Chapter 9

Editing Junction, Reach, and Diversion Data

Table of Contents

Section **Page**

List of Figures

Figure Number **Page**

Chapter 9

Editing Junction, Reach, and Diversion Data

Once you have established the connectivity of your Reservoir Network (see Chapter 8), you can define and edit the properties of elements in the network. You will need to specify parameters for all junctions, reaches, diversions, and reservoirs using ResSim's specialized editors. You will also need to edit the physical and operations data for reservoirs, as described in Chapters 10-11.

This chapter explains how to enter and edit data for junctions, reaches, and diversions. Refer to Chapter 8 for developing a Reservoir Network within the **Reservoir Network Module**.

9.1 Editing Junction Properties

To edit junction data, select the **Junction Tool**  and right-click on a junction. From the shortcut menu, select **Edit Junction Properties**. The **Junction Editor** will open.

The **Name** field at the top of the editor contains a list of all of the junctions in your reservoir network, with the name of the selected junction displayed. You can access all of the junctions both from this list and by using the VCR-style buttons to navigate through the available junctions.

The Junction Editor has four tabs that allow you to edit junction information (**Info**), **Local Flow**, **Rating Curve**, and **Observed Data**, as described in the following sections.

9.1.1 Junction Editor: Info Tab

The **Info** tab (Figure 9.1) displays the description, stream station, stream, and tributaries you have defined within the Watershed Setup Module and during the process of creating your reservoir network within the Reservoir Network Module.

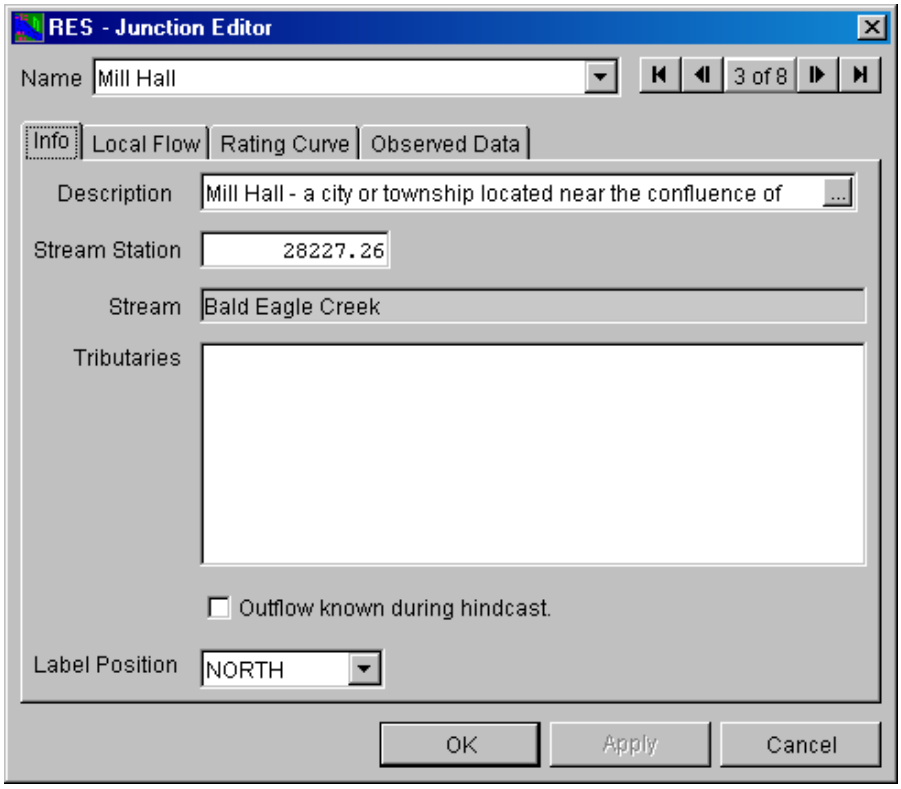

Figure 9.1 Junction Editor: Info Tab

The **Stream** and **Tributaries** fields are not editable, but you may revise the junction **Description** and **Stream Station**. If the historic flow for a junction is available for the hindcast (lookback) period, then click on the **Outflow known during hindcast** box. An entry will appear in the time-series table that you will complete during the creation of an Alternative (see Chapter 13).

You may also choose the position of the name label in your map display from the options in the **Label Position** field.

To edit properties for other junctions without exiting the Junction Editor, click the **Apply** button and then select another junction from the **Name** list or use the navigator buttons.

9.1.2 Junction Editor: Local Flow Tab

On the **Local Flow** tab (Figure 9.2) of the Junction Editor, you can input a **Name** and **Factor** in the table for each external flow entering the junction.

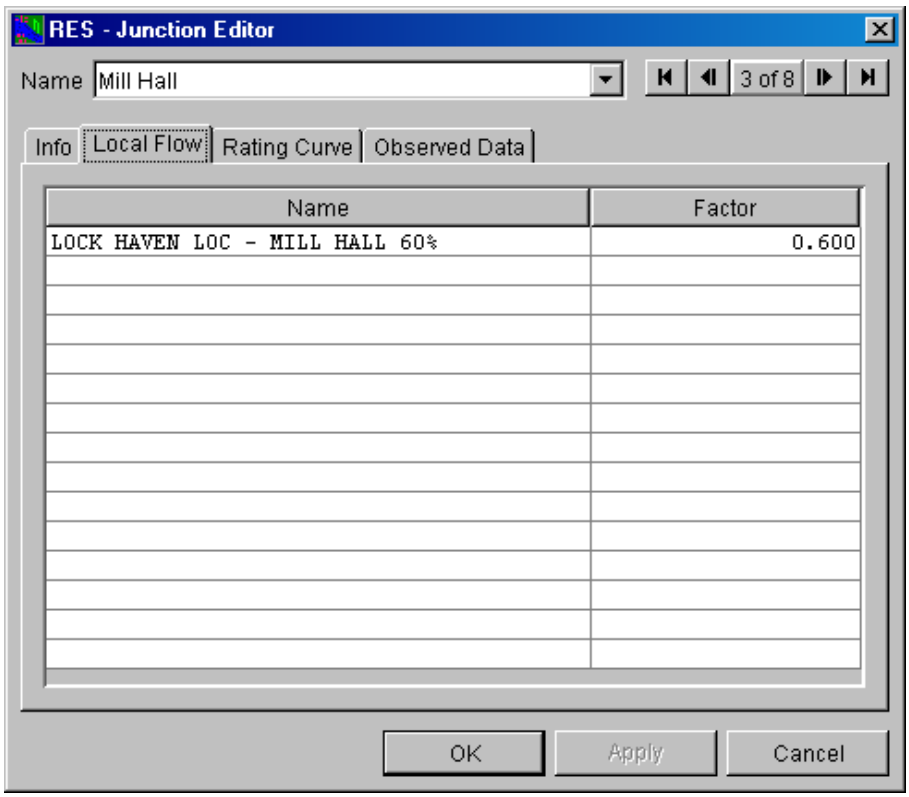

Figure 9.2 Junction Editor: Local Flow Tab

- **Name:** The name you enter will appear during the Alternative definition to identify the local flow for time-series mapping (see Chapter 13).
- **Factor:** The factor value is available to increase or decrease the local flow hydrographs. If you enter a value of 1.0, then the local flow will be equal to the hydrograph reference.

You must define flows for headwater junctions. Flows are optional at other junctions.

To edit properties for other junctions without exiting the Junction Editor, click the **Apply** button and then select another junction from the **Name** list or use the navigator buttons.

9.1.3 Junction Editor: Rating Curve Tab

The **Rating Curve** tab (Figure 9.3) is optional. However, if you need stages calculated at a junction, you can enter **Stage** and **Discharge** values for the junction here. The mini-plot will reflect the values you enter and can be viewed in full size when you double-click on it.

