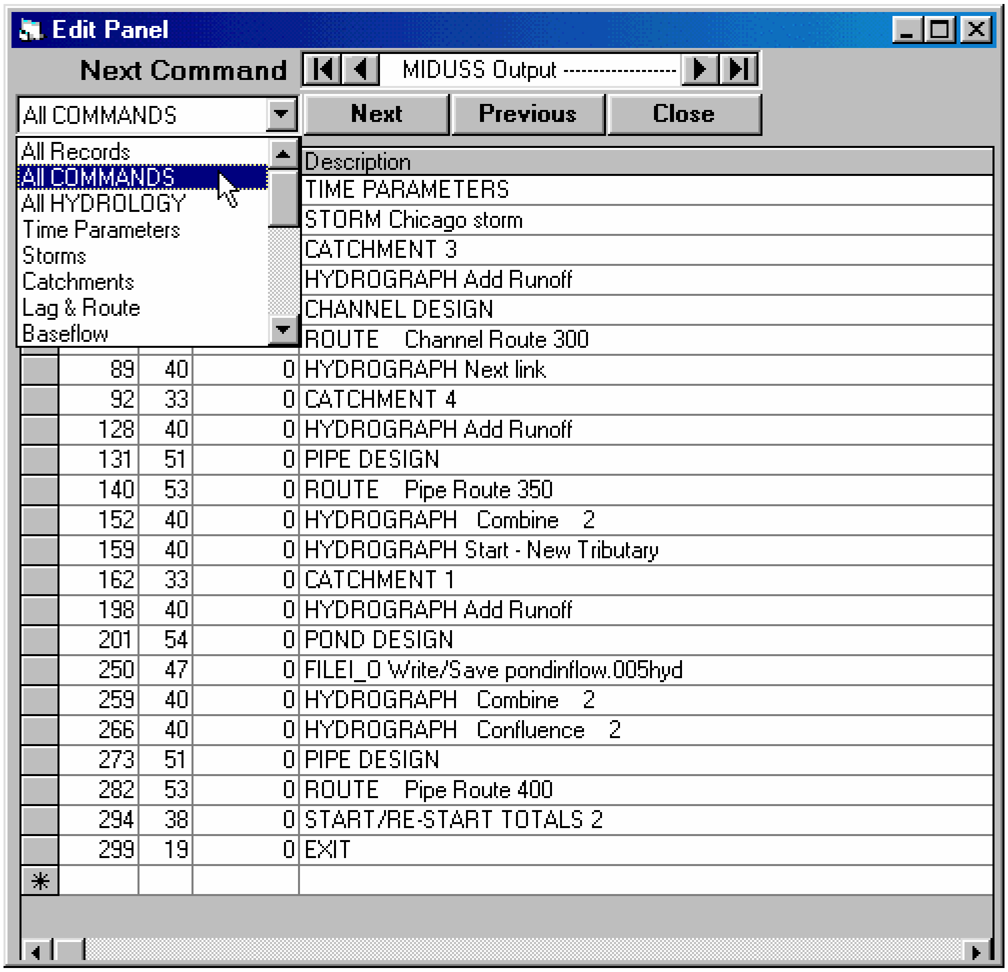
When the process is complete the command button [Finish] is enabled and you can click on this to close the window. The command will create a database file with the name MIDUSS.Mdb which resides in the MIDUSS98 folder.

The same procedure is automatically carried out when you select the **Files / Open Input File** command from the **Files** menu.

Once you have created MIDUSS.Mdb the next step is often to review or edit the MIDUSS.Mdb Database

10.2.2 Edit the Input Database MIDUSS.Mdb

The next step is to review the database either to ensure that the correct file has been created, or to make some change to the data. The diagram below shows the top portion of a typical database contained in the Edit Panel window.



			Description
			TIME PARAMETERS
			STORM Chicago storm
			CATCHMENT 3
			HYDROGRAPH Add Runoff
			CHANNEL DESIGN
			ROUTE Channel Route 300
89	40	0	HYDROGRAPH Next link
92	33	0	CATCHMENT 4
128	40	0	HYDROGRAPH Add Runoff
131	51	0	PIPE DESIGN
140	53	0	ROUTE Pipe Route 350
152	40	0	HYDROGRAPH Combine 2
159	40	0	HYDROGRAPH Start - New Tributary
162	33	0	CATCHMENT 1
198	40	0	HYDROGRAPH Add Runoff
201	54	0	POND DESIGN
250	47	0	FILEI_O Write/Save pondinflow.005hyd
259	40	0	HYDROGRAPH Combine 2
266	40	0	HYDROGRAPH Confluence 2
273	51	0	PIPE DESIGN
282	53	0	ROUTE Pipe Route 400
294	38	0	START/RE-START TOTALS 2
299	19	0	EXIT

Figure 10-6 - The header of the Edit MIDUSS.Mdb window

The rectangle above the three command buttons is called the Data Control. The arrow symbols at the left and right ends of the Data Control allow you to move forward or backward through the file.

◀ Backwards one record at a time, or

|◀ Move directly to the beginning of the file.

▶ Forward one record at a time, or

▶| Move directly to the end of the file.

The three command buttons let you do the following.

1)	[**Next**]	Move the current record to the record containing the next command
2)	[**Previous**]	Move the current record to the record containing the previous command
3)	[**Close**]	Close the Edit Panel window.

When you use the [Next] and [Previous] command buttons the pointer is located on the record containing the next or previous command and the name of the command is displayed in the text section of the Data Control. Pressing [Next] at the end of the file moves the pointer (i.e. the current record) to the start of the file. Similarly, pressing [Previous] near the top of the file moves the pointer to the last command of the database - normally the EXIT command.

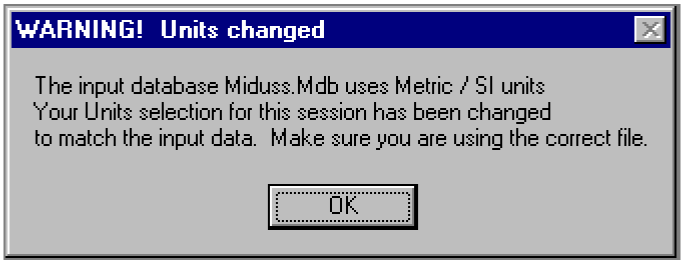
The 'Drop-down' List box lets you select from various types of record such as 'All HYDROLOGY', or 'Catchment'. Selecting one of these causes the full database to be replaced with a list of records corresponding to your choice. This is a useful way to review (say) all of the Catchment commands used. The description will also show the Catchment ID number and you can quickly find the Index number corresponding to a particular Catchment. Selecting 'All Records' from the Drop-down List box causes the full data base to be restored and you can navigate to the desired record.

