# Tutorial WaterElements

Inhaltsverzeichnis

# User Interface



After starting the program you see the main window with the following topics:

- **Input:** is divided into three areas:

  ○ At the top right the **command editor** for input of commands which are executed by the program

  ○ At the top left a **tree of all commands** that can be selected (the button „H" hides the tree. Click it again to show the tree)

  ○ at the bottom in the **control window** are the messages which appear after clicking the button "Execute Commands" or the menu item "Run"

- **Network:** graphical user interface for creation of elements and the network. Further details can be found below (see **Note**)

- **Results:** summary of simulation results in text form is shown here

- **Structures:** table of defined structures and their characteristics (serves as input control)

- **Overflows:** table of overflow statistics for structures as result of a simulation (volume, frequency, duration, loads per year)

- **Catchments:** table of defined catchments and their characteristics (serves as input control) , only elements of type **CatchmentRunoff** are listed, elements of type **Area** and **SurfaceRunoff** are not shown

- **Pipes:** table of defined pipes and their characteristics (serves as input control) + Qdim und Qmax for capacity control

- **Immissions:** table of calculated immissions for defined receiving waters

- **Tools:** calculation of water level, hydraulic radius, shear stress and critical discharge i.d. (serves as control during program development, will be removed)

**Layout window** (separate window)**:** Used to display the network (see example below). The elements are displayed as rectangles (or in case of enhanced layout option with different shapes and colors depending on the element type), the connections (**Connect** commands) with arrows. Unlike in hydrodynamic programs such as SWMM5 or Mike Urban, the connections do not represent sewers, but simply show which elements are connected to each other. Pipes (sewers) are also displayed as elements. However, it is not customary in WaterElements (as in other hydrological programs such as SASUM, KOSIM etc.) to map all existing sewers, but greatly simplify the sewer system. This layout of the network is automatically determined by an algorithm. It is not possible to manually change this layout (i.e. to move the elements with the mouse). Under "Layout", limited changes in the layout can be performed (orientation, type of connections and display (simple or enhanced)).

**Note:** An editable layout can be created under **Network** using the "**Create Network**" button. However, this layout is currently not updating if the network has been changed or supplemented via commands. When clicking "Create Network" again the layout is updated, but it will be aligned like the automatic layout. "Create Network" is currently only useful if the network is completed and no changes (adding or deleting elements, changes of connections) are needed. The parameter values can be changed in an edit box that appears with a double-click on the respective element. The associated command is updated automatically.
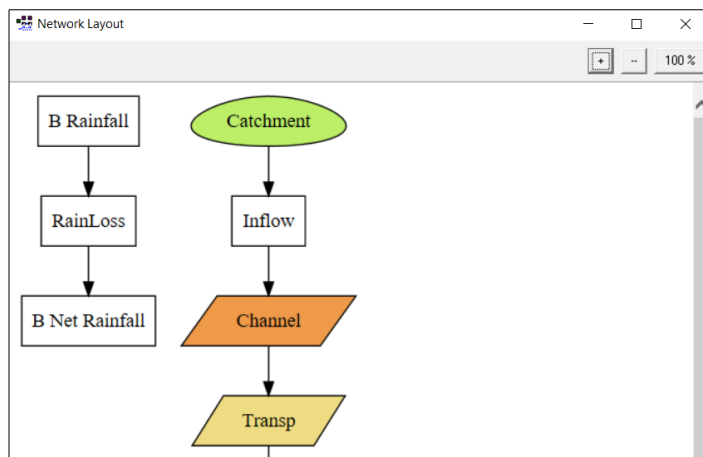


The main menu (File, Settings, Analyse, Layout, Refresh, Run) and its items are largely self-explanatory. **Analyse** serves to control the network (when the automatic layout of the network has not yet existed). Most menu items of **Analyse** are only active after applying "Execute Commands" and "Run".