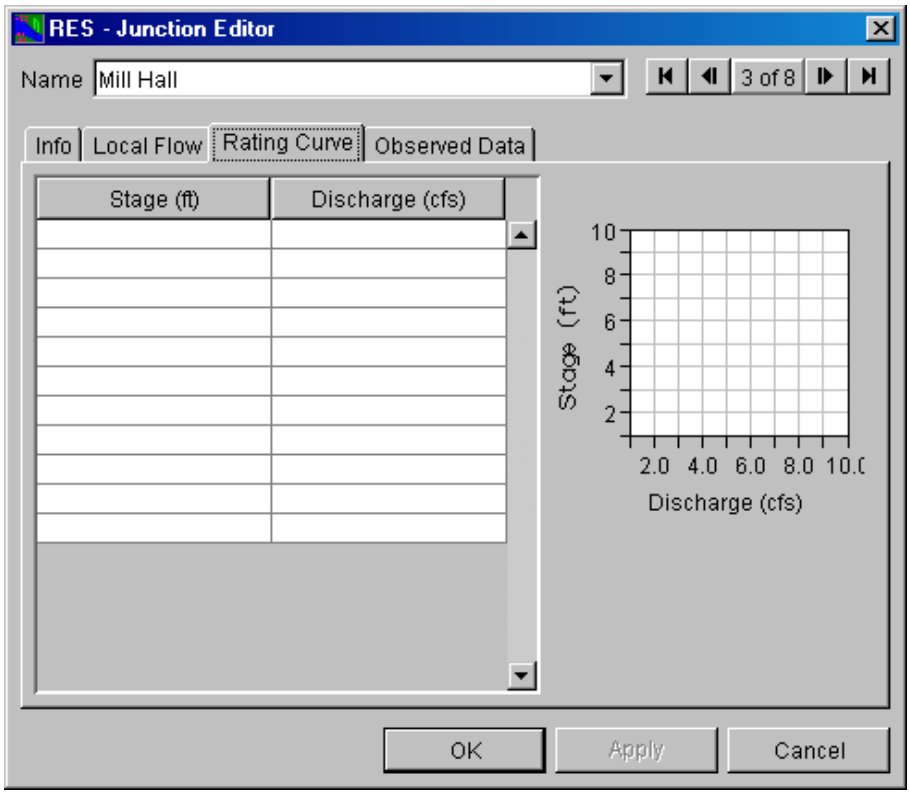

Figure 9.3 Junction Editor: Rating Curve Tab

- **Stage:** Enter Stage values (in meters or feet) that correspond to the Discharge values.
- **Discharge:** Enter Discharge (flow) values (in cms or cfs) that correspond to the Stage values.

To edit properties for other junctions without exiting the Junction Editor, click the **Apply** button and then select another junction from the **Name** list or use the navigator buttons.

9.1.4 Junction Editor: Observed Data Tab

Use the **Observed Data** tab (Figure 9.4) to indicate that observed data is available for comparison purposes. If the **Observed** box in the table is checked (as shown in Figure 9.4), then there will be a corresponding entry in the Observed Time-Series mapping table when you create an Alternative (see Chapter 13).

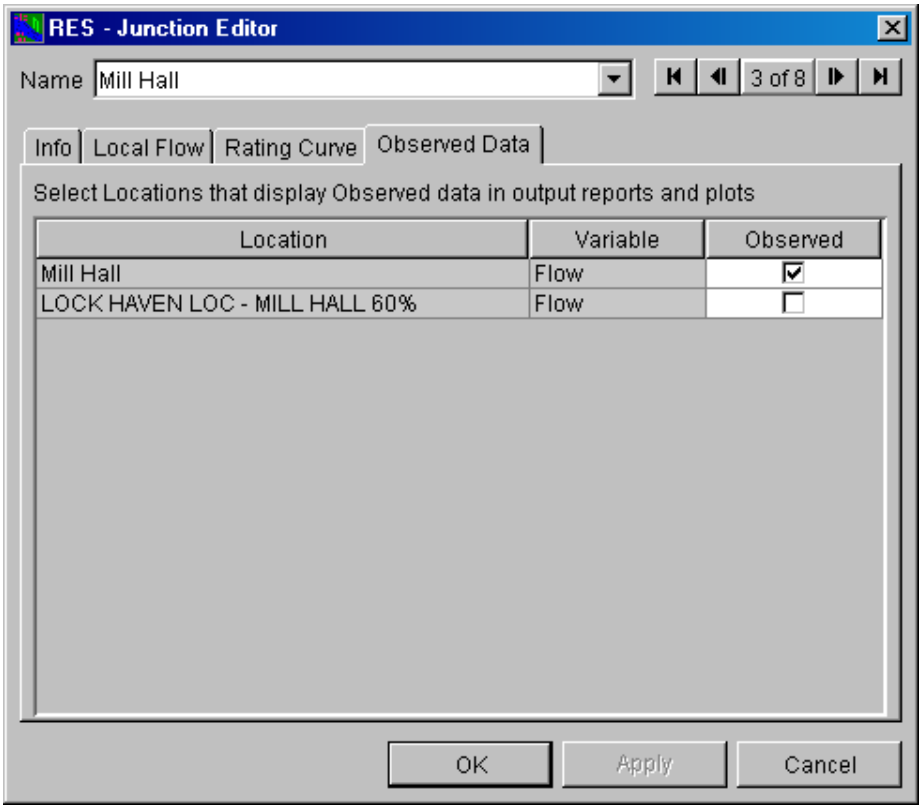

Figure 9.4 Junction Editor: Observed Data Tab

To edit properties for other junctions without exiting the Junction Editor, click the **Apply** button and then select another junction from the **Name** list or use the navigator buttons.

9.2 Editing Reach Properties

To edit routing reach data, select the **Reach Tool**  and right-click on a routing reach. From the shortcut menu, select **Edit Reach Properties**. The **Reach Editor** will open (Figure 9.5).

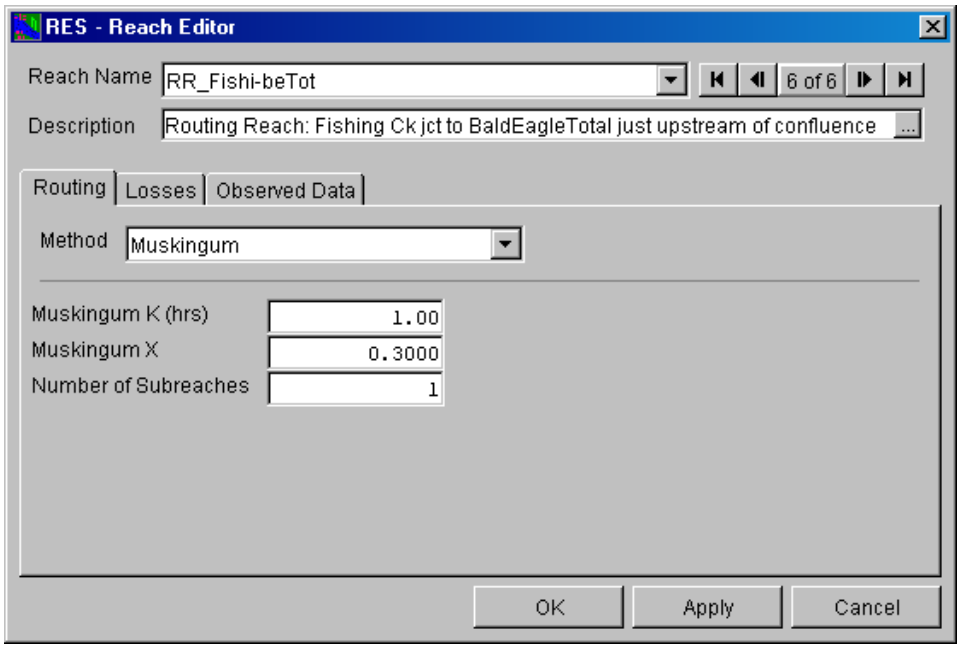

Figure 9.5 Reach Editor

The **Reach Name** field at the top of the editor contains a list of all of the routing reaches in your reservoir network, with the name of the selected reach displayed. You can access all of the reaches from this list or by using the VCR-style buttons to navigate through the available reaches.

Below the Reach Name list is the **Description** of the current reach; this field is editable.

The Reach Editor has three tabs that allow you to edit **Routing** methods, **Losses**, and **Observed Data**, as described in the following sections.

9.2.1 Reach Editor: Routing Tab

The Reach Editor's **Routing** tab allows you to choose from six routing methods: Coefficient Routing, Muskingum, Muskingum-Cunge 8-pt Channel, Muskingum-Cunge Prismatic Channel, Modified Puls, and SSARR Routing, each method with its own set of parameters. If there is no lag or translation of the hydrograph through the reach, you can select **Null Routing**. Select the routing method for the reach from the **Method** list. The appropriate fields for data entry will appear according to the method you have selected.

9.2.1.1 Coefficient Routing Method

For the **Coefficient Routing** method (Figure 9.6), you will need to enter coefficients for each time step. Each coefficient equates to the fraction of the flow entering the reach that will reach the downstream end at the end of each time step in the table. The values in the table must sum up to 1.