Now that the input database is ready you can run it using the automatic Control Panel. However, it is important that the units used are consistent. MIDUSS takes care of this for you.

10.2.3 Using Consistent Units

When you start a new session of MIDUSS one of the things you are required to specify is the system of units to be used. If you now start to run the current MIDUSS.Mdb in automatic mode it is essential that the units being used for the session must match the units in the input database.

In case you have made an error, MIDUSS compares the units used in MIDUSS.Mdb with your current selection. If these are not the same MIDUSS changes the selection of units for the session to match the units in the input file. A warning message similar to that shown below is displayed and the current output file is re-written with the modified Units parameter.



The input database Miduss.Mdb uses Metric / SI units
Your Units selection for this session has been changed
to match the input data. Make sure you are using the correct file.

Figure 10-7 – MIDUSS checks to make sure the units are consistent

The error may have been due to either

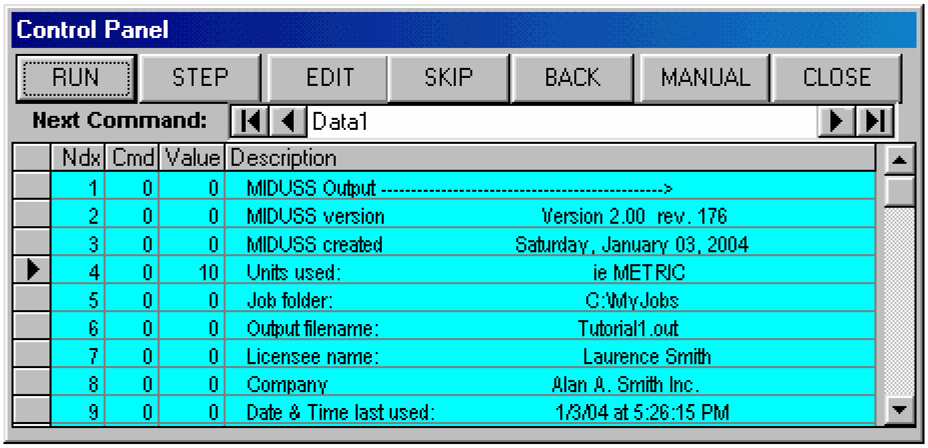
- selecting the wrong units from the Options/Units menu, or
- using the wrong input file.

If the wrong file has been used you will need to either Exit from and re-run MIDUSS or use the Files/Quit and Start Over command. Although MIDUSS protects you from this type of error it is always advisable to use the Edit panel to review the contents of MIDUSS.Mdb before starting the run.

Now that the input database is ready you can run it using the Automatic Control Panel.

10.2.4 Using the Automatic Control Panel

When you are ready for the run use the **Automatic / Run MIDUSS.Mdb** command. The Automatic Mode Control Panel is displayed. This contains a grid which is linked to the database MIDUSS.Mdb and a number of control buttons.



Next Command: Data1

	Ndx	Cmd	Value	Description
	1	0	0	MIDUSS Output --->
	2	0	0	MIDUSS version Version 2.00 rev. 176
	3	0	0	MIDUSS created Saturday, January 03, 2004
▶	4	0	10	Units used: ie METRIC
	5	0	0	Job folder: C:\MyJobs
	6	0	0	Output filename: Tutorial1.out
	7	0	0	Licensee name: Laurence Smith
	8	0	0	Company Alan A. Smith Inc.
	9	0	0	Date & Time last used: 1/3/04 at 5:26:15 PM

Figure 10-8 - The Automatic Mode Control Panel

The Control Panel is usually positioned in the lower, right corner of the MIDUSS window, immediately above the Peak flow summary table and normally displays only the section of the database which contains the current record. The current record is indicated by the arrow-head in the left column (circled in red in Figure 10-8 above). You can increase the number of records displayed by dragging the top edge of the Control Panel upwards and you can use the vertical scroll-bar to move through the database either to see commands still to be executed or to edit data which is yet to be read and used.

However, be careful if you manually change the current record by clicking on a row to edit some data. Processing of the database will continue ***from that record*** and you should re-set the current record before continuing unless you deliberately want to skip to the edited record to ignore or repeat a section of the database.

Figure 10-9 – An Automatic run can use three modes

The first three command buttons let you control the manner in which the automatic run is made. These are:

- The [RUN] Command button
- The [STEP] Command button
- The [EDIT] Command button

The next two buttons let you move forward or backward through the database by skipping from one command to another. There is a minor difference in the manner in which you navigate through the database in the Edit Panel and in the Control Panel.

The two buttons are:

- The [SKIP] Command button

- The [BACK] Command button

In the Edit Panel pressing [Next] or [Previous] positions the pointer on a command line.

In the Control Panel the Current Record (as indicated by the pointer in the extreme left column) is positioned 1 record before the actual command line.

Finally, the last two buttons let you terminate the automatic session.

- The [MANUAL] Command button
- The [CLOSE] Command button

The MANUAL command closes the Control Panel but creates a 'bookmark' at the record where this was done. Then, if you use the **Automatic/Run MIDUSS.Mdb** command again, processing will start from this bookmark.

10.2.4.1 The Control Panel RUN command

[RUN] This causes commands to be executed consecutively and continuously until one of three things occur.

- The end-of-file is reached as signified by the EXIT command (Command #19).
- You press either the [EDIT] or [STEP] or [MANUAL] commands
- A negative command number is encountered. Changing a command number to be negative in the Edit Panel is a convenient way of causing an automatic run to stop at a pre-determined point.

In this mode you will not be able to see much detail and the only indication of progress is from the current record displayed in the Control Panel or by the summary of peak flows which is continuously updated.

10.2.4.2 The Control Panel STEP command

[STEP] This mode executes and closes commands one at a time, one command being completed for each mouse click on the [STEP] button. As with the [RUN] command, this mode also provides little opportunity to see the results of the command and progress can be monitored by watching the Control Panel or the Peak Flows summary table. However, between commands you can always open the current (new) output file using the **Show/Output File** menu command which will contain a record of the run up to this point.