## Use and Simulation

- First of all, commands must be present in the input editor. This can be done either by:
    - **Manual input of commands:**
        - Commands may be manually typed or
        - can be chosen from a list located above the editor window and inserted by clicking the "**Add Command**" button. Then the command is inserted with a description for each parameter. The description must be replaced by values. This can be done directly in the editor or by double-clicking on the command. This opens an inplace editor where the values can be changed.
    - **Drag und Drop von Kommandos** aus der Kommandoliste (links neben Editor) :
        - Im Editor zuerst den Elementnamen schreiben
        - In der Kommandoliste entweder den Befehl doppelklicken, dann wird er an die Cursorposition im Editor kopiert.
        - oder den Befehl mit der Maus in den Editor ziehen und am gewünschten Ort loslassen.
    - **Interaktive creation of the network** in the window „**Network**" (see Example 6)
    - or loading a previously created command file via the menu item "**File / Open**"
- If you **click in a command line** a hint appears with a list of command parameters
- If you **double-click on a command** an edit box appears where you can input or change parameter values. Clicking the OK button closes the edit box and the command is adjusted according to the inputs. Of course, the parameter values can also be changed directly in the editor.
- **The commands can be saved by "File / Save Data"** (if a file was loaded before, the commands will be saved in this file otherwise it asks for a filename.). With "File / Save Data and Results as" commands and potential results can be saved under a new filename (commands will be saved in [filename].dat, results in [filename].res).
- **After input or loading of commands** they must be executed by clicking the button "**Execute Commands**". In doing so **no simulation is performed** but the commands are converted in a network of elements and connections. If the commands contain errors, this will be shown in the control window below the editor window. Therefore you should check after "Execute Commands", if the control window shows the line "No errors found!". In this case, the „Run" button is no longer disabled.

    If an error occured you should click on "**Find Error**". Then the cursor jumps to the first error message in the lower window. Double-clicking on this message puts the cursor in the editor to the beginning of the line with the error. This line must now be checked and the input error must be fixed. Warnings should also be observed and checked which command has caused it and whether the command was really interpreted correctly.
- A **simulation is performed by choosing** the menu item "**Run**". If the simulation is finished, the time required for the simulation is shown in the control window.
- The **results are displayed on** the pages "Results", "Structures", "Overflows", "Catchments", "Pipes" and "Immissions".
- If you want to see **time series of specific properties**, you have to select them before running the simulation under "**Settings / Define Time Series**". **Warning:** Time series can be selected only after translation of the commands by clicking "Execute Commands" because the program needs to know which elements are present. After a simulation by "Run" time series are shown on the page

"**Monitor**". You should select at least a rainfall time series but only one. This series is displayed upside-down on the top axis as usual in hydrology. All other quantities (runoff, concentrations and loads) are displayed on the bottom axis. The y-axis is automatically scaled for the whole time period (normally 10 years) and not for the actual page which has the disadvantage that values are often small. It is possible to zoom into a range by holding the left mouse button and moving to the right. You will see a white frame as zoom window. If you release the mouse button the chart is zoomed into this white frame. To unzoom choose right-click and "Undo Zoom" or move to the left with left mouse button down.

- The menu item " **Layout** " defines how the **layout of the network is displayed** in the layout window. With " **Alignment** " the **alignment of the network** can be defined (left to right, right to left, top to bottom, bottom to top). " **Line Type** " defines the type of connecting lines: spline (curves), orthogonal (orthogonal lines), polyline (lines), line (individual lines). " **Hide Rainfall** " does not display the rain elements, which may make the display clearer or more like the scheme of a drainage concept. " **Enhanced layout** " shows the elements in ver - seemed which colors and forms (depending on type). This makes the display clearer than with the basic setting, in which all elements are displayed as rectangles with a white background. The changed settings must be updated with " **Refresh** ".
- With "**Refresh**" the **layout can be updated manually** (e.g. after changed settings in "Layout").

## Examples

The program will be explained by means of examples.

## Example 1 - Rainfall Statistics

- Please load the file "rainfall_statistics.dat" in the folder "examples" with "File / Open"
- The file contains only three commands. All lines with a # character are **comment lines and colored green**.
- The first line which is not a comment is:

Rainfall = **RainfallFromFile**(zuer8695.kmd, kmd)

It defines an element "Rainfall" which is defined by the function RainfallFromFile. The function has two parameters: the filename which contains the rainfall series and the format of the rainfall data. You can also see this if you move with the mouse over the command (a hint "Usage: RainSeriesName = RainfallFromFile(filename, time format)" is displayed) or if you double-click the command (then you can edit the parameter values in the Element editor). **Element names** may contain spaces and special characters and may begin with a number but the following characters are not allowed: (),#@ .

All **element names** are colored **purple**. The **function names** (RainfallFromFile, Balance etc.) are shown in **blue**.

- The second line: Balance = **Balance**(years) defines a balance element. Balance elements - as their name implies - make up the balance for water and mass that flow through the element. They may be inserted anywhere a balance is of interest in the network. The parameter determines over which time periods the balance should be made up. No parameter means to sum up over the whole simulation time period, "**years**" sum up over years and "**events**" over events. In the present example the balance is made up over years. You can see that if you go to the page "**Results**" after running a simulation: for each year the rainfall sum is listed and at the end the mean over all years. Because balances must be defined very often there exists a short way to define them. You can write

the element name of the balance and right-click to insert the rest of the command by selecting the appropriate item from the popup menu.