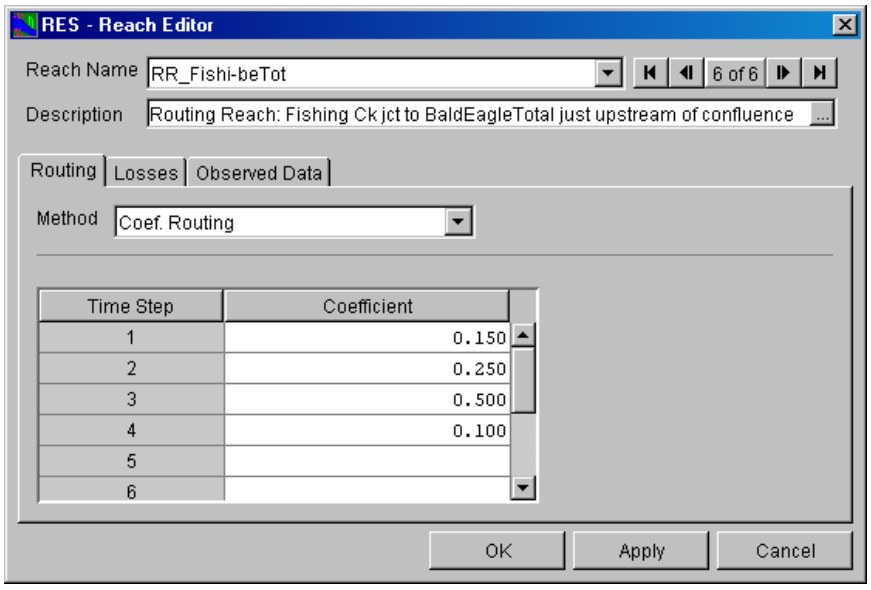

Figure 9.6 Reach Editor: Coefficient Routing Method

If you enter a 1 in the first row and leave the rest of the table blank, you have effectively described null or no-route routing. This means that 100% of the flow entering the reach instantaneously leaves the reach in that time step.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.1.2 Muskingum Routing Method

The **Muskingum** routing method (Figure 9.7) requires you to enter values for **Muskingum K** (in hours) and **Muskingum X**. You will also need to specify the **Number of Subreaches**.

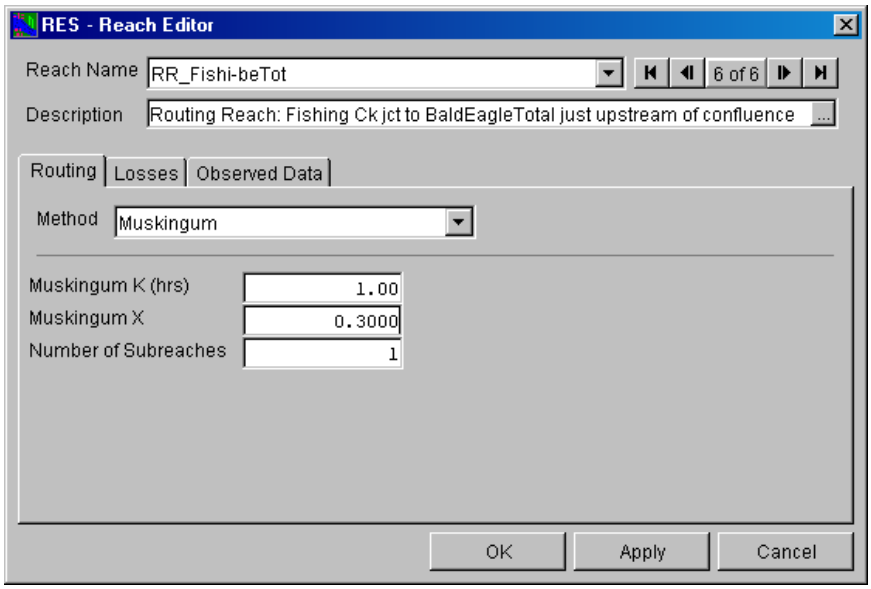

Figure 9.7 Reach Editor: Muskingum Routing Method

- **Muskingum K**: Enter the number of hours (travel time) for the reach, *i.e.*, how long it takes for a drop of water that enters the reach to exit the reach.
- **Muskingum X:** Enter the attenuation coefficient (from 0.0 to 0.5). A value of 0.0 indicates maximum attenuation of the hydrograph through the routing reach. A value of 0.5 indicates a "direct translation" of the hydrograph through the reach.
- **Number of Subreaches:** Number of steps (subreaches) applicable for the routing reach. This parameter should be approximately equal to travel time divided by the computation interval.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.1.3 Muskingum-Cunge 8-pt Channel Routing Method

The **Muskingum-Cunge 8-pt Channel** routing method (Figure 9.8) uses a representative eight-point channel cross section to perform the Muskingum-Cunge hydrologic routing method. This method requires that you enter the **Channel Length** and **Channel Slope**, as well as eight **Station** and **Elevation** values to describe the cross section. The cross section displays in both a generalized illustration of a cross section and a mini-plot of the data you enter and can be viewed in full size when you double-click on it. Other parameters you will need to enter are **Manning's n Values** for the **Left Overbank**, **Main Channel**, and **Right Overbank**. The **Default Reference Flow** value is used to compute the celerity for the reach.

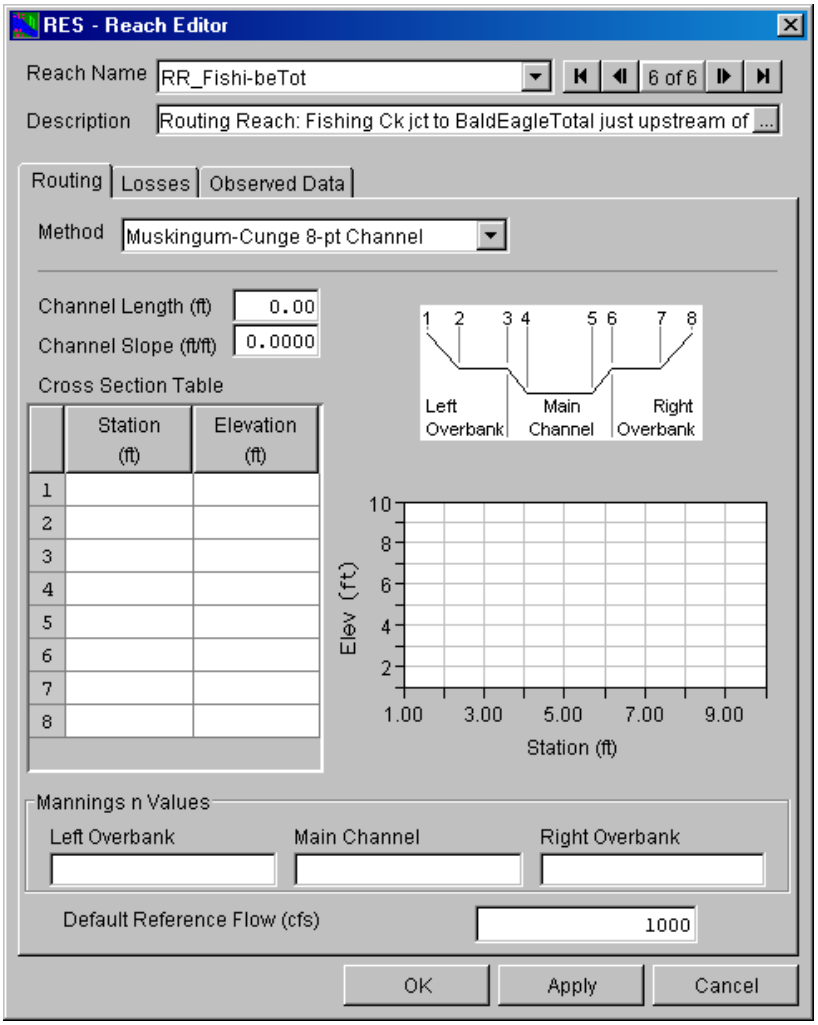

Figure 9.8 Reach Editor: Muskingum-Cunge 8-pt Channel Routing Method

- **Channel Length:** Enter the length of the routing reach.
- **Channel Slope:** Enter the slope of the routing reach, $\Delta y/\Delta x$.
- **Station:** Enter the horizontal position of a point used to describe a stream cross section.
- **Elevation:** Enter the vertical position of a point used to describe a stream cross section.
- **Manning's n Values:** Enter the unitless loss coefficients for **Left Overbank**, **Main Channel**, and **Right Overbank** to be used in the normal depth calculations of the cross section.
- **Default Reference Flow:** The flow value is used to compute the celerity (and thus the travel time) for the reach. If the travel time is less than the time-step size, the reach is divided into subreaches for the computation. Travel time (and thus the number of subreaches) is greater for low flow; therefore, the reference flow value should approximate the base flow for the reach.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.1.4 Muskingum-Cunge Prismatic Channel Routing Method

The **Muskingum-Cunge Prismatic Channel** method (Figure 9.9) provides two more options for describing the cross section to be used by the Muskingum-Cunge hydrologic routing method. In this method, you can describe the cross section as either a trapezoid or a circle.