10.2.4.3 The Control Panel EDIT Command

[EDIT] This mode provides the greatest control on the automatic execution of the commands in the input database. With each click on the [EDIT] command the next command is executed but the final result is displayed on the screen and remains there until you press the [Accept] button on the appropriate form. The mouse pointer is automatically located over the [Accept] button on the just completed command and the [EDIT] button on the Control Panel is Disabled. When you click on [Accept] the [EDIT] button is enabled and the mouse pointer is located over it to let you quickly proceed to the next command

The [EDIT] mode lets you change any of the data on the form from that which was read from the input database. Typical uses of this feature might be to change the magnitude of the storm event at the beginning of the session in order to test a design under the impact of a more severe event, or increase the base width of a channel.

10.2.4.4 The Control Panel SKIP Command

[SKIP] This command moves the pointer (which indicates the current record) from the current position immediately before a command to the record immediately before the next command. The text display on the Data Control displays the name of the next command. In some commands, an additional parameter is shown (such as a catchment ID number) to help you judge where you are in the file. When the EXIT command is encountered the pointer moves to the beginning of the file (BOF) and points to the record before the first command.

You can also navigate forward through the file by pressing the forward arrow at the right hand end of the Data Control or simply by using the vertical scroll bar of the grid and clicking the mouse on the left column of any record. Note, however, that if you position the pointer exactly on a command line, pressing [EDIT] will NOT execute that command but the next one. Position the pointer on the record immediately prior to the next command to be executed.

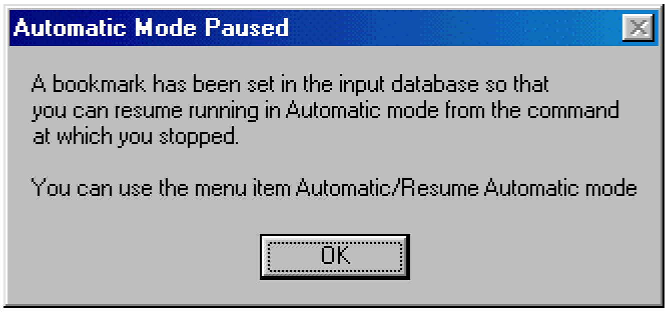
10.2.4.5 The Control Panel BACK Command

[BACK] This button performs the reverse of the [SKIP] command. The current record and its pointer move back towards the Beginning Of File (BOF) stopping at the record immediately before the previous command. At the BOF or the first command pressing [BACK] moves the pointer to the last command in the database which should be the EXIT command.

You can also move backwards through the database by clicking on the back arrow at the left end of the Data Control. You can also position the pointer (and therefore the current record) by clicking on the left (grayed) column in any record. Note, however, that if you position the pointer exactly on a command line, pressing [EDIT] will NOT execute that command but the next one.

10.2.4.6 The Control Panel MANUAL Command

[MANUAL] When running in automatic mode pressing this button causes execution to stop and MIDUSS reverts to manual mode. The Control Panel is closed but a 'bookmark' is stored to identify the point at which Automatic processing was stopped. If you re-start Automatic processing by using the **Automatic / Run MIDUSS.Mdb** command processing will start from where you left off.



Automatic Mode Paused

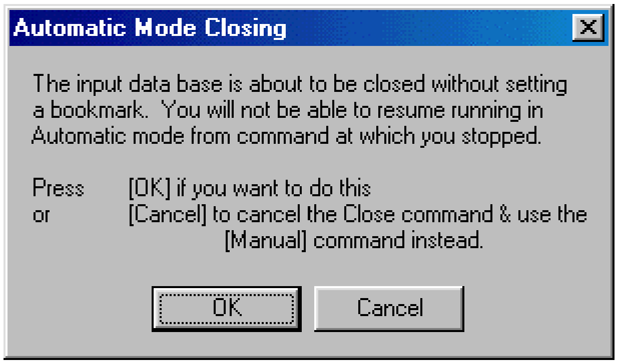
A bookmark has been set in the input database so that you can resume running in Automatic mode from the command at which you stopped.

You can use the menu item Automatic/Resume Automatic mode

Figure 10-11 – MIDUSS tells you a bookmark has been set.

10.2.4.7 The Control Panel CLOSE Command

[CLOSE] This button stops any automatic processing and closes the Control Panel. The [CLOSE] button is disabled (i.e. Grayed out) until the input file being read has encountered and processed the EXIT command. Clicking on [CLOSE] when it is enabled will close the Control Panel window and return MIDUSS to manual mode. At this point you may re-define and create a new input database MIDUSS.Mdb and run that in Automatic mode. In this way you can use the [SKIP] button to select and combine portions of 2 or more input files.



Automatic Mode Closing

The input data base is about to be closed without setting a bookmark. You will not be able to resume running in Automatic mode from command at which you stopped.

Press [OK] if you want to do this
or [Cancel] to cancel the Close command & use the [Manual] command instead.

Figure 10-10 – On closing automatic mode, MIDUSS will provide this message.

Notes:

Chapter 11 MIDUSS Tools

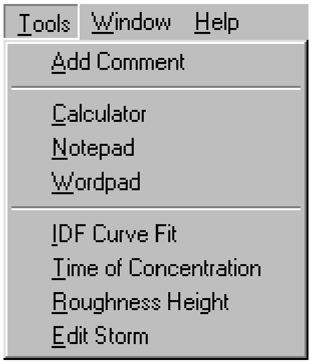


Figure 11-1 The Tools Menu

Add Comment	Lets you enter a comment into the current MIDUSS session
Calculator	Launches the Microsoft CALC utility
Notepad	Launches the Microsoft Notepad editor
WordPad	Launches the Microsoft WordPad editor
IDF Curve Fit	Computes Chicago storm parameters ‘a’, ‘b’ & ‘c’ for observed data
Time of Concentration	Estimates the time of concentration at various locations in the drainage network
Roughness Height	Converts roughness element height to a Manning ‘n’ value
Edit Storm	Lets you modify an existing *.MRD storm file or create a new one

