Ministry of Environment of Denmark
Environmental Protection Agency

GrundRisk Landfill
User Guideline

Environmental Project no. 2180

March 2022

Publisher: The Danish Environmental Protection Agency

Editors:
Luca Locatelli
Philip Binning
Majken Frederiksen
Poul L. Bjerg

ISBN: 978-87-7038-313-4

Forbehold

Denne rapport er en del af ”Metodik til stedsspecifik risikovurdering ved deponering af affald”. Rapporten er foreløbig og af oplysende karakter, og indholdet kan på nuværende tidspunkt ikke alene danne grundlag for en konkret sagsbehandling og myndighedsafgørelse. Baggrunden for dette er, at Miljøstyrelsen arbejder for at afklare særlige forhold omkring vandrammedirektivets betydning for stedsspecifik vurdering af deponeringsanlæg og påvirkningen heraf i receptor. Afklaringerne kan give anledning til konsekvensrettelser i metodikken, som den er formuleret for nuværende, og det kan være nødvendigt at rettelserne skal indarbejdes i metodikkens værktøjer herunder modelværktøjer, brugervejledninger og dokumentationsrapporter. Således må offentliggjorte rapporter og værktøjer under metodikken for nuværende betragtes som foreløbige.

Miljøstyrelsen offentliggør rapporter og indlæg vedrørende forsknings- og udviklingsprojekter inden for miljøsektoren. Det skal bemærkes, at en sådan offentliggørelse ikke nødvendigvis betyder, at det pågældende indlæg giver udtryk for Miljøstyrelsens synspunkter. Offentliggørelsen betyder imidlertid, at indlægget udgør et væsentligt indlæg i debatten omkring den danske miljøpolitik.

Risikovurderingsværktøjet er beregningsteknisk forberedt til at kunne regne med nedbrydning når et bedre datagrundlag er tilvejebragt. Derfor indgår nedbrydning i sammenfatningen, brugervejledninger og som en del af transportmodellen. Miljøstyrelsen finder på nuværende tidspunkt ikke tilfredsstillende dokumentation for at nedbrydning kan indgå som en aktiv del i sagsbehandlingen ved brug af værktøjet. Såfremt at der på et senere tidspunkt tilvejebringes ny viden er Miljøstyrelsen åben for at lade nedbrydning indgå. Det betyder at metoden er forberedt til at inkludere nedbrydning, men at Miljøstyrelsen mangler den nødvendige viden for at kunne vurdere denne i sagsbehandlingen. Ny viden kan bl.a. bestå i en opdateret samlet videnskabelig rapport, hvor det er beskrevet, hvorledes der kan regnes med nedbrydning konkret i perkolatfaner fra deponeringsanlæg.

Contents

1. Introduction

This is a guidance document to assist the use of the user interface GrundRisk landfills. The models of the interface are described in Miljøstyrelsen, 2018. The user interface includes 2 different models (Affald-A and Affald-B) to simulate contaminant transport from landfills. Each model has 2 different graphical user interfaces: one to simulate contaminant transport from a single source unit and one for multiple source units since landfills can be made either of a single or multiple units.

The following sections describe the details of the user interface. This user manual includes some practical tutorials/examples so that the user can see how to set up a similar case study. The input files for these tutorials are provided together with the model.
The models simulates the contaminant water phase concentration and mass discharge in the aquifer downstream the landfill. The concentrations are simulated over a 2 m well screen at a user specified distance (Point of compliance) downstream the landfill/unit. The input time series of concentration and water discharge from the landfill are obtained from the Kildestyrke-model (2017).

2. System requirements

The program was implemented in MATLAB and it is distributed in a standalone version that does not require a MATLAB license. The program is compatible with Microsoft Windows systems and it might not work with other operating systems. An internet connection is required during the installation process. Once the program is installed it can be used without internet.

3. How to install

The program needs to be installed in order to be used. The user needs to run the distributed .exe file by double clicking on it (it can take few minutes until the installation window pops up). The installation procedure will download approximately 600 MB and therefore an internet connection is required during the process. The user needs to follow the instruction similarly to other software.

4. Model input parameters

The input model parameters of the model Affald-A are shown in the following table. The report of Miljøstyrelsen (2018) shows some examples about setting the parameter values (it was recommended that the groundwater velocity *u* should be the one at the most downstream point of the landfill).

TABLE 1. Input parameters of the model Affald-A (from Miljøstyrelsen, 2018)

	Parameter	Description
Single source parameters	$C_0(t)$ [M/L³]	Concentration time series in the water phase at the source, this is provided by the Kildestyrkemodel
	$Q_0(t)$ [L³/T]	Landfill water discharge time series through the source area, this is provided by Kildestyrkemodel
	Z [L]	Distance between the bottom of the landfill unit and the top of the aquifer
	L_x [L]	Source length, this is provided by Kildestyrkemodel
	L_y [L]	Source width, this is provided by Kildestyrkemodel
Vertical model	$k_{\_v}$ [T^{-1}]	First order degradation rate
	$\theta_{\_v}$ [-]	Water content (fraction of the total volume)
	$R_{\_v}$ [-]	Retardation factor
Horizontal model	H [L]	Thickness of the aquifer
	I [L/T]	Groundwater recharge
	u [L/T]	Groundwater velocity
	k [T^{-1}]	First order degradation rate
	n [-]	Porosity
	α_L [L]	Longitudinal dispersivity (x direction)
	α_T [L]	Transversal dispersivity (y direction)
	α_V [L]	Vertical dispersivity (z direction)
	R [-]	Retardation factor
	POC [L]	Distance to the point of compliance

Similarly, the input model parameters of the model Affald-B are shown in the following table.