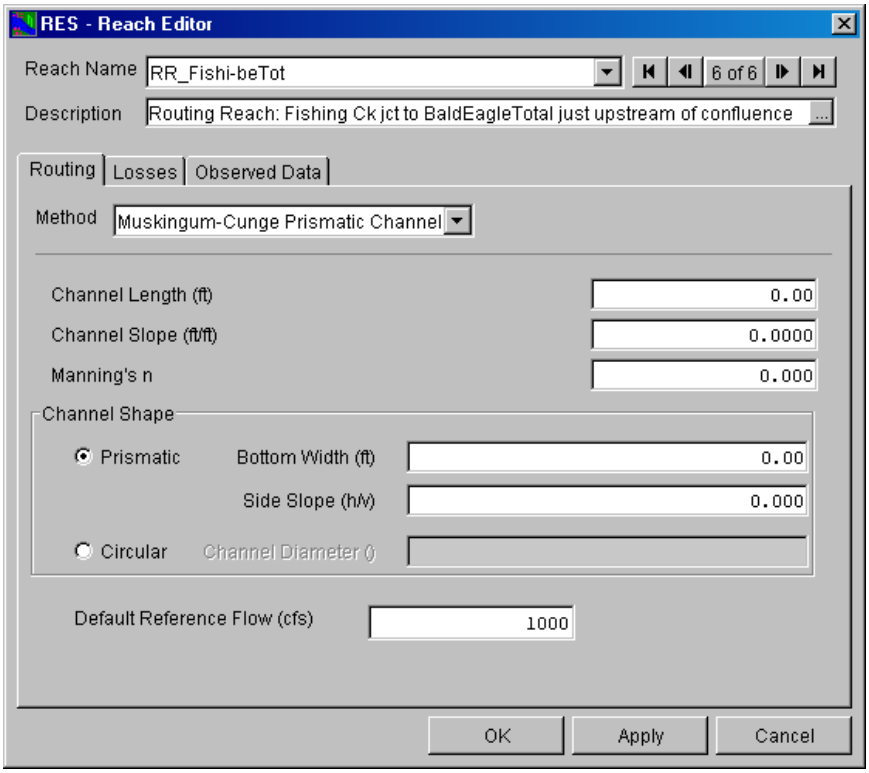

Figure 9.9 Reach Editor: Muskingum-Cunge Prismatic Channel Routing Method

For the Muskingum-Cunge Prismatic Channel method, enter the appropriate parameters, described below:

- **Channel Length:** Enter the length of the routing reach.
- **Channel Slope:** Enter the slope of the routing reach, $\Delta y/\Delta x$.
- **Manning's n:** Enter the unitless loss coefficient to be used in the normal depth calculations of the cross section.

- **Channel Shape:** Select either Prismatic (trapezoid) or Circular (pipe).
  - **Prismatic:** This shape is a basic trapezoid with the bottom of the trapezoid less than or equal to the width of the top. When you choose Prismatic as the Channel Shape, you also need to specify the Bottom Width and Side Slope.
    - **Bottom Width:** the width of the bottom of the trapezoidal section.
    - **Side Slope:** the slope of the sides of the trapezoid entered as $\Delta x/\Delta y$.
  - **Circular:** This shape is a simple circle used to represent a pipe or culvert. This method is valid for flow depth up to 0.77*diameter. For depths greater than this, the depth used by the method is held to 0.77*diameter and a warning message is generated. If you choose Circular as the Channel Shape, you also need to specify the Channel Diameter.
    - **Channel Diameter:** Define the size of the circle.

- **Default Reference Flow:** The flow value is used to compute the celerity (and thus the travel time) for the reach. If the travel time is less than the time step size, the reach is divided into subreaches for the computation. Travel time (and thus the number of subreaches) is greater for low flow; therefore, the reference flow value should approximate the base flow for the reach.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.1.5 Modified Puls Routing Method

The **Modified Puls** channel routing method (Figure 9.10) describes the reach as a set of cascading reservoirs, the number of which is specified by the number of subreaches. The outflow from the reach is defined as a function of the storage. An option is available within this method to account for seepage losses within the routing computation. To use this option, check the **With Channel Losses** box and enter the average **Invert Elevation** of the reach, the **Percolation Rate** to describe the seepage, and extend the outflow-storage table by adding associated elevation data.

*If you use the **Channel Losses** option in this method, do not define seepage losses on the losses tab for this reach – you will be double accounting for the seepage losses.*

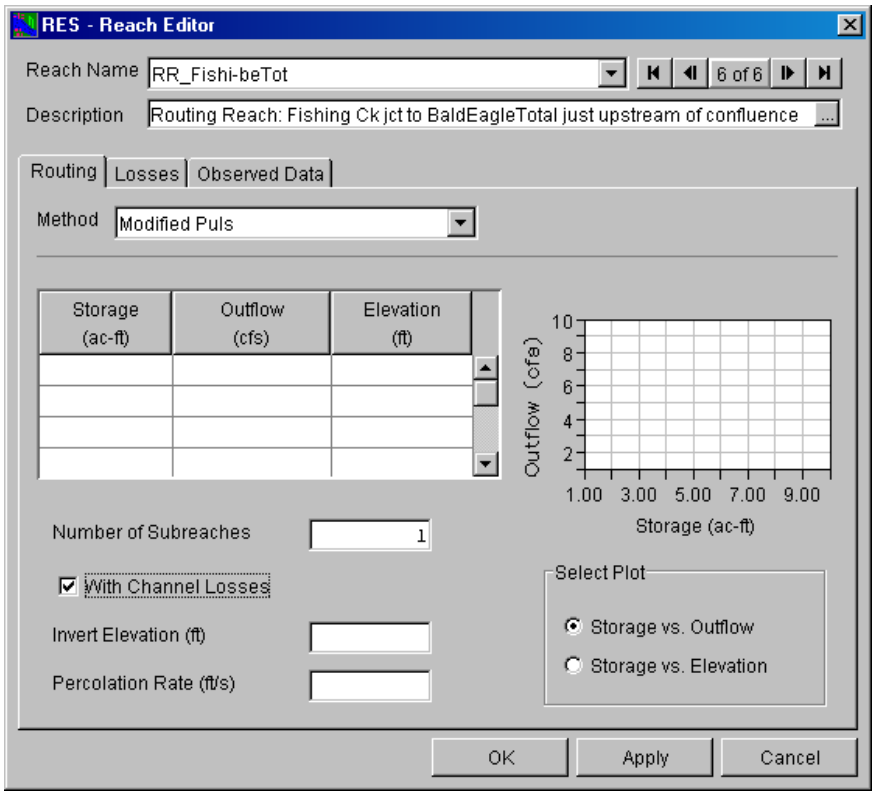

Figure 9.10 Reach Editor: Modified Puls Routing Method

- **Storage:** Enter monotonically increasing values of storage to describe the storage-outflow relationship needed for this routing method.
- **Outflow:** Enter the outflow values associated with each storage value in the storage-outflow relationship.

- **Elevation:** Enter associated elevation values; ResSim uses these values as extensions to the storage-outflow relationship when you select the **With Channel Losses** option. This column is only available when this option is active.
- **Number of Subreaches:** This parameter is similar to that used in the Muskingum routing method. Enter the reach travel time divided by the computation interval.
- **Invert Elevation:** Enter the average minimum elevation of the channel in this reach.
- **Percolation Rate:** Enter the flow rate identifying the loss of water to the soil/groundwater.
- In the **Select Plot** area, you can choose whether the plot shows **Storage vs. Outflow** or **Storage vs. Elevation**. The plot will reflect the values you enter.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.1.6 SSARR Routing Method

The SSARR routing method (Figure 9.10) describes the channel routing based on the computer program *Streamflow Synthesis & Reservoir Regulation (SSARR)*, from the Corps' Northwestern Division (USACE, 1991).