11.1 Adding a comment

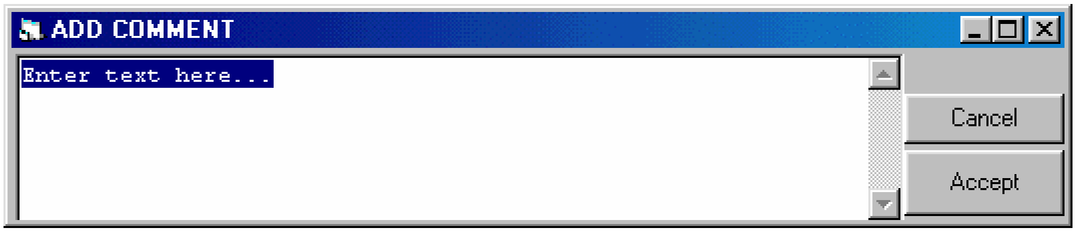


Figure 11-2 The Add Comment window

When you run MIDUSS there is always an Output file, either specified explicitly by the user or the default output file ‘Default.out’ in the currently selected Job Directory. This menu command lets you add explanatory comments to the output file.

Clicking on the **Tools / Add Comment** menu item causes a small window to open with a multi-line text box. You simply type in the Text box using normal text editing features such as Backspace, Delete or mouse controls. The font used is a uniformly spaced Courier font to allow columns to be aligned if desired. The text box entry provides automatic word wrap but this may not correspond exactly to the location of new lines in the output file.

The double quote character cannot be included in a comment and is automatically trapped and converted to a single quote.

The comment window can be re-sized by dragging the right side or the bottom of the window. The text entry box will also be re-sized. You can enter as many lines of comment as you wish. A vertical scroll bar appears when there are more lines than the text box can display.

The maximum line length which is written to the output file is 60 characters and you cannot use a ‘word’ or string of characters more than 59 characters in length.

Another use for the Add Comment command is to add a visual description to the MIDUSS window before printing hardcopy of the screen. If you don’t want this added to the output file press [Cancel] to close the window.

11.2 Microsoft Accessory Programs

All Microsoft Windows operating systems come installed with Notepad, WordPad and Calculator. These three programs can be accessed directly from MIDUSS using from the Tools menu.

MIDUSS processes a variety of data files that are created during a design session. Most of these are Text files that can be opened and read by Notepad. These Text files include:

- Output file - *.out
- Design log - *.log
- Mass rainfall - *.mrd
- Hyetograph files - Storm files with a custom extension (typically the storm return period e.g. *.005)
- Hydrograph files - *.hyd
- Junction files - *.jnc
- Old junction files - *.jnk
- Qpeaks.txt – used to store data on hydrograph manipulation

MIDUSS provides direct access to Notepad as a convenience should you wish to view these files from their working folder.

MIDUSS also uses Notepad directly within a session to display the Design log and Output files. To show these files go to the Main Menu and select the appropriate Show item.

Microsoft WordPad is also provided as a convenience to you. In some cases a Text file can become too large to be loaded by Notepad. If this is the case then you can use WordPad to view the data. When using the **Show / Output File** command, MIDUSS automatically selects Notepad.exe or Wordpad.exe depending on the size of the current output file.

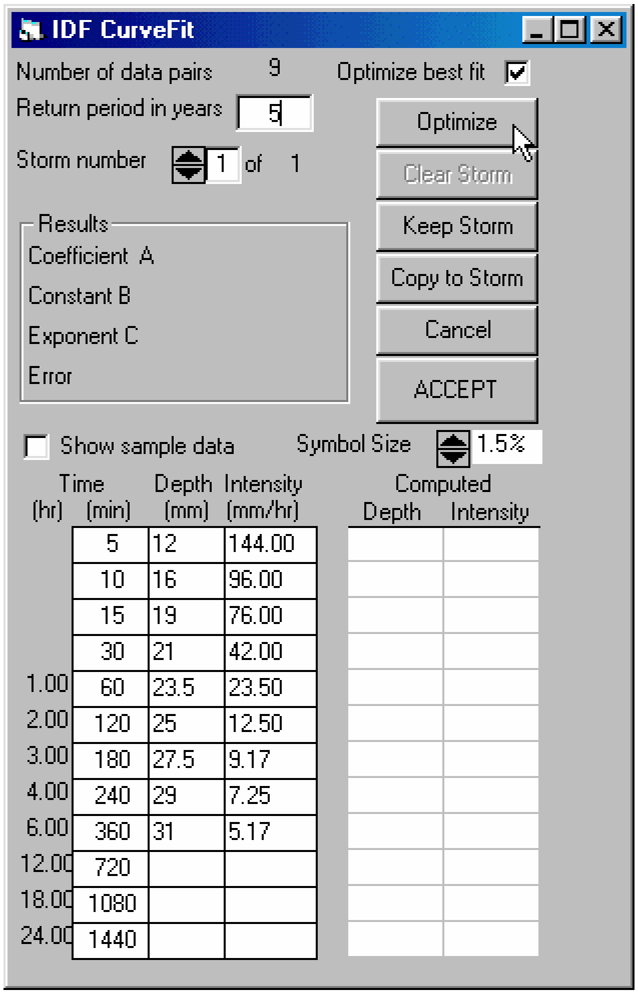
Microsoft Calculator is provided as a tool should you need to do a quick calculation. It is not used by MIDUSS in any other design session role.

11.3 Using IDF Curve Fit

The IDF Curve Fit tool manipulates data describing an Intensity-Duration-Frequency relates for a particular geographical locality and can be used in two modes:

(1) To compute the ‘a’, ‘b’ and ‘c’ parameters of a Chicago hyetograph that most closely approximates a set of observed rainfall data.

(2) To compute the IDF curve for user-supplied values of the three coefficients and compare this with observed data.