TABLE 2. Input parameters of the model Affald-B (from Miljøstyrelsen,2018)

	Parameter	Description
Single source parameters	$C_0(t)$ [M/L³]	Concentration time series in the water phase at the source, this is provided by Kildestyrke-model model
	$Q_0(t)$ [L³/T]	Landfill water discharge time series through the source area, this is provided by Kildestyrke-model model
	L_z [L]	Source depth, this is provided by Kildestyrkemodel model
	L_y [L]	Source width, this is provided by Kildestyrkemodel model
Horizontal model	H [L]	Thickness of the aquifer
	u [L/T]	Groundwater velocity
	k [T^{-1}]	First order degradation rate
	n [-]	Porosity
	α_L [L]	Longitudinal dispersivity (x direction)
	α_T [L]	Transversal dispersivity (y direction)
	α_V [L]	Vertical dispersivity (z direction)
	R [-]	Retardation factor
	POC [L]	Distance to the point of compliance

5. Start the program. The main window

Once the program has been successfully installed it can be launched from the main Windows menu.
The main window of the user interface is shown in the figure below.

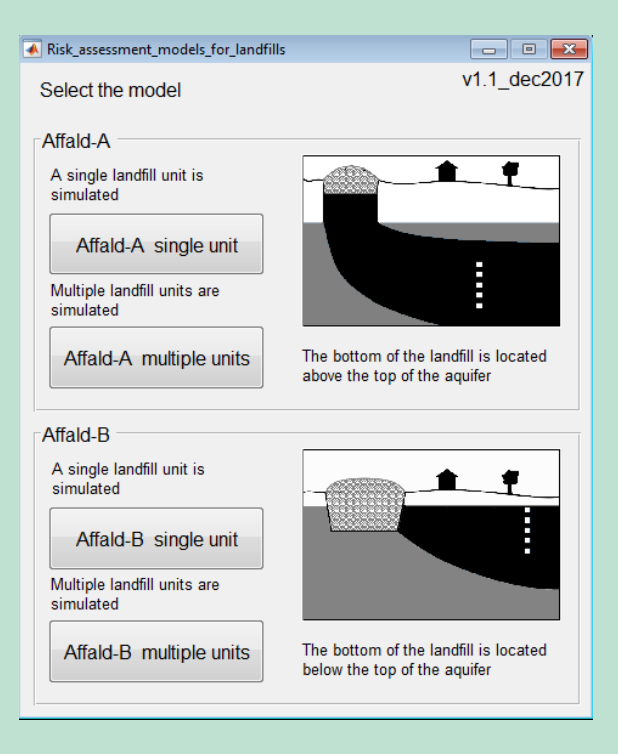


FIGURE 1. The main window of the user interface.

The main window of the user interface allows the user to choose between 4 different options. Each of the 2 models (Affald-A and Affald-B) has 2 options: the first simulates a single source unit of a landfill, the second a landfill made of several source units.

The following figure shows the 4 models (from Miljøstyrelsen, 2018).