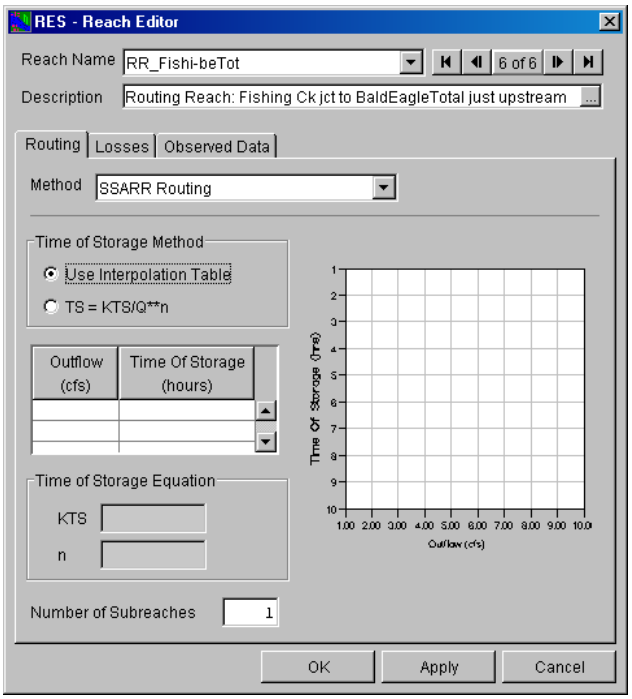

Figure 9.11 Reach Editor: SSARR Routing Method

The storage in the routing reach is defined by Time of Storage values (T_s), in units of hours. Therefore, the storage is defined like Muskingum K values (described in Section 9.2.1.2). Instead of using Modified Puls storage-outflow, this method uses T_s versus outflow.

An alternative method for defining T_s is by the following equation, which defines T_s as a power function of flow.

$$T_s = \frac{KTS}{Q^n}$$

where:

T_s = Time of storage per increment in hours

KTS = Constant determined by trial and error or estimated from physical measurements of flow and corresponding routing times.

Q = Discharge in cubic meters (or feet) per hour

n = Coefficient usually between -1 and 1

As evident from the above equation, T_s is a nonlinear function of discharge except when n = 1. It is possible to use a negative value of n if time of storage increases as discharge increases. According to the SSARR User's Manual, a value of n = 0.2 is reasonable for most streams in the Columbia River Basin.

For the SSARR routing method, enter the appropriate parameters, described below:

- **Time of Storage Method:** Select either the **Use Interpolation Table** or the **TS = KTS/Q**n** method.
- **Outflow** and **Time of Storage:** If the Use Interpolation Table method is selected, enter the outflow and corresponding T_s values.
- **Time of Storage Equation:** If the TS = KTS/Q**n method is selected, enter values for KTS and n.
- **Number of Subreaches:** Number of steps applicable for the routing reach.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.2 Reach Editor: Losses Tab

The **Losses** tab of the Reach Editor (Figure 9.12) provides two options for computing losses in the routing reach: **Constant Seepage** or **Seepage as a function of Flow**. The default is no losses in the reach.

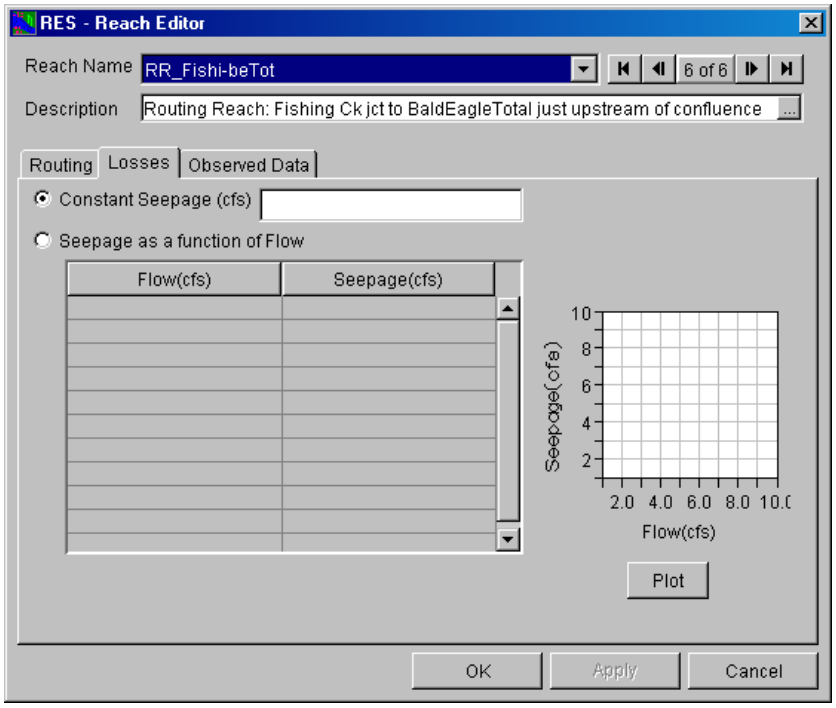

Figure 9.12 Reach Editor: Losses Tab

- **Constant Seepage:** Enter the flow (in cms or cfs) lost to the reach through the soil, non-varying throughout the simulation.
- **Seepage as a function of Flow:** With this option, you can define the loss as a function of the routed flow of the reach. Reach losses are computed after the flow is routed through the reach; therefore, the final outflow from the reach is the routed flow minus the loss.
- **Flow:** Enter a monotonically increasing set of flow values to define the relationship of seepage with respect to flow.
- **Seepage:** Enter the associated seepage values for each flow value in the table, expressed in flow units.

The mini-plot will reflect the values you enter and can be viewed in full size when you double-click on it.

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.2.3 Reach Editor: Observed Data Tab

Use the **Observed Data** tab (Figure 9.13) to indicate that observed data is available for comparison purposes. If the **Observed** box in the table is checked (as shown in Figure 9.13), then there will be a corresponding entry in the Observed Time-Series mapping table when you create an Alternative (see Chapter 13).

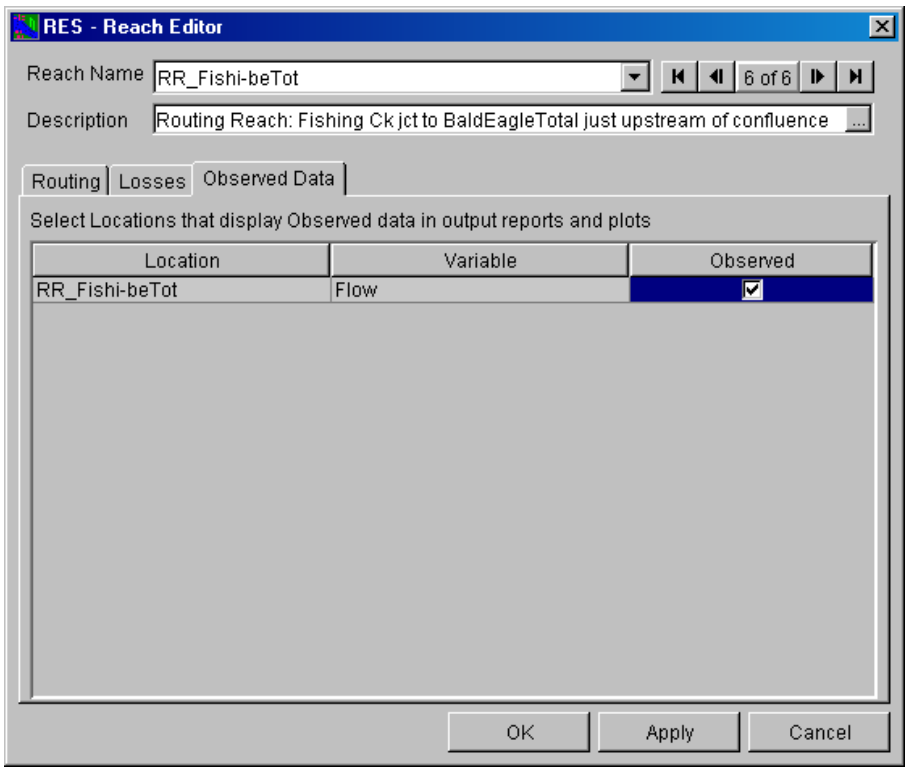

Figure 9.13 Reach Editor: Observed Data Tab

To edit another reach without exiting the Reach Editor, click the **Apply** button and then select another reach from the **Name** list or use the navigator buttons.