Number of data pairs 9

Optimize best fit ☑

Return period in years 5

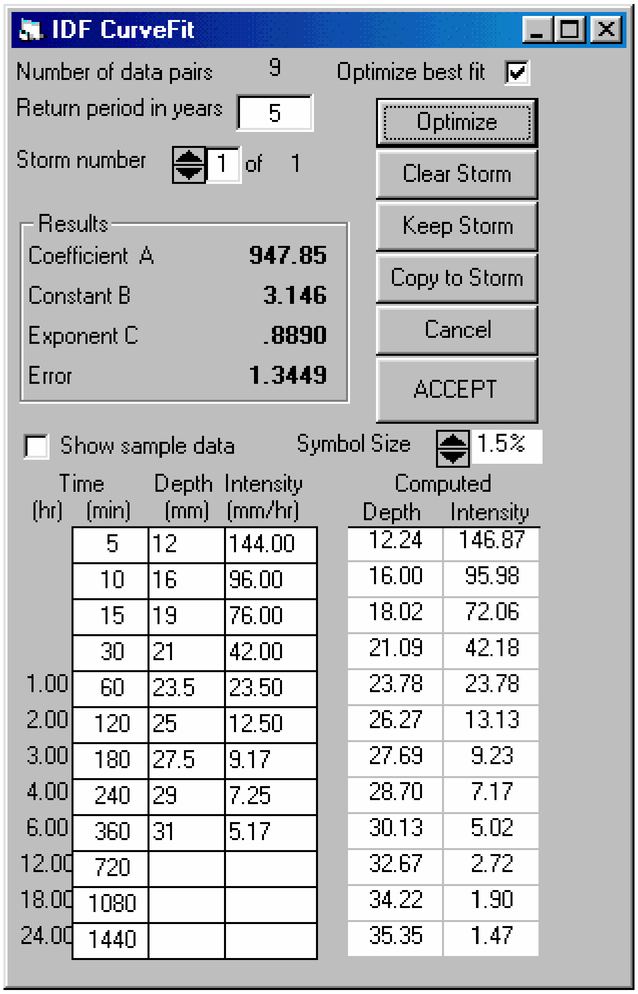
Storm number 1 of 1

☐ Show sample data

Symbol Size 1.5%

Time (hr)	Time (min)	Depth (mm)	Intensity (mm/hr)	Computed Depth	Computed Intensity
	5	12	144.00		
	10	16	96.00		
	15	19	76.00		
	30	21	42.00		
1.00	60	23.5	23.50		
2.00	120	25	12.50		
3.00	180	27.5	9.17		
4.00	240	29	7.25		
6.00	360	31	5.17		
12.00	720				
18.00	1080				
24.00	1440				

Figure 11-3 The IDF Curve Fit window with observed rainfall data entered



Number of data pairs 9
Optimize best fit ☑
Return period in years 5
Storm number 1 of 1

Results
Coefficient A **947.85**
Constant B **3.146**
Exponent C **.8890**
Error **1.3449**

☐ Show sample data
Symbol Size 1.5%

Time (hr)	Time (min)	Depth (mm)	Intensity (mm/hr)	Computed Depth	Computed Intensity
	5	12	144.00	12.24	146.87
	10	16	96.00	16.00	95.98
	15	19	76.00	18.02	72.06
	30	21	42.00	21.09	42.18
1.00	60	23.5	23.50	23.78	23.78
2.00	120	25	12.50	26.27	13.13
3.00	180	27.5	9.17	27.69	9.23
4.00	240	29	7.25	28.70	7.17
6.00	360	31	5.17	30.13	5.02
12.0	720			32.67	2.72
18.0	1080			34.22	1.90
24.0	1440			35.35	1.47

Figure 11-4 The IDF Curve Fit window with computed results

The mode is selected by checking the ‘Optimize’ check box on the form or clearing it to simply compute the curve for specified values of ‘a’, ‘b’ and ‘c’.

Figure 11-3 shows data that has been entered for the first “Optimize’ mode of operation.

In the grid on the left for Time, Depth and Intensity a column of time intervals is displayed as shown. These values can be customized if desired.

For any time interval the rainfall can be defined either as a total depth of rainfall or as an average intensity over the time interval. Entering either value automatically calculates and displays the other. The number of data pairs is automatically displayed in the top of the form and not every time interval need be entered.

When the [Optimize] button is clicked several pieces of information are displayed:

- The optimal values of the three parameters
- The computed values of Depth and Intensity for each time interval. These are shown in the right hand grid.
- A ‘log-log’ graph of both observed and computed values is displayed. A typical result is shown in Fig. 11-5 below

Data for different return periods (e.g. 5, 10 & 50 years) can be processed and the IDF graphs are accumulated on a single graph.

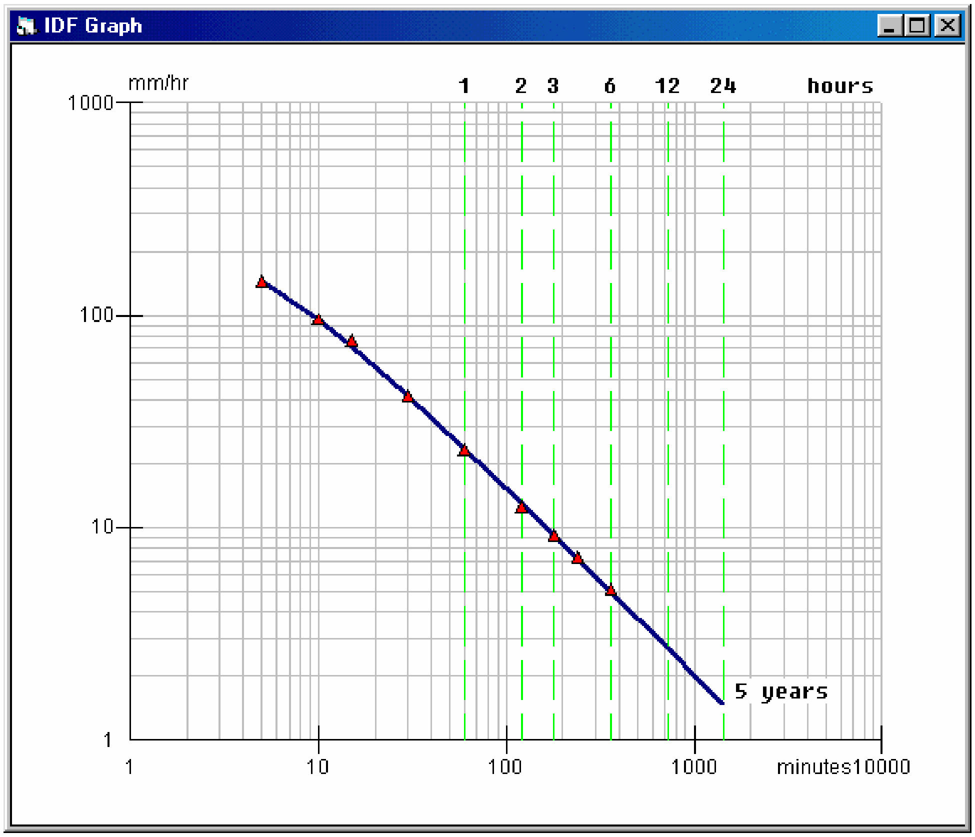


Figure 11-5 Optimized and observed rainfall intensity on a log-log plot.