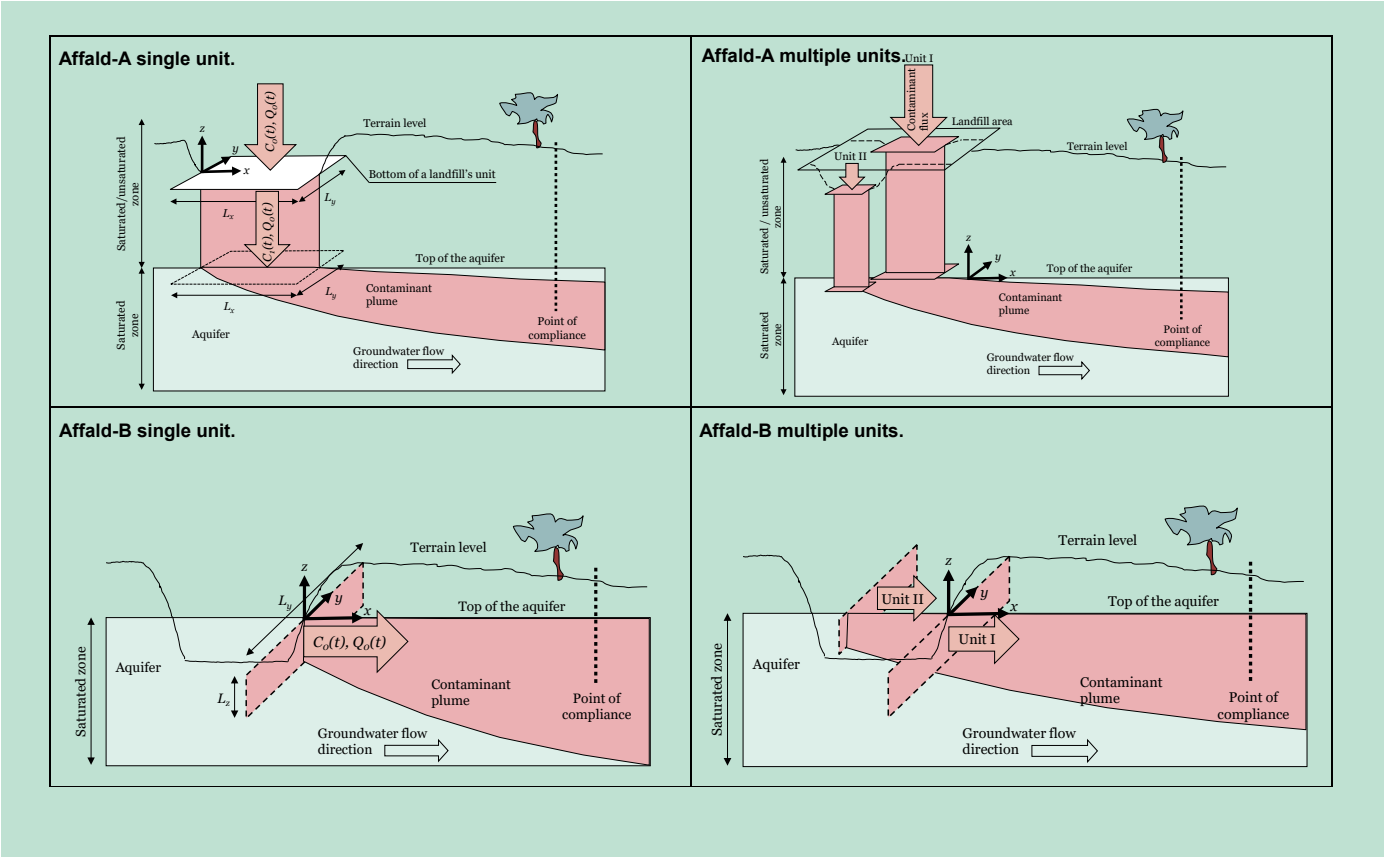


FIGURE 2. Overview of the different models (from Miljøstyrelsen, 2018).

6. Affald-A single unit

The model “Affald-A single unit” (Figure 2) can be used to simulate concentration and mass discharge downstream a single landfill unit that is located above the aquifer (the bottom of the unit is above the top of the aquifer).
The interface of the model “Affald-A single unit” is shown in the following figure. The user has different options (Back to the main menu; Load model inputs).
First of all, the user must enter the input time series of concentration and water discharge that are provided by the Kildestyrkemodel (2017) (see the section “How to create the input .txt file”). The user needs to press “Browse” within the “Source inputs” box. A new window pops up where the user needs to select the input .txt file. The input file must be created correctly according to the instructions given in the section “How to create the input .txt file”. If the input .txt file is made correctly the user will be able to see the 2 graphs of the input time series (water discharge and concentration) in the user interface.
Once the input time series is included then the user can save (this is optional) the input parameters by pressing “Save” in the “Save all the inputs” box. The user can also load previously saved inputs by pressing “Load model inputs”. When all the input parameters are given the user can select the result file name. The result file name is recommended to have the case study name and the compound being modeled (i.e. Ryparken_Ammonium.txt). Finally, the user can run the model by clicking on the “RUN” button. The model will run and automatically pop-up a result window and write the output .txt file in the same location as the selected .txt input file.

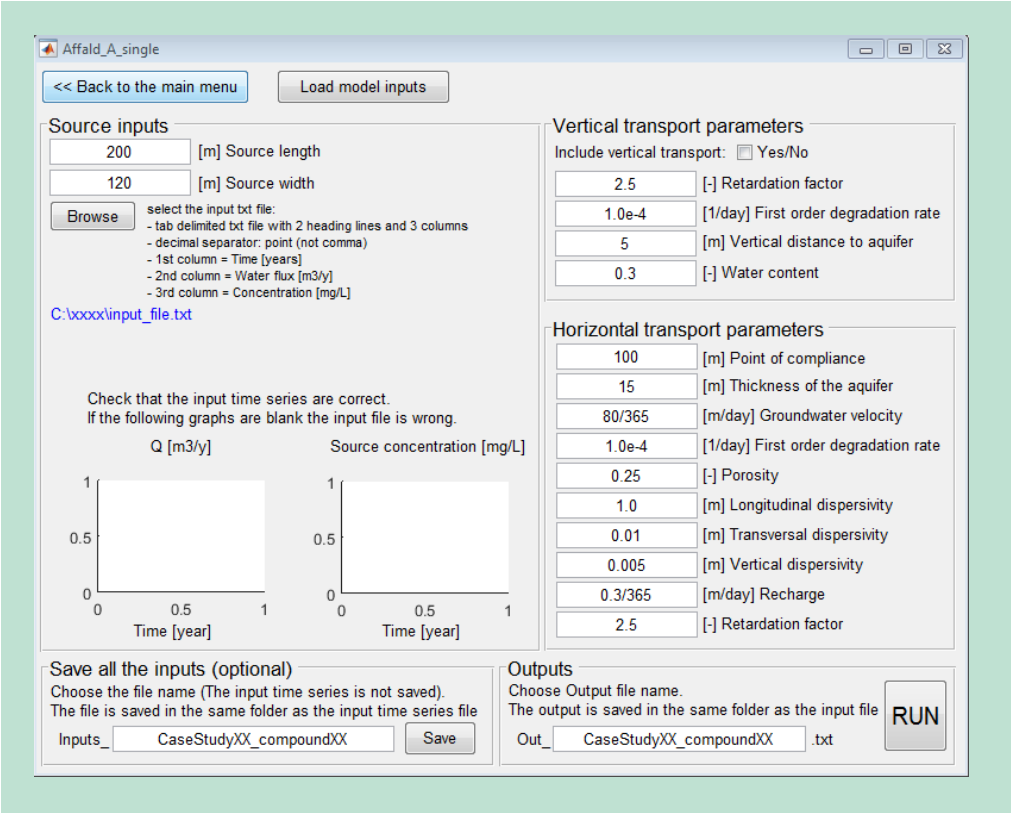


FIGURE 3. Screenshot of the model ”Affald-A single unit”.

a. Affald-A single unit. Step by step tutorial

In the following we show a step by step example of how to use the model:

1) Open the “Affald-A single unit” window.
2) Load the provided input file “EX_1_Source_inputs_AffaldA_SingleUnite_constantQ_varyingC” (4 different examples are provided and the user can choose among them).
3) Let us type a case study name, i.e. “CASE_AAA” as shown in the following figure (at the bottom of the windows).