9.3 Editing Diversion Properties

To edit diversion data, select the **Diversion Tool**  and right-click on a diversion. From the shortcut menu, select **Edit Diversion Properties**. The **Diversion Editor** will open (Figure 9.14).

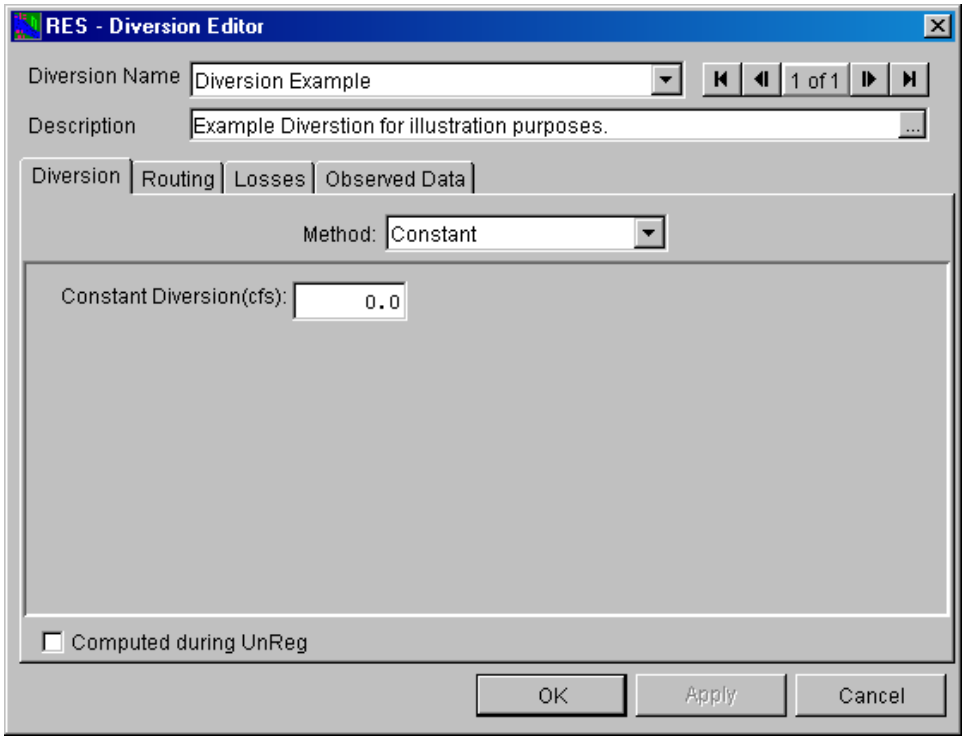

Figure 9.14 Diversion Editor

The **Name** field at the top of the Editor contains a list of all of the diversions in your reservoir network, with the name of the current diversion displayed. You can access all of the diversions both from this list and by using the VCR-style buttons to navigate through the available diversions.

If the **Computed during UnReg** box is checked, then the diversion will operate during the unregulated flow calculation. If the box is left unchecked, then the unregulated flow will reflect flow conditions without the diversion.

The Diversion Editor has four tabs that allow you to edit diversion information (**Diversion** tab), **Losses**, **Routing**, and **Observed Data**, as described in the following sections.

Note that the **Losses** and **Routing** tabs are available only for diversions that connect to downstream nodes.

9.3.1 Diversion Editor: Diversion Tab

The Diversion Editor's **Diversion** tab allows you to choose from six methods: Constant, Monthly Varying, Seasonal, Function of Flow, Function of Pool Elevation, and Time Series, each with its own parameters. The default method is **Constant**.

Select the diversion method from the **Method** list. The appropriate editing fields will appear according to the method you have selected.

9.3.1.1 Constant Diversion Method

For the **Constant** method (Figure 9.15), you will need to enter a diversion flow value (in the appropriate units of your watershed). This constant value will apply to each period of your simulation.

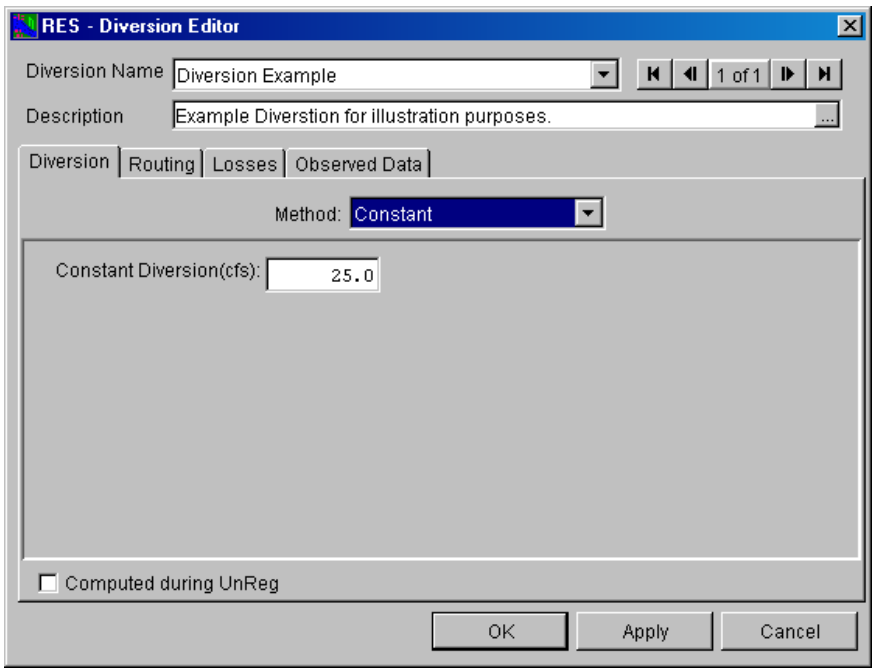

Figure 9.15 Diversion Editor: Constant Diversion Method

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.1.2 Monthly Varying Diversion Method

For the **Monthly Varying** diversion method (Figure 9.16), you will need to enter the diversion flow value (in the appropriate units of your watershed) corresponding to each month of a year. These monthly values will apply to each year in your simulation.

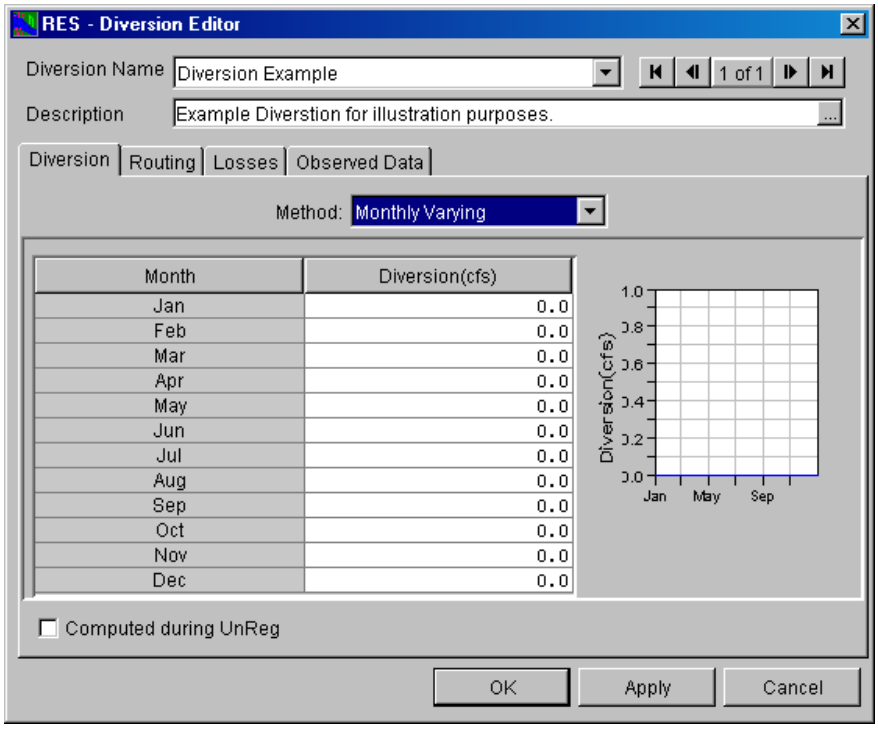

Figure 9.16 Diversion Editor: Monthly Varying Diversion Method

- **Diversion:** Enter a flow value (in cms or cfs) for each month.