By clearing the ‘Optimize best fit’ box the top command button is re-labeled [Calculate]. You can then enter values of ‘a’, ‘b’ and ‘c’ that differ from the optimized values previously calculated.

Clicking [Calculate] causes the computed values of depth and intensity to be revised and the blue curve on the graph is re-plotted.

Typically the error reported between observed and computed values of rainfall will be increased. This lets you experiment with parameter values for practical use.

The other command buttons on the form have the following uses.

Clear Storm	Clears all data for the current storm
Keep Storm	Store the data for the current storm and prepare for entry of another storm
Copy to Storm	Copy the optimized ‘a’, ‘b’, ‘c’ coefficients to the Storm/Chicago command
Cancel	Close the window without saving data
Accept	Close the window and save all data to the output file

11.4 Using the Time of Concentration tool

With few exceptions, peak runoff will occur when the entire catchment area is contributing to the outflow. Thus the storm duration should be long enough for the runoff from the most remote area – in terms of time of travel – to reach the outflow point. This is commonly referred to as the Time of Concentration ***Tc***

The time of concentration is calculated as the sum of up to three components of travel time. These are:

- Flood wave travel time of overland flow
- Travel time in relatively small collector channels or gutters
- Travel time in a storm conduit such as a circular pipe or a channel of general trapezoidal cross-section.

For the overland flow you can select one of two equations:

Friend's equation

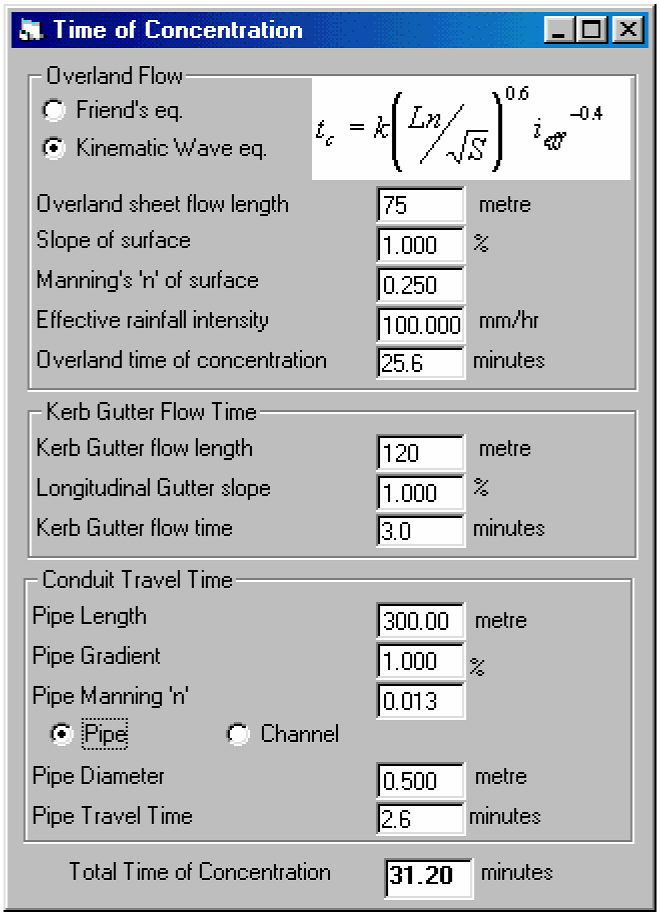
$$t_c = k\left(nL^{0.333} \big/ S^{0.2} \right)$$

where *k* = 107 (metric) or 72.042 (imperial)

Kinematic equation

$$t_c = k\left(Ln \big/ \sqrt{S} \right)^{0.6} i_{eff}^{\ -0.4}$$

where *k* = 6.989 (metric) or 0.939 (imperial)



Time of Concentration

Overland Flow

○ Friend's eq.

● Kinematic Wave eq.

$$t_c = k\left(\frac{Ln}{\sqrt{S}}\right)^{0.6} i_{eff}^{-0.4}$$

Overland sheet flow length	75	metre
Slope of surface	1.000	%
Manning's 'n' of surface	0.250	
Effective rainfall intensity	100.000	mm/hr
Overland time of concentration	25.6	minutes

Kerb Gutter Flow Time

Kerb Gutter flow length	120	metre
Longitudinal Gutter slope	1.000	%
Kerb Gutter flow time	3.0	minutes

Conduit Travel Time

Pipe Length	300.00	metre
Pipe Gradient	1.000	%
Pipe Manning 'n'	0.013	

● Pipe ○ Channel

Pipe Diameter	0.500	metre
Pipe Travel Time	2.6	minutes

Total Time of Concentration **31.20** minutes

Figure 11-6 Typical results using the Kinematic Wave equation

Each of the three components requires entry of data to describe the length, gradient and roughness of the conduit or surface. In addition, overland flow may also depend on the intensity of the effective rainfall.

On entry of a finite length, the time is computed for each component and the total is displayed as the Time of Concentration. If required, one or two of the flow components can be ignored by entering a zero length in the appropriate data field for length.

Either metric or U.S.Customary (Imperial) units can be used depending on the units selected in running MIDUSS

You will find that the Friend and Kinematic equations give significantly different answers for the same values of Overland flow length ***L***, Manning roughness ***n*** and slope ***S*** . If you decide to use the Friend eq. you should try to satisfy yourself that the answers seem appropriate. It is possible that the equation was based on data obtained during less extreme events than are appropriate for your geographical area of interest.

The Kinematic equation gives shorter times and also shows the effect of response being dependant on the magnitude of the inflow (i.e. rainfall intensity). This is a common characteristic of nonlinear systems.

Both equations use Manning's 'n' as a measure of surface roughness. Typical values for different types of surface are listed in the Table below.