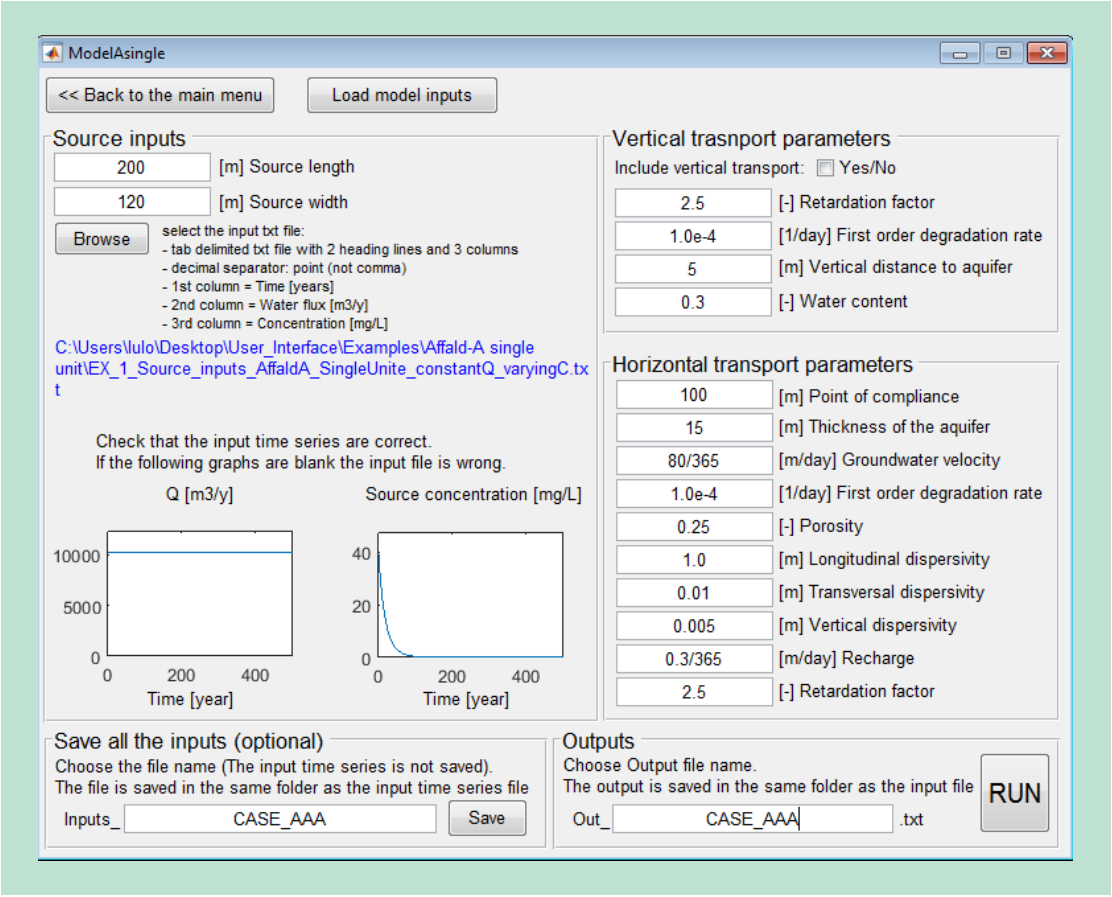


FIGURE 4. Screenshot for the step by step tutorial of the model ”Affald-A single unit”. The input time series are properly displayed and the user has called the case study “CASE_AAA”.

4) The user can save the inputs (this is optional) by pressing the “Save” button. Saved inputs can then be loaded with the “Load model inputs” button.
5) The user can change the input parameters (remember to use a dot as decimal separator and not a comma).
6) Run the model by pressing the button ‘RUN’ and wait until the results pop-up. The result window that pops up is shown in the following.