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.1.3 Seasonal Diversion Method

For the **Seasonal** method (Figure 9.17), you will need to enter a **Date** and **Time** along with a corresponding **Diversion** flow value (in the appropriate units of your watershed). The seasonal diversion is applicable for each year of your simulation.

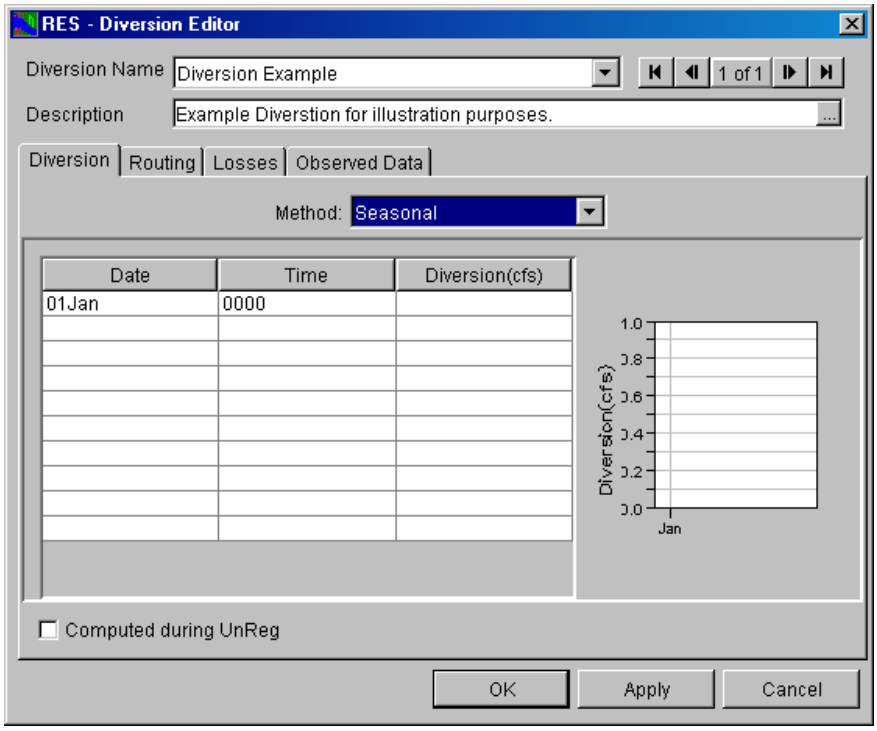

Figure 9.17 Diversion Editor: Seasonal Diversion Method

- **Date:** Enter the Day and Month in the format ddMMM (e.g., 01Jan).
- **Time:** Enter the Hour in the range of 0000–2400.
- **Diversion:** Enter a Flow value in cms or cfs.

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.1.4 Function of Flow Diversion Method

For the **Function of Flow** diversion method (Figure 9.18), you will need to enter a relationship between a junction's net flow and the resulting diverted flow (in the appropriate units of your watershed). The **Flow Location** junction can be selected from the list and need not be the same location as where the diversion occurs. This allows for a diversion to limit or augment flow based on any junction's flow. For a given net **Flow** value entering the diversion element, the **Diversion** flow will be removed from the junction. This relationship applies to your entire Simulation period.

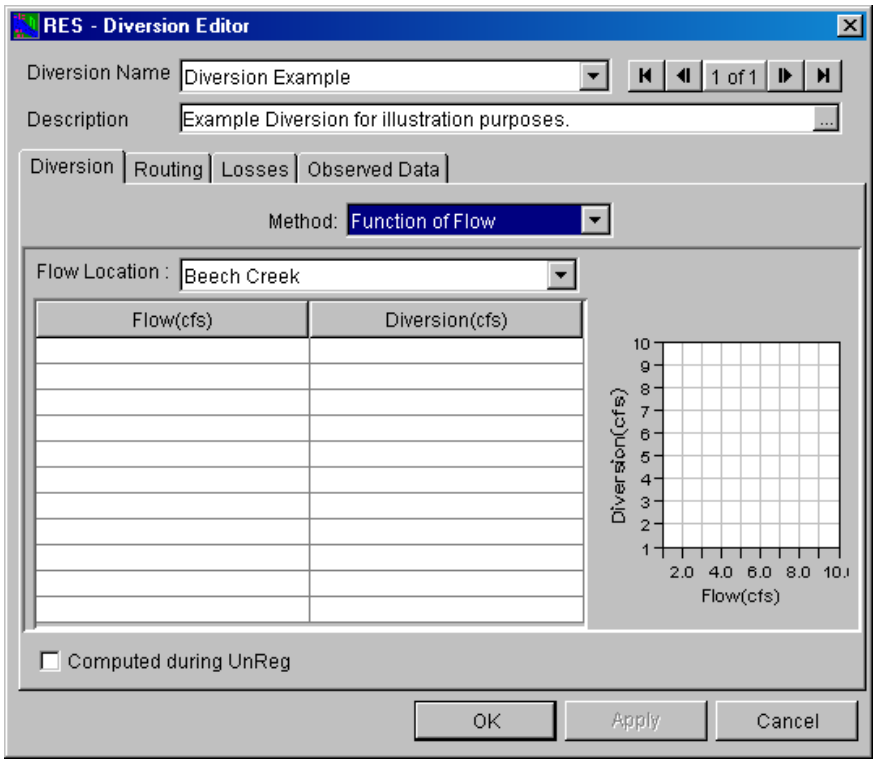

Figure 9.18 Diversion Editor: Function of Flow Diversion Method

- **Flow:** Enter the net flow (in cms or cfs) entering the junction.
- **Diversion:** Enter the flow (in cms or cfs) that is diverted from the junction.

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.1.5 Function of Pool Elevation Diversion Method

For the **Function of Pool Elevation** diversion method (Figure 9.19), you will need to enter a relationship between reservoir elevation and diversion flow (in the appropriate units of your watershed). Also, you will need to select the appropriate reservoir from the **Reservoir** list. When a given **Elevation** at the selected reservoir occurs, then the corresponding **Diversion** flow will be diverted from the junction. Linear interpolation will be made between the elevations (and corresponding diversions) that you enter. This relationship applies to your entire simulation.

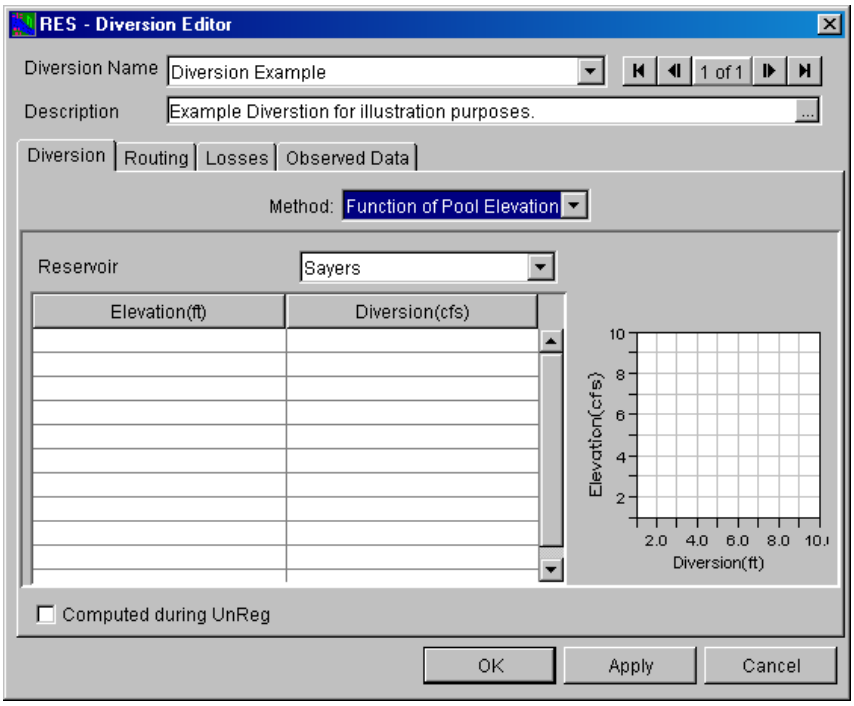

Figure 9.19 Diversion Editor: Function of Pool Elevation Diversion Method

- **Elevation:** Enter the reservoir elevation (in meters or feet) on which the diversion is based.
- **Diversion:** Enter the flow (in cms or cfs) that is diverted from the diversion element.

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.1.6 Time-Series Diversion Method

For the **Time-Series** diversion method (Figure 9.20), you only need to select **Time Series** from the **Method** list. An entry for this diversion will appear in the Time-Series mapping table when you create an Alternative (see Chapter 13). The time series of diversion flow values must be in the appropriate units (cms or cfs) of your watershed.

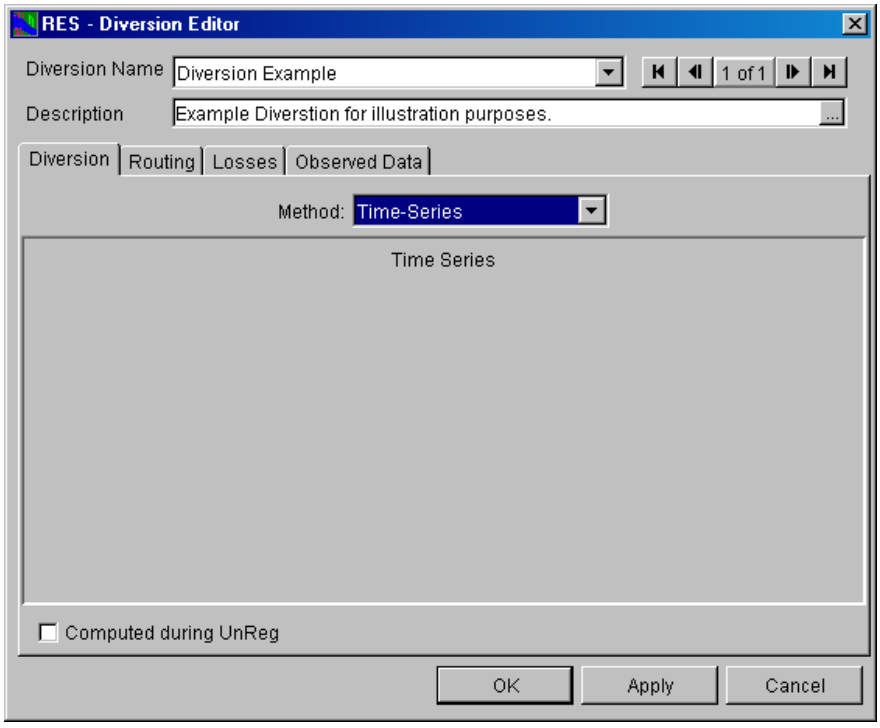

Figure 9.20 Diversion Editor: Time-Series Diversion Method

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.2 Diversion Editor: Routing Tab

For diversions connected to downstream junctions, the Diversion Editor's **Routing** tab allows you to choose from six routing methods: Coefficient Routing, Muskingum, Muskingum-Cunge 8-pt Channel, Muskingum-Cunge Prismatic Channel, Modified Puls, and SSARR routing, each with its own set of parameters. If there is no lag or translation of the hydrograph through the diversion reach, you can select Null Routing (which is the default routing method).

The Routing tab of the Diversion Editor is identical to the Routing tab of the Reach Editor. For details about using the Routing tab, refer to Section 9.2.1.

9.3.3 Diversion Editor: Losses Tab

Available for diversions connected to a downstream junction, the **Losses** tab of the Diversion Editor (Figure 9.21) provides three options for computing losses in the diversion's routing reach: **Return Ratio**, **Constant Seepage**, or **Seepage as a function of Flow**.

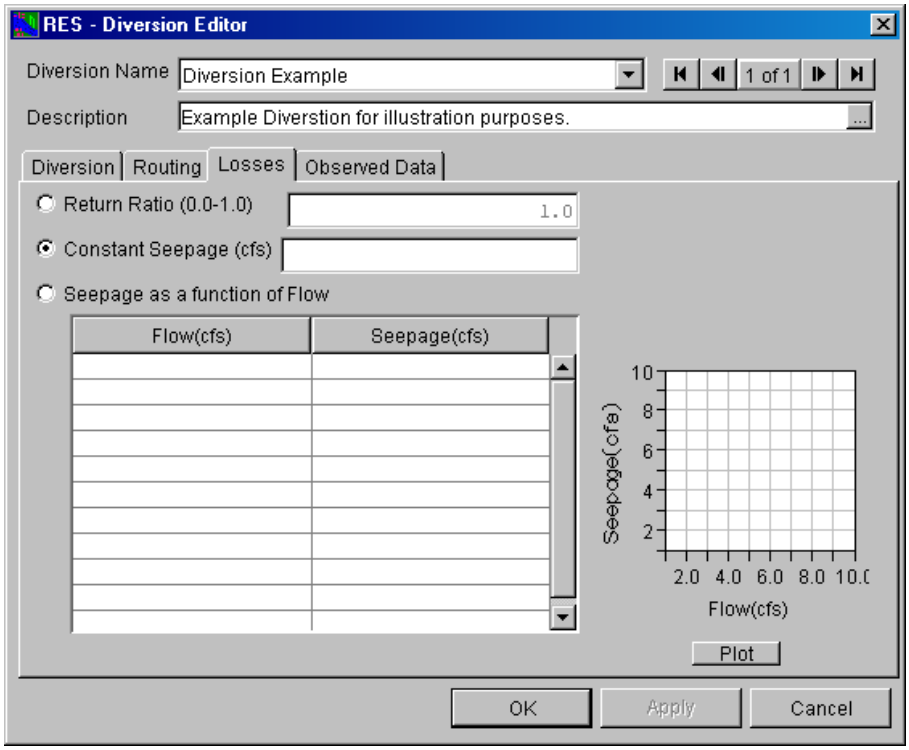

Figure 9.21 Diversion Editor: Losses Tab

- **Return Ratio:** Enter a value between 0.0 and 1.0 (the default is 1.0) to indicate how much diversion continues through the diversion to the downstream junction.

- **Constant Seepage:** Enter a seepage value (in cms or cfs) to indicate how much diversion flow will "seep into" the ground and therefore be lost from the system.
- **Seepage as a function of Flow:** Enter a Flow vs. Seepage relationship (in cms or cfs) to indicate how much of the diversion flow is lost from the diversion to the ground.

The mini-plot will reflect the values you enter and can be viewed in full size when you double-click on it.

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.

9.3.4 Diversion Editor: Observed Data Tab

Use the **Observed Data** tab (Figure 9.22) to indicate that observed data is available for comparison purposes. If the **Observed** box in the table is checked, then there will be a corresponding entry in the Observed Time-Series mapping table when you create an Alternative (see Chapter 13).

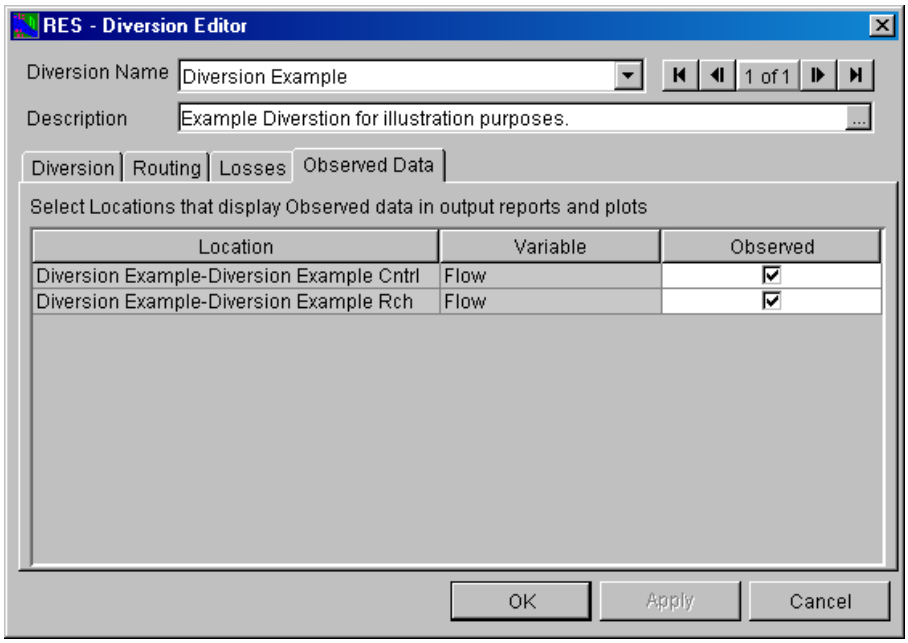

Figure 9.22 Diversion Editor: Observed Data Tab

To edit another diversion without exiting the Diversion Editor, click the **Apply** button and then select another diversion from the **Diversion Name** list or use the navigator buttons.