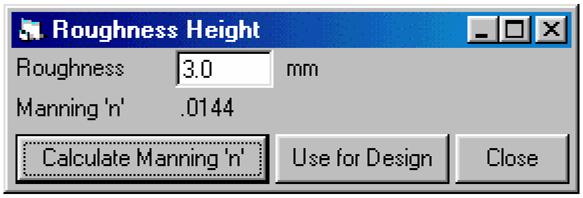
For the Kinematic eq. you will need to provide an estimate of the effective rainfall intensity. This should be an average value (over time).

Suggested values of Manning Roughness for Overland Flow.

Surface	**Suggested Manning 'n'**		
	Minimum	**Average**	**Maximum**
Paved surfaces	0.010	0.011	0.013
Sand, no vegetation	0.010	0.010	0.060
Clay-loam, no vegetation	0.012	0.020	0.033
Gravel	0.012	0.020	0.030
Short grass	0.10	0.15	0.20
Light turf		0.20	
Lawns	0.20	0.25	0.30
Dense turf		0.35	
Pasture	0.30	0.35	0.40
Dense shrubbery, forest litter		0.40	

11.5 Roughness Height

MIDUSS design routines use the Manning ‘n’ to describe surface roughness. Users who prefer to define roughness in terms of the equivalent roughness height can use the Roughness Height tool to convert from roughness height to Manning ‘n’. Once calculated, the computed value can be imported into the next design command by clicking the [Use for Design] button.



Roughness 3.0 mm

Manning 'n' .0144

Figure 11-7 Converting roughness height to Manning ‘n’

The conversion uses the following equation where g = gravitational acceleration and k = roughness height in the units shown in the form.

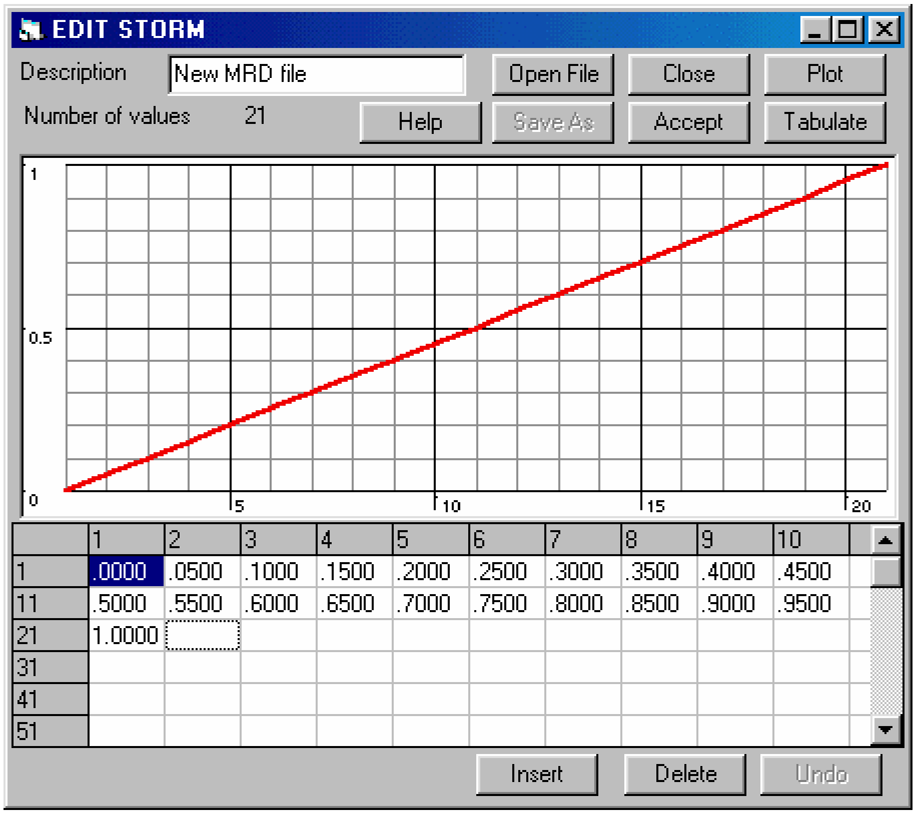
$$n = \frac{k^{1/6}}{8.41\sqrt{g}}$$

The equation is valid for fully developed, rough turbulent flow.

11.6 Editing a Storm

One of the options in the Storm command is to use a pre-defined curve known as a Mass Rainfall Distribution curve. These files are given the extension *.MRD and define the fraction of rainfall depth R(t)/ Rtot as a function of the ratio of elapsed time over total storm duration. Typical examples are the various Huff storm quartiles and the SCS hyetographs.

When the Edit Storm window opens it displays an *.MRD curve that is a straight line – i.e. the intensity is constant throughout the duration of the storm. You may use this as starting point for a new distribution but it is often more convenient to start with an existing *.MRD file and then modify it to suit. From the grid of data you will see that the first and last ordinate values must be 0.0 and 1.0 respectively for every *.MRD storm file.



Description: New MRD file

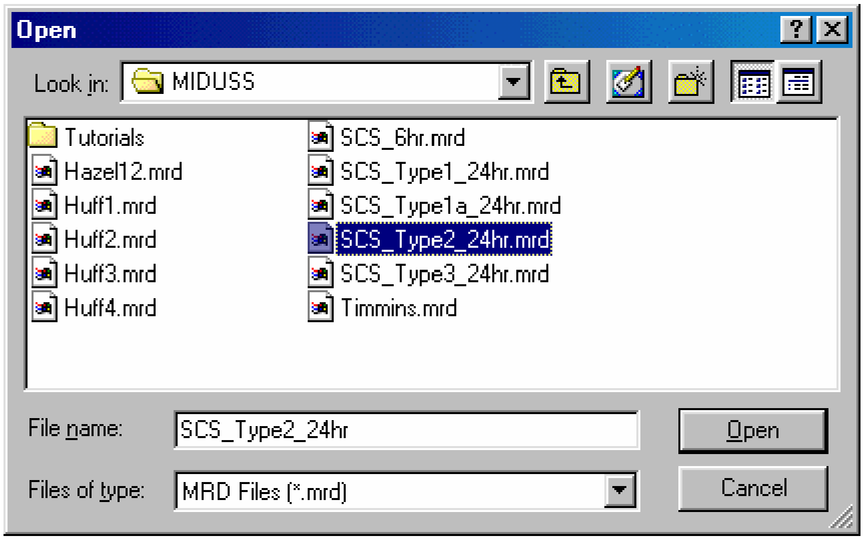
Number of values: 21

	1	2	3	4	5	6	7	8	9	10
1	.0000	.0500	.1000	.1500	.2000	.2500	.3000	.3500	.4000	.4500
11	.5000	.5500	.6000	.6500	.7000	.7500	.8000	.8500	.9000	.9500
21	1.0000									
31										
41										
51										