- In order for the balance element to balance the rainfall, it must be connected to the rainfall element. Connections between elements are established with the command **Connect**(Element1, Element2). Element1 and Element2 are two elements that were previously defined. The order of the elements determines the direction of flow. In the present case, water and substances flow from element 1 to element 2. For the example, the command is **Connect**(Rainfall, Balance). If the elements are not defined, an error message is shown. Because many connect commands have to be written when setting up a network, these can also be created more conveniently with a right-click and menu selection. The first selection creates "Connect(", the second selection is useful if several connect commands are to be created one after the other, as it automatically inserts the last element based on the previous line. The menu item "'**Connect' for new element**" is useful when an existing network is to be expanded and a new element that has already been defined with a command has to be inserted between two existing elements.

- The elements and their connections are displayed in the **layout window**. The **layout** of the network **is generated automatically** and it is not possible to change it manually (i.e. to move or position the elements with the mouse). Limited changes to the layout can be made under "**Layout**" (alignment, type of connections and display).

- The commands are checked by clicking on " **Execute Commands** ". If there is no error message in the info window resp. "**Find Error** " remains gray, then the menu item "R**un**" is enabled and a simulation can be carried out by clicking on "**Run**".

- After completion of the simulation the results are shown under "**Results**": Rainfall depth per year in mm and the average sum over these years

## Extension of the example

We can now expand the example a bit by adding an initial loss:

- RainLoss = **InitialLoss** (2.0, 6) defines an initial **loss** of 2 mm, whereby a new event begins when no rain falls for 6 time steps. At a time step of 10 min (Simulation time step is determined by the time resolution of the rainfall series) a new event is defined when the dry period is greater than 1 h.
- We also define a balance element for the net rainfall: NettoRegen = **Balance** (years) and combine the new elements:

      **Connect** (Bilanz, RainLoss)
      **Connect** (RainLoss, NettoRegen)

After the simulation, we now see two annual balances in the results, before and after initial losses.

## Use of global constants

The command "ConstantName = **Constant** (number)" can be used to define a global constant that can then be used multiple times as a parameter value. Defining a constant is only worthwhile if the parameter value occurs more than once and it is likely that it will need to be adjusted.

The project "**Rain Statistics with Konstanten.dat** " shows an example of how constants can be used. Load this project. Compared to the previous example, two lines have been added:

DryPeriod = **Constant** (6)

Loss = **Constant** (2.0)

The 1st line defines a constant **DryPeriod** with the value 6 (6 time steps of 10 min = 1 h), the 2nd line a constant Loss of 2 (2 mm).

These two constants can now be used as parameter values:  RainLoss = **InitialLoss (** Loss, DryPeriod)

## Example 2 - Russikon case study, only emissions, without immissions

This example includes a part of the Russikon case study, which is described in the REBEKA documentation. It consists of a combined sewer system with tank and a separate sewer system with overflow.
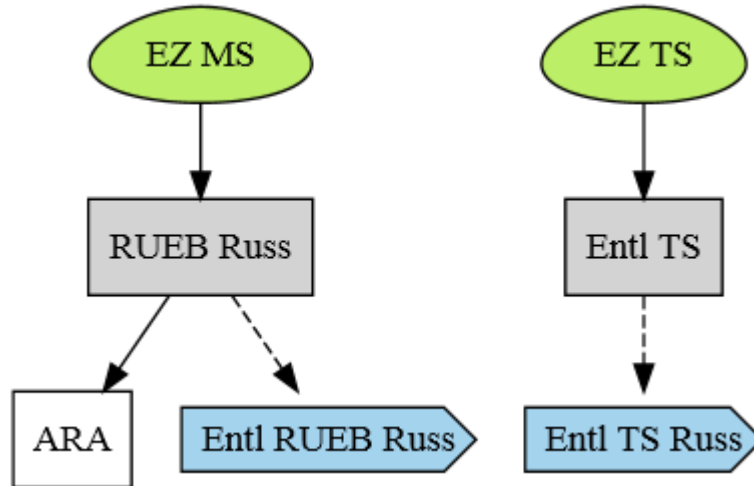


Load this project into the editor by choosing the file "**Projekt1 ohne Immissionen.dat**" with "**File / Open**". The network is displayed in the layout window. It is possible that not all elements are visible. Then you can zoom out by clicking the "--" button.

The first command we haven't used yet is:

**CatchmentRunoff**(initial loss [mm], dry period [time steps], area [ha], linear reservoir constant [min], DWF [m3/s])

This command defines a catchment which internally consists of several elements: initial loss, surface runoff, linear reservoir and constant inflow. Two catchments are defined in the example: for combined and separate sewer system.

The command **Overflow**(tank volume [m3], retained flow [m3/s]) defines overflow structures without and with tank. In the present case there is a CSO tank with a volume of 120 m$^3$ and a retained flow of 72 l/s:

CSO tank = **Overflow**(120.0, 0.072)

and for the separate sewer system only an overflow:

SSO = **Overflow**(0.0, 0.0)

With maximal retained flow 0 for the separate system, all of the water is discharged by the overflow. The resulting  number of discharges per year is then very high because each rainfall event that generates runoff leads to a discharge. If you want to compare the results with REBEKA2, you should set the retained flow to 10 % of the base discharge of the receiving water.

The retained flow is set to 10% of the baseflow of the receiving water. This is the default setting in REBEKA 2 in order to count significant discharges to the receiving water which are discharged via overflow. In case

of the separate sewer system, overflow and retained flow are discharged to the receiving water. Afterwards several balances for overflows and flow to the WWTP are defined:

> WWTP = **Balance**(years)
> B CSO tank = **Balance**(years)
> B SSO = **Balance**(years)

**The balance elements downstream of the discharges of the overflow elements are absolutely necessary** to calculate overflow characteristics (volume, loads, duration and frequency) which are presented in the "Overflows" table.

Then the connections have to be defined:

> # combined sewer system
> #
> **Connect (** Zuerich, EZ MS)
> **Connect** (EZ MS, RUEB Russ)
> **Connect** (RUEB Russ, ARA)
> #
> # Since an overflow element has two outputs (overflow and throttle discharge)
> # there is another connect command to forward the overflow
> # the normal Connect command forwards the throttle outflow
> #
> **ConnectOverflow** (RUEB Russ, Entl RUEB Russ)
> #
> # Separate sewersystem
> #
> **Connect** (Zuerich, EZ TS)
> **Connect** (EZ TS, Entl TS)
> **ConnectOverflow** (Entl TS, Entl TS Russ)

The branch "RainLoss => B Loss" is only for balancing the initial losses and could also be left out. At the very end, the pollutants have to be defined. For this, the following command is used:

**Pollutant** (rainwater concentration, wastewater concentration)

> # Pollutants
> #
> NH4 = **Pollutant**(0.0, 40) defines a pollutant NH4 with rainwater concentration of 0 and wastewater concentration of 40 g/m$^3$
> TSS = **Pollutant**(60, 200) defines a pollutant TSS with rainwater concentration of 60 g/m$^3$ and wastewater concentration of 200 g/m$^3$

## Variants

- The parameter values of the example can now be changed to see how this affects the emissions.
- Introduction of global constants, e.g. psi = **Constant** (0.85)     The constant can then be used like a number, e.g. in EZ MS = **CatchmentRunoff** (2.0, 6, psi * 19.8, 20, 0.009838)
- etc......

## Example 3 - Russikon with immissions

The previous example will now be expanded by defining a natural catchment. The command **NaturalCatchment** (...) differs only by the fact from **CatchmentRunoff**(...) that the pollutant concentrations are set equal to the rain water concentration for surface runoff and base flow.

Additional balances are inserted:

> B RecWater = **Balance**()
> RecWater + Discharges = **Balance**()

The next two commands define a receiving water section and the NH3 immission in this section:

Receiving Water = **ReceivingWater**(100.0, 2.5, 0.02, 1.0, 35, 0.015, 0.03)
**SetNH3Immission**(Receiving Water, NH4, B CSO tank, 5, 20, 8, 3, 8, 3, 1.5, 0.025)

The first command defines the receiving water section by length, bottom width, slope, side slope, Strickler coefficient, mean grain size and 90% grain size. This will automatically calculate the hydraulic stress for this section during simulation. The second command defines a NH3 impact. The following parameters must be specified:

- Element name of the receiving water section (here "**Receiving Water**")
- Pollutant name of ammonium (here NH4). $NH_3$ concentration in the receiving water is calculated from $NH_4$ concentrations, temperatures, pH values and alkalinity of overflow and receiving water.
- Name of the overflow or discharge element (here **B CSO tank**). This information is used to compare accumulated $NH_3$ dose with critical dose. This is carried out if Q(Overflow) > 0. If Q(Overflow) = 0 then the actual dose is set to 0 and the accumulation of the dose begins again if Q(Overflow) > 0.
- Tmin (here 5 °C) und Tmax (here 20 °C)
- pH value and alkalinity in the receivng water and in the discharged water (here both pH = 8 and alkalinity = 3)
- The STORM limits for NH3 immission (1.5 g·min/m$^3$ and 0.025 g/m$^3$, e.g. the maximal allowed concentration for an exposition d [min] is (1.5/d + 0.025) g/m$^3$)

In the example, the two substances NH4 and TSS were defined. NH4 was indeed already used in **SetNH3Immission** command and must therefore be defined. In contrast to elements, it is not necessary to define the pollutant prior to use in **SetNH3Immission**. When pollutants are defined, they will be automatically created for each element. In **CatchmentRunoff**(...) the concentrations of rainwater and wastewater are automatically set to the specified values. If you want to change concentrations, use the command **SetConc**(item name, material name, RW Conc., SW-conc). The input (i.e. default) concentrations are not changed during the simulation. The concentration in the dry weather flow can be changed with the definition of a dry weather variation.

To connect all elements a new element will be used: the element **Union**(), which combines two inflows. We need this element to connect the runoff in the receiving water with the overflows from combined and separate sewer system so that we finally obtain the total runoff in the receiving water.

To create the entire system already 15 connections must be defined. Here a disadvantage of the command language is visible. The connections can quickly become confusing.

## Example 4 - Definition of a Variation in Dry Weather Flow

In this example we will vary the amount of dry weather flow. Until now, the DWF in the combined sewer system has been constant over time. We will now take the combined system from Project1, duplicate it and then model it with constant and variable DWF.

As a starting point, we use the project "**Projekt1 ohne Immissionen**" (Project 1 without emissions**)**:

- We duplicate the elements of the combined system and add a 2 to the element names (e.g. EZ MS => EZ MS2 etc.)

- We delete all elements and connections of the separate system

- Now we have to insert two elements:

  - a **DWFVariation** that defines the DWF variation. For the sake of simplicity, we only choose a daily variation and leave the weekly and annual variation constant:

    fTWA = **DWFVariation**(Tagesgang, 1, 1)

  - and a **SeriesFromFile** element that defines the daily variation. It is saved in the file tagesgang_kleinstadt.txt. The command is therefore:

    Tagesgang = **SeriesFromFile(**tagesgang_kleinstadt.txt)

  and it must be defined before DWFVariation so that DWFVariation knows the variable „Tagesgang".

- With the command EZ MS2 = **CatchmentRunoff** (2.0, 6, 19.8, 20, fTWA * 0.009838), fTWA can now be used as a multiplicative factor with which the average value is multiplied.

- " **Execute Commands** " can be used to check whether the definition of the system is free of errors

- If this is the case, a simulation can be carried out with " **Run** "

- Are the differences in the overflow characteristics for the two networks (with and without DWF variation) large?

We can also compare the time series with each other:

- In order to select the time series, the system must be defined. This is the case after executing "**Execute Commands**". Then under "**Settings**" "**Define Time Series**" can be selected.

- Select one source element in the time series list, so that the rainfall time series is shown.

- Then choose time series depending on your interests, for example, downstream of the catchments or downstream of overflows

- Close the window and **run** a simulation with "**Run**". The time series are then visible under "**Monitor**".

- ......

## Example 5 – Off-line CSO tank

Select **File / New / Catchment - Pumped Offline CSO tank** . This loads a prepared example of an off-line CSO tank into the editor. The individual commands are commented. Please read through and then let it run. At the moment only a simple local control is implemented (On / Off). A pipe and a transport element are added to show the effects of these elements. The pipe element works in a similar way to a linear reservoir. The transport element delays the inflow by a certain number of time steps.

- Why is the maximum inflow to the WWTP a little higher than 25 l/s (maximum throttle flow for the overflow element "**CSO overflow**")?

- Modify the control so that the maximum flow to the WWTP is not higher than 25 l/s without changing the pump output (10 l/s) and the switch-on point (5 l/s).

- Optimize the switch-on point so that the discharge volume is as low as possible and the flow to the WWTP is not higher than 25 l/s.

- Is there an optimal pump capacity so that the discharge volume is as low as possible and the transfer to the WWTP is not higher than 25 l/s?

- Gibt es eine optimale Pumpleistung, so dass die Entlastungsmenge möglichst gering ist und die Weiterleitung zur ARA nicht höher ist als 25 l/s ?

## Example 6 - Interactive Build-up of a Network (since version 1.0.8.4)

Because the build-up of the network by commands could be tedious it is possible – since version 1.0.8.4) – to build up the network interactively in a graphical editor (like in a CAD or GIS). To do this, you have to switch to the " **Network** " window. The main elements can be selected and created from a palette: CatchmentRunoff, Overflow, CircularPipe, Union, Receiving-Water, Balance



To create an element select it with the mouse and click on the drawing layer. After the click, a window appears for entering the element name. Then the symbol is drawn together with the name (top right of the symbol). The corresponding command is automatically generated in the command editor. In the case of catchments, the command to connect the rainfall with the catchment is also generated. For the first catchment the command to read in the rainfall file is also generated (RainfallFromFile, if not already created).

Connections can be made with the symbols  . The first two types are recommended to use. To create the connection select a type, then move the mouse onto an anchor of source element (four blue crosses on the border of the elements) until the cursor shows a hand, then click to establish the connection, then move to the target element until the cursor shows a hand and then click again. This ends the build-up and creates the connection between the elements.
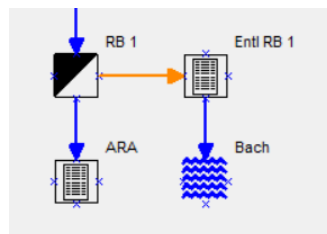


The arrow shows the direction of the water flow. This must also be taken into account when creating the connection by first selecting the source and then the target element.

In the case of an overflow structure (**overflow**), a balance element (**balance**) should always be created to balance the discharge. The following has proven to be useful for naming: "B " + name of the structure, for example "B RB 1" if the structure is called "RB 1". The balance element is necessary so that the overflow structure is listed in the **Overflows** table. It calculates the overflow characteristics (overflow volume, duration, frequency and maximum discharge), which are displayed in the "**Overflows**" table .

Normal connections are created with the **Connect** command. The **ConnectOverflow** command **must be** used for overflow connections. In the layout, a connection is converted into an overflow connection by **double-clicking it**. It is then colored orange instead of blue. In order to generate the corresponding commands for the connections, the connection must be selected. Then right-click in the pop-up menu and select "**Create Command from Connection**". This **appends** the command (**Connect** or **ConnectOverflow**) in the command editor. Multiple connections can be selected to generate commands from these. It is best to generate the commands for the connections at the end, when the entire network is graphically built-up and no more changes are expected. To do this, first **select** "**Select all Connections**" and then "**Create Command (s) from Connection (s)**". This will write all Connect and ConnectOverflow commands to the command editor.

Since standard parameter values were already created for the created element commands, "**Execute Commands**" can be used to check whether the network has been set up correctly. If an error occurs, read the error message carefully and correct the error. One of the most frequent errors is that the rainfall file can not be found because the path is incorrect. If no error occurs, a simulation could be carried out with "**Run**", but this is pointless as the parameter values are not yet correct.

Under "**Network / Help**" there are tips on useful keys for changing the layout: **s**(ource), **t**(arget), **l**(ink type) for connections and **v**(ertical), **h**(orizontal) for aligning when selecting several elements.

The created network can be saved with "**Save Diagram**". The project name with the extension .dgr (diagram) is used as the file name. **Note:** At the moment this layout will be saved when saving the project but not when you open another project. It will automatically re-loaded when opening the project again.

The parameter values can be entered by **double-clicking** on the elements. For this purpose, a small window, the **element editor**, appears with a list of the parameter values (two columns with description and parameter value). The changes must be confirmed with **OK**. The element editor is closed and the corresponding command in the command editor is updated with the changed values. The element editor can also be called up in the command editor by **double-clicking the command line** .

When all elements and connections have been defined, **save the layout** with "**Save Diagram**" and commands with "**File / Save Data**". **Check** with " **Execute Commands** " that there are no errors.

**Attention:** At the moment it is not possible to update the manual layout resp. it will not be automatically updated if additional elements and connections are defined via commands. The update is possible with "**Create Network**", but the manual positioning of the elements is lost. The positions of the automatic layout are adopted.
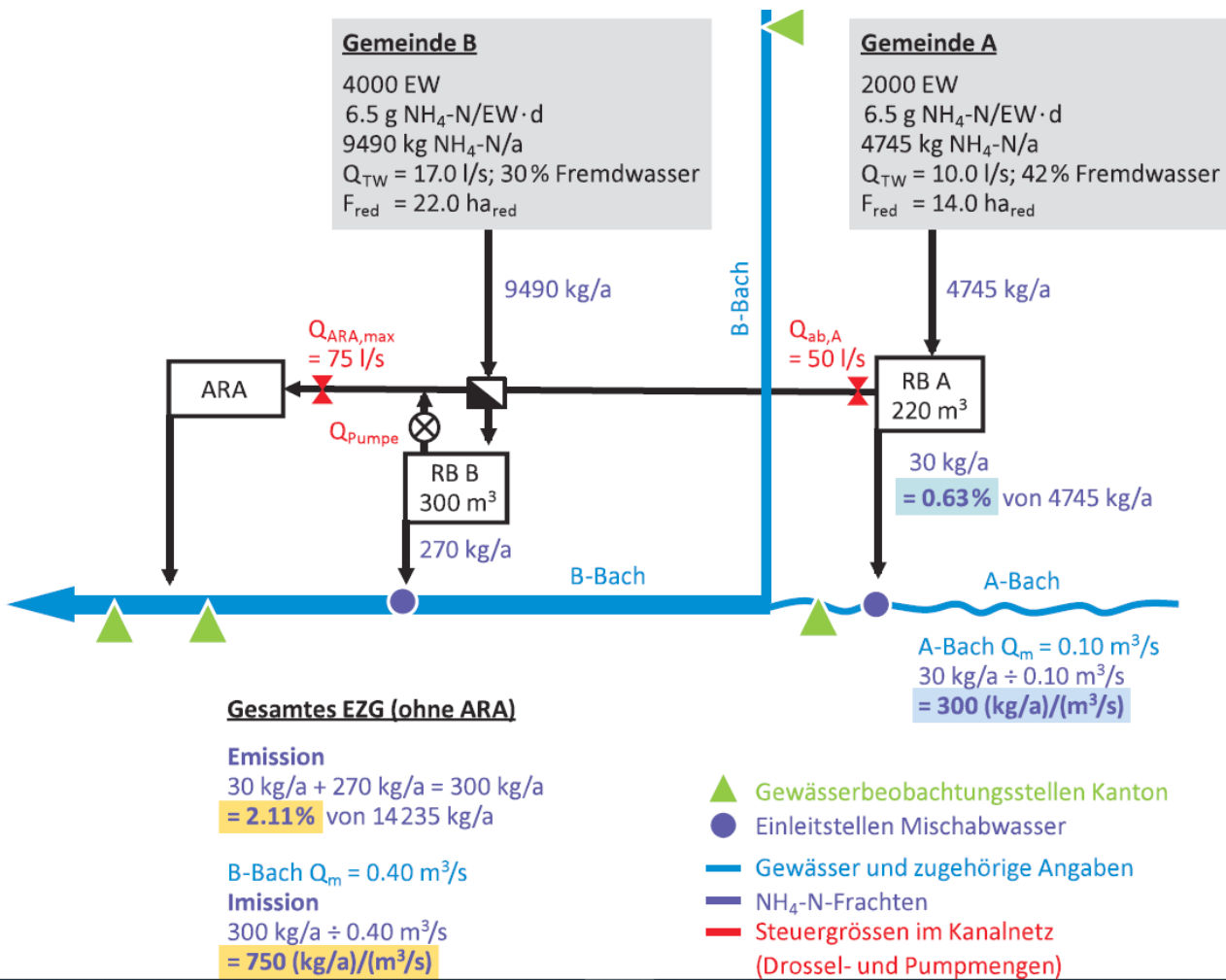
If a new command is entered in the editor (element or connection), then it is possible to create the corresponding graphical element in "**Network**" with right-click on the command and "**Create graphical element for this Command**". Elements are always created in the upper left corner of the layout canvas and must then be positioned manually. In the case of connections, the anchors may have to be changed with s or t.

### Interactive Build-up of Case Study III from the new VSA guideline
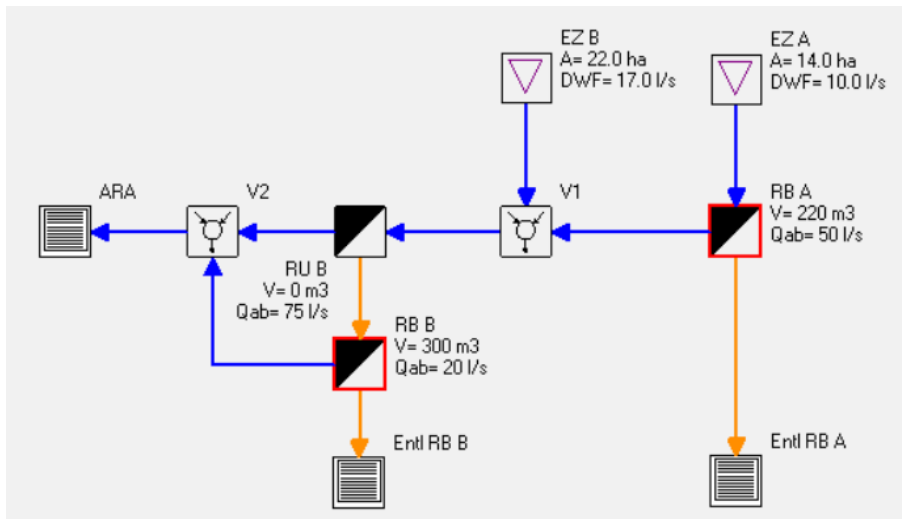
In the new VSA guideline "**Wastewater management during rainy weather**" there is case study III in module B, Appendix 2: **Combined Sewer Overflow** (see next figure)**.**

We will now build up this (already schematized) system interactively and run a simulation. When building up the network in WaterElements, it must be taken into account that all elements except an union can only

have one inflow. Therefore real systems must be schematized and possibly some fiktive unions must be inserted to fullfill the above rules for connections.



**Gemeinde B**

4000 EW
6.5 g $NH_4$-N/EW·d
9490 kg $NH_4$-N/a
$Q_{TW}$ = 17.0 l/s; 30 % Fremdwasser
$F_{red}$ = 22.0 $ha_{red}$

**Gemeinde A**

2000 EW
6.5 g $NH_4$-N/EW·d
4745 kg $NH_4$-N/a
$Q_{TW}$ = 10.0 l/s; 42 % Fremdwasser
$F_{red}$ = 14.0 $ha_{red}$

9490 kg/a

B-Bach

4745 kg/a

$Q_{ARA,max}$ = 75 l/s

ARA

$Q_{Pumpe}$

RB B 300 m³

270 kg/a

B-Bach

$Q_{ab,A}$ = 50 l/s

RB A 220 m³

30 kg/a = **0.63 %** von 4745 kg/a

A-Bach

A-Bach $Q_m$ = 0.10 m³/s
30 kg/a ÷ 0.10 m³/s
= 300 (kg/a)/(m³/s)

**Gesamtes EZG (ohne ARA)**

**Emission**
30 kg/a + 270 kg/a = 300 kg/a
= **2.11 %** von 14 235 kg/a

B-Bach $Q_m$ = 0.40 m³/s
**Imission**
300 kg/a ÷ 0.40 m³/s
= **750 (kg/a)/(m³/s)**

▲ Gewässerbeobachtungsstellen Kanton
● Einleitstellen Mischabwasser
— Gewässer und zugehörige Angaben
— $NH_4$-N-Frachten
— Steuergrössen im Kanalnetz
 (Drossel- und Pumpmengen)

The following figure shows the implementation of the above system in WaterElements. In this case, two unions are required, one (**V1**) upstream and one (**V2**) downstream of "**RU B**".



EZ B
A= 22.0 ha
DWF= 17.0 l/s

EZ A
A= 14.0 ha
DWF= 10.0 l/s

ARA   V2   V1   RB A
V= 220 m3
Qab= 50 l/s

RU B
V= 0 m3
Qab= 75 l/s

RB B
V= 300 m3
Qab= 20 l/s

Entl RB B

Entl RB A

The creation of the system is best started with the catchments. The further steps were already explained in [Example 6](#).

The pumping out of **RB B** must be defined with a **SetControl** command (RB B, RU B, …) (see [example 5](#)). According to the information in the case study, the pump is only switched on when the discharge at **RU B** falls below 30 l/s. It can then be pumped out at 40 l/s (70 l/s - 30 l/s), i.e. Qab, max of **RB B** can be set to 0.040 m³/s and **SetControl**(**RB B**, **RU B**, 0.030). An even more optimal control can be achieved with **SetControl**(**RB B**, RU B, 0.075, 1). Option 1 (last parameter value) has the effect that the pumping flow rate is always set to 75 l/s - (Qab of RU B) (if the amount of water in the tank allows this), so that the flow rate to the WWTP is not higher than 75 l /s.

The simulation results can deviate from those in the case study because a different rainfall series and a different simulation program (possibly SWMM) may have been used.

# Important Terms

## Comment Lines

All lines starting with a # character are **comment lines**. You can insert comment lines everywhere. Note: Once a # sign is on the line, the whole line is ignored. Comments are colored green by default.

## Element Names

Element names may contain spaces or special characters and may also start with a number. They are case-sensitive. An element name must always be written the same, otherwise it is interpreted as a new element name. The following characters can not be used in element names: (),#
Element names are shown in purple.

## Event Definition

Some elements such as **InitialLoss** and **Balance**(**events**) require a rainfall event definition. An event is defined by the length of the dry period between rainfall. In REBEKA2 the rainfall events are divided so that the dry period between two events will take at least 1 hour.

With WaterElements, the rainfall event separation is defined via the dry period in the element **CatchmentRunoff** or **InitialLoss**. If several such elements are defined, the dry period must be the same in all elements, otherwise an error message will appear.

In the counting of overflows not more than 1 overflow is counted per event. The number of overflows is therefore not a good quantity because it depends on the definition of the rainfall events. The command **CalcEvents** allows an alternative counting of events.

## Simulation Time Step

The time step of the simulation is determined by the temporal resolution of rainfall data. The default is 10 minutes, as the data from MeteoSwiss have this resolution.

**Undocumented Feature:** With the keyword "half" in the command **RainfallFromFile**([File Name], kmd, half) the temporal resolution is halved. The rain intensity of the original time interval is not interpolated but assumed to be constant over the two new intervals.