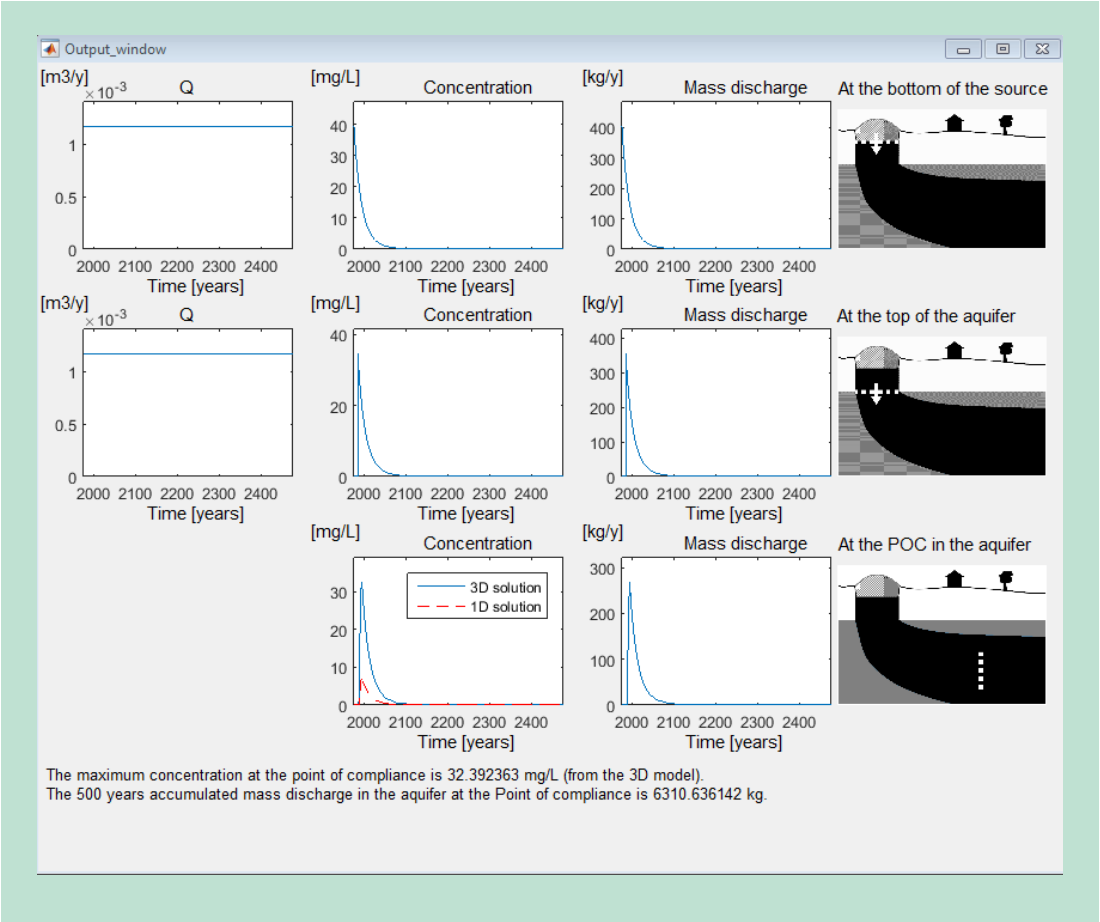


FIGURE 5. Example of a result window of Affald-A single unit.

7) Reading the results from the pop-up window.
   The top 3 graphs show the water discharge Q, the concentration and the contaminant mass discharge at the source.
   The middle 3 graphs show the water discharge Q, the concentration and the contaminant mass discharge after vertical transport (at the top of the aquifer).
   The 2 bottom graphs show the concentration over a 2 m well screen at the point of compliance and the contaminant mass discharge over an infinite plane (perpendicular to the groundwater flow direction) located at the point of compliance. The concentration at the POC is always computed using both a 3D model and a 1D model. The model giving the highest concentrations is the one to be used. The idea is that the 3D model is suitable for thick aquifers whereas in the case of thin aquifers the 1D model is more accurate (because the 3D model is designed for aquifers of infinite extent). Further details can be found in the main report.
   At the bottom of the result window there are 2 sentences reporting the maximum concentration at the point of compliance and the 500 year accumulated contaminant mass discharge.
8) A result file called “Out_CASE_AAA.txt” is automatically created in the same folder as the input time series file. The output file contains all the output time series and it is shown in the following (the tab delimited .txt file can be opened in Excel). Each column of the output file include a time series:
   Column 1. Time vertical trans. [y]. Time of the vertical transport model outputs.
   Column 2. Conc. vertical trans. [mg/L]. Concentration after vertical transport (the related time is in column 1).
   Column 3. Mass vertical trans. [kg/y]. Mass discharge to the aquifer after vertical transport (the related time is in column 1).

Column 4. PoC_t_3D. [y]. Time of the 3D model outputs of the horizontal transport.
Column 5. PoC_c_3D. [mg/L]. Concentration at the Point of compliance from the 3D model (the related time is in column 4).
Column 6. PoC_t_1D. [y]. Time of the 1D model outputs of the horizontal transport.
Column 7. PoC_c_1D. [mg/L]. Concentration at the Point of compliance from the 1D model (the related time is in column 6).
Column 8. PoC_Mass_1D. [kg/y]. Mass discharge at a plane at the Point of compliance distance from the 1D model (the related time is in column 6).

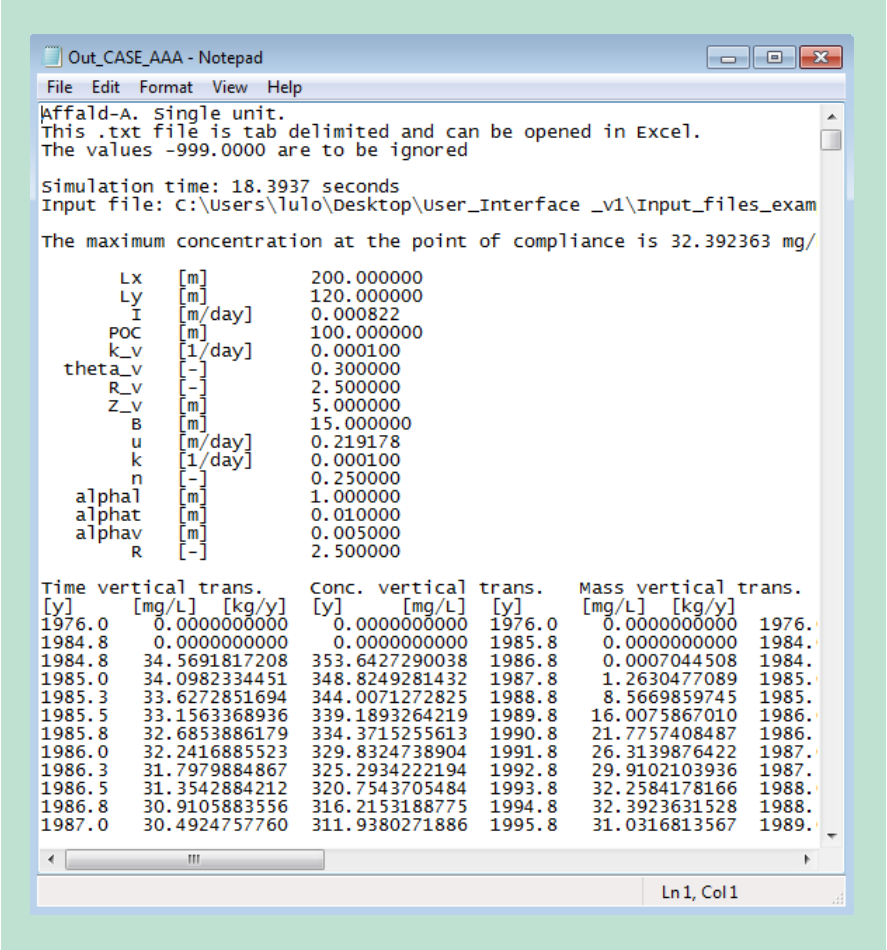


FIGURE 6. Example of an output .txt file from the model ”Affald-A single unit”. This .txt tab delimited file can be opened in Excel, where it will be easier to read.

7. Affald-A multiple units

The model “Affald-A multiple units” (Figure 2) can be used to simulate concentration and mass discharge downstream a landfill made of several units that are located above the aquifer (the bottom of the landfill is above the top of the aquifer).
The interface of the model “Affald-A multiple units” is shown in the following figure. The user has different options (Back to the main menu; Save all the inputs; Load model inputs).

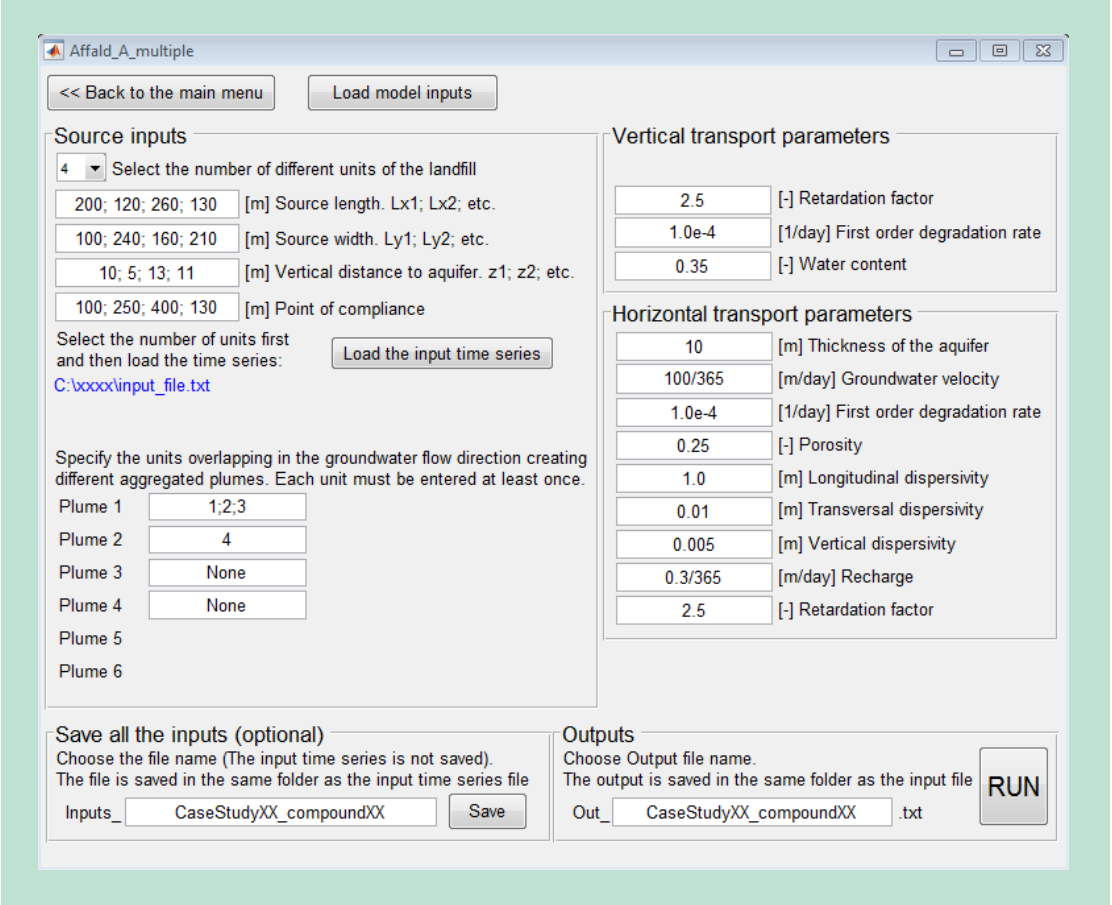


FIGURE 7. Screenshot of the model ”Affald-A multiple units”. This model does not have the user friendly option to omit the vertical transport, nevertheless the user can set the “Vertical distance to aquifer” to zero (for one or more units).

First of all the user must enter the right number of units to be simulated using the drop-down menu in the “Source inputs” window. Then the fields “Source length”, “Source width”, “Vertical distance to the aquifer” and “Point of compliance” must be entered. Note that there must be the same number of entries as the number of units (i.e. if there are 4 units then there must be 4 numbers in the field “Source length” and in the other fields). The number of entries must be separated by a semicolon. For instance, the landfill of Tandskov (one of the examples included Miljøstyrelsen, 2018) has 4 different units and the following figure shows how to input the 4 unit geometries, distances to the PoC, etc.

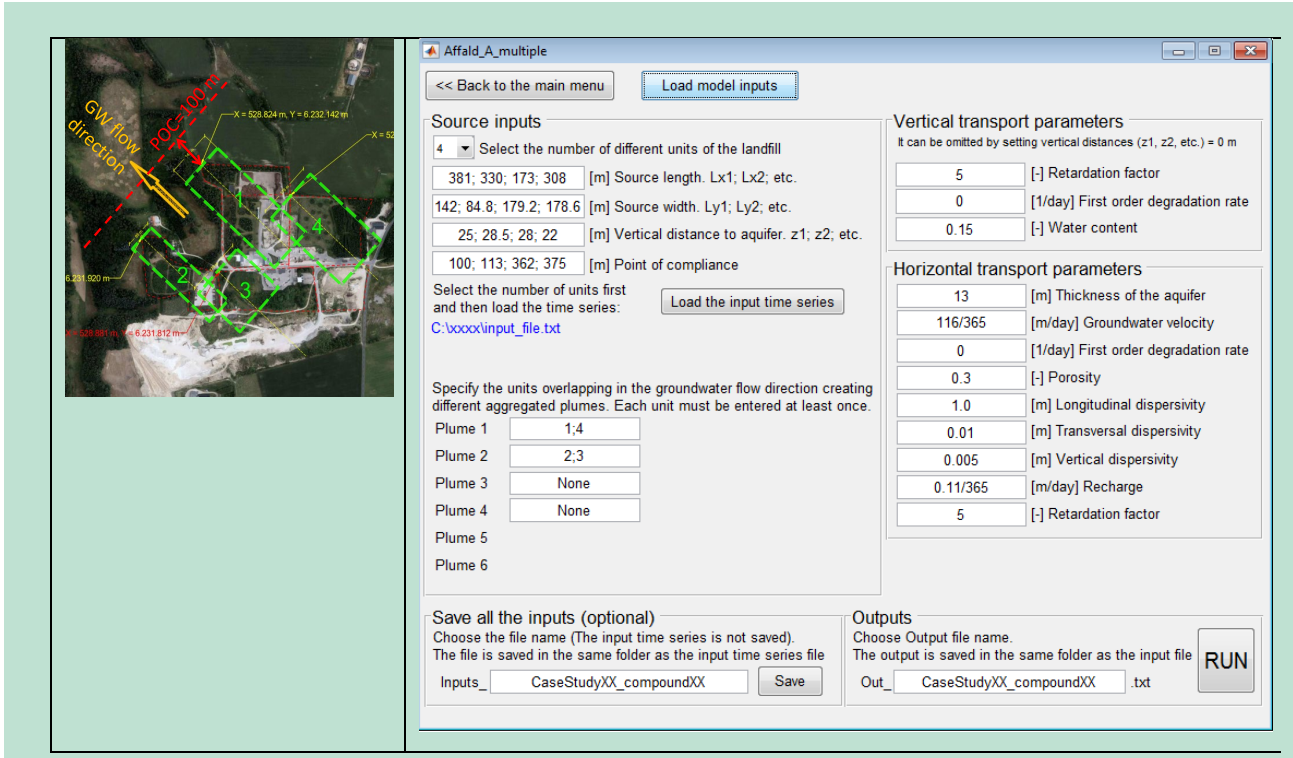


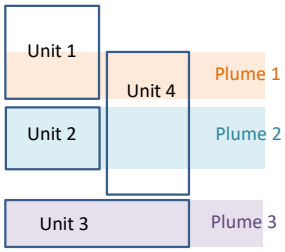
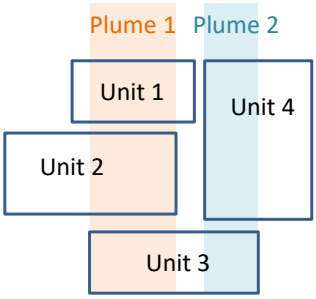
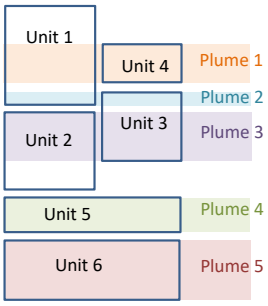
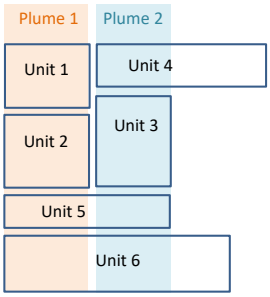
FIGURE 8. Example of how to enter the 4 units of the landfill of Tandskov. The input data are provided together with the model interface and are located in the folder "Input_files_examples_for_the_tutorials"'

Now the user must enter the input time series by pressing the "Browse" button within the "Source inputs" box. A new window will open where the user needs to select the input .txt file. The input file must be created correctly according to the instructions given in the section "How to create the input .txt file".

Afterwards, the user needs to specify how the different source units overlap in the groundwater flow direction forming aggregated plumes. The following figure gives some examples that show how to specify the different plumes resulting from the overlapping of single plumes from single units. The following rules apply in order to specify the overlapping in the groundwater flow direction:

- The unit numbers must be separated by a semicolon
- Each unit must appear at least once
- If a unit does not overlap with others then the user should specify a plume made of that single unit; however, if that unit already overlap with other units, the user must not specify another plume made of that single unit.
- If there are not further plumes then the user must type "None"

TABLE 3. Examples of how to specify the overlapping of different units in the case of multiple source units.

Groundwater flow direction	Plan view of the different units of a landfill and the aggregated plumes.	How to specify the overlapping of units in the user interface	Comments
GW flow		Plume 1 1;4 Plume 2 2;4 Plume 3 3 Plume 4 None	The user should not specify another plume made of Unit 1 alone; or Unit 4 alone. These units are already included in the other plumes.
GW flow		Plume 1 1;2;3 Plume 2 3;4 Plume 3 None Plume 4 None	The user should not specify other plumes made of Unit 2 alone; or Unit 1+2; or Unit 1+3; or for Unit 4 alone). These combinations are already included in the other plumes.
GW flow		Plume 1 1;4 Plume 2 1;3 Plume 3 2;3 Plume 4 5 Plume 5 6 Plume 6 None	Unit 1 and 3 slightly overlap. Therefore, the user could choose to run a further scenario where the 2 units are assumed to not overlap.
GW flow		Plume 1 1;2;5;6 Plume 2 3;4;5;6 Plume 3 None Plume 4 None Plume 5 None Plume 6 None	The user should not specify other plumes made of Unit 4 alone; or Unit 4+6). These combinations are already included in the other plumes.

It must be kept in mind that that the source geometry Lx and Ly is simplified as discussed in the Kildestyrkemodel. The source geometry is always (1) rectangular and (2) aligned with the groundwater flow direction. This means that complex shapes and even rectangular shapes not aligned with the groundwater flow direction are always simulated as rectangular and aligned with the groundwater flow direction. This can affect the way the overlapping of the plumes is simulated. Discussion about the effect of a simplified source geometry is given in the following section.

Once the input time series is included then the user can save (this is optional) the input parameters. When all the input parameters are given the user can select the result file name and run the model by clicking on the “RUN” button. The model will run and automatically pop-up a result window and write the output .txt file in the same location as the selected .txt input file.

a. Discussion about the effects of a simplified source geometry

The simplified source (unit) geometry (always rectangular and aligned with the groundwater flow direction as discussed in the Kildestyrkemodel) can influence the way units overlap to form aggregated plumes. Therefore, the user interface was designed to give the user the possibility to choose how the plumes from single units overlap forming aggregated plumes.

The following figure shows an example of a real case study. The simulated units (the blue and yellow continuous lines) do not overlap with each other in the groundwater flow direction. Therefore the user will specify 2 different aggregated plumes (in the user interface), one made of Unit 1 (DEL 1 in the figure) and one made of Unit 2 (DEL 2). Nevertheless, a further scenario is plausible; in fact the 2 "real" units (the green lines in the figure) do overlap in the groundwater flow direction. Therefore, the user could run a further scenario where an aggregated plume (from Unit 1 and 2) is simulated.

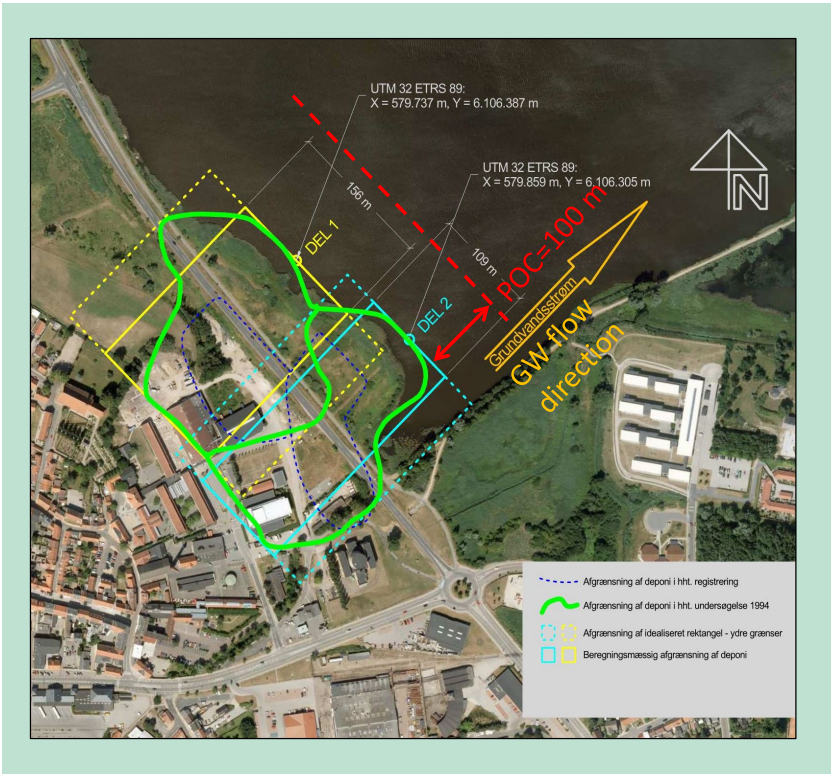


FIGURE 9. Example of a real landfill showing how the source geometry is modified to be rectangular and aligned with the groundwater flow direction in order to be simulated. The green lines show the 'real' extension of the units and the blue and yellow continuous lines the simulated geometries of each unit.

b. Affald-A multiple unit. Step by step tutorial

In the following we show a step by step example of how to use the model:

1) Open the "Affald-A multiple unit" window.
2) Under the field "Select the number of different units of the landfill" choose "5".
3) Click on "Load model inputs" and select the file "EX_4_inputs_AffaldA_MultipleUnits_5_Units.affald_a_multiple". This example contains 5 different units (Several examples are provided and the user can choose among them).
4) Click on "Load the input time series" and load the provided input file "EX_4_Source_inputs_AffaldA_MultipleUnits_5_Units.txt".
5) Let us call the case study "CASE_BBB" as shown in the following figure.