Figure 11-8 The initial Edit Storm screen

11.6.1 Loading an Existing Storm

By clicking the [Open File] button an Open File dialogue box is displayed as shown in Figure 11-9 below. The figure shows that the Type 2 SCS storm has been selected. Click [Open] to load this data into the Edit Storm window



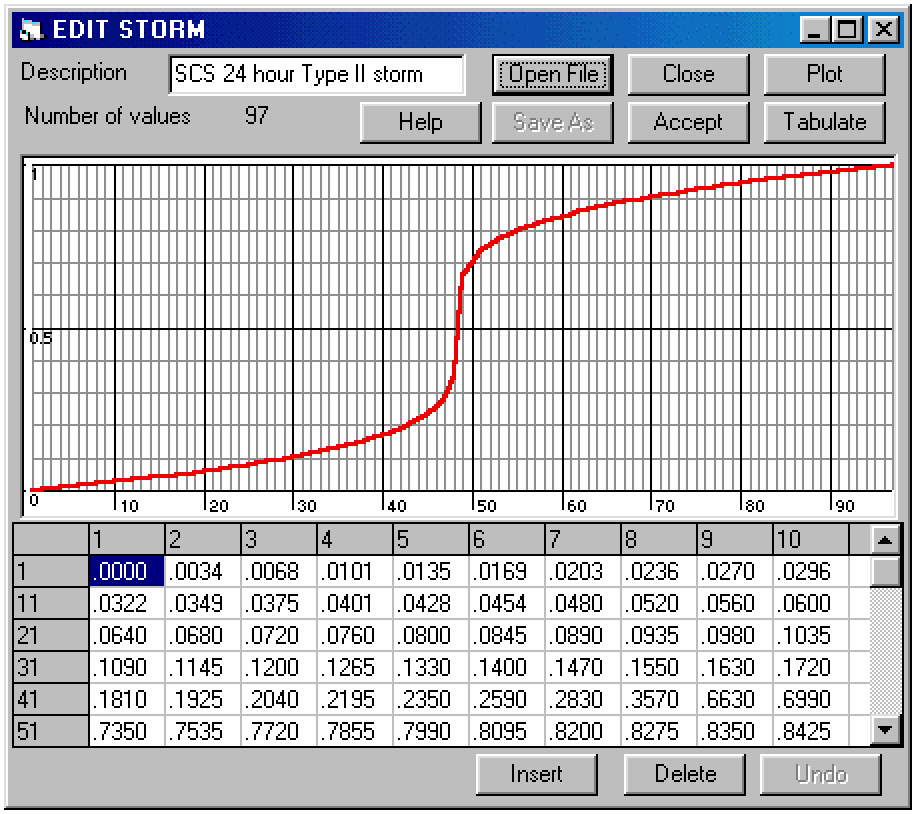
Look in: MIDUSS

File name: SCS_Type2_24hr

Files of type: MRD Files (*.mrd)

Figure 11-9 Selecting an existing storm file to load into the Edit Storm window

The result is shown in Figure 11-10 below. Because this file contains a large number of ordinate values (97) the data grid requires 10 rows which causes a vertical scroll bar to be added to the grid.



Description SCS 24 hour Type II storm

Number of values 97

	1	2	3	4	5	6	7	8	9	10
1	.0000	.0034	.0068	.0101	.0135	.0169	.0203	.0236	.0270	.0296
11	.0322	.0349	.0375	.0401	.0428	.0454	.0480	.0520	.0560	.0600
21	.0640	.0680	.0720	.0760	.0800	.0845	.0890	.0935	.0980	.1035
31	.1090	.1145	.1200	.1265	.1330	.1400	.1470	.1550	.1630	.1720
41	.1810	.1925	.2040	.2195	.2350	.2590	.2830	.3570	.6630	.6990
51	.7350	.7535	.7720	.7855	.7990	.8095	.8200	.8275	.8350	.8425

Figure 11-10 The SCS Type 2 storm is loaded into the Edit Storm window

11.6.2 Editing the Mass Rainfall Distribution Curve

The values can be edited in two ways – by graphical manipulation or numerically.

Graphical Edits	Position the mouse pointer on one of the vertical grid lines and either above or below the red line. Each mouse click causes the numerical value in the table to increase or decrease by 0.01 and the plotted red curve shows the change. Holding down the Shift key while clicking the mouse key increases the numerical change to 0.1.
Numerical Edits	Click on any cell in the grid with the exception of the 0.0 and 1.0 values and type in the desired value.

Any change in either the graphical or tabular display is reflected in the other. The array of values must start with zero and end with 1.0 and the intermediate values must increase monotonically. The steepness of an incremental line segment defines the intensity of the resultant storm so clearly a negative slope would imply a negative rainfall intensity.

11.6.3 Using the Command Buttons

A number of command buttons at the top of the Edit Storm form have the following functions.

[Open File]	Load an existing MRD file.
[Accept]	Accept the editing that you have done so far. This enables the [Save As] button
[Save As]	Lets you save the new MRD data as a file. The first two records in the file will contain a brief description and the total number of values including the 0 and 1 values. If you have modified an existing MRD file the description will be shown in the ‘Description’ label. You will be prompted to either accept the current description or change it.
[Plot]	Causes the current MRD data to be displayed as a hyetograph of relative rainfall intensities. This shows only the ‘shape’ of the storm and not actual intensities.
[Tabulate]	Displays a table of Mass Rain Intensity values for a total rainfall depth of 1 unit. The number of intensity values in the table is one less than the number of points in the MRD distribution.
[Insert]	Insert an additional element in front of the currently highlighted cell in the grid. The new point is given a value that is the mean of the values before and after it and the ‘Number of values’ is increased by one. If the first zero value is currently highlighted no action is taken. The graphical display is updated to show the change.
[Delete]	Removes the currently selected cell in the grid and reduces the number of values by one. The graph is updated.
[Undo]	Reverses the last use of either the [Insert] or [Delete] command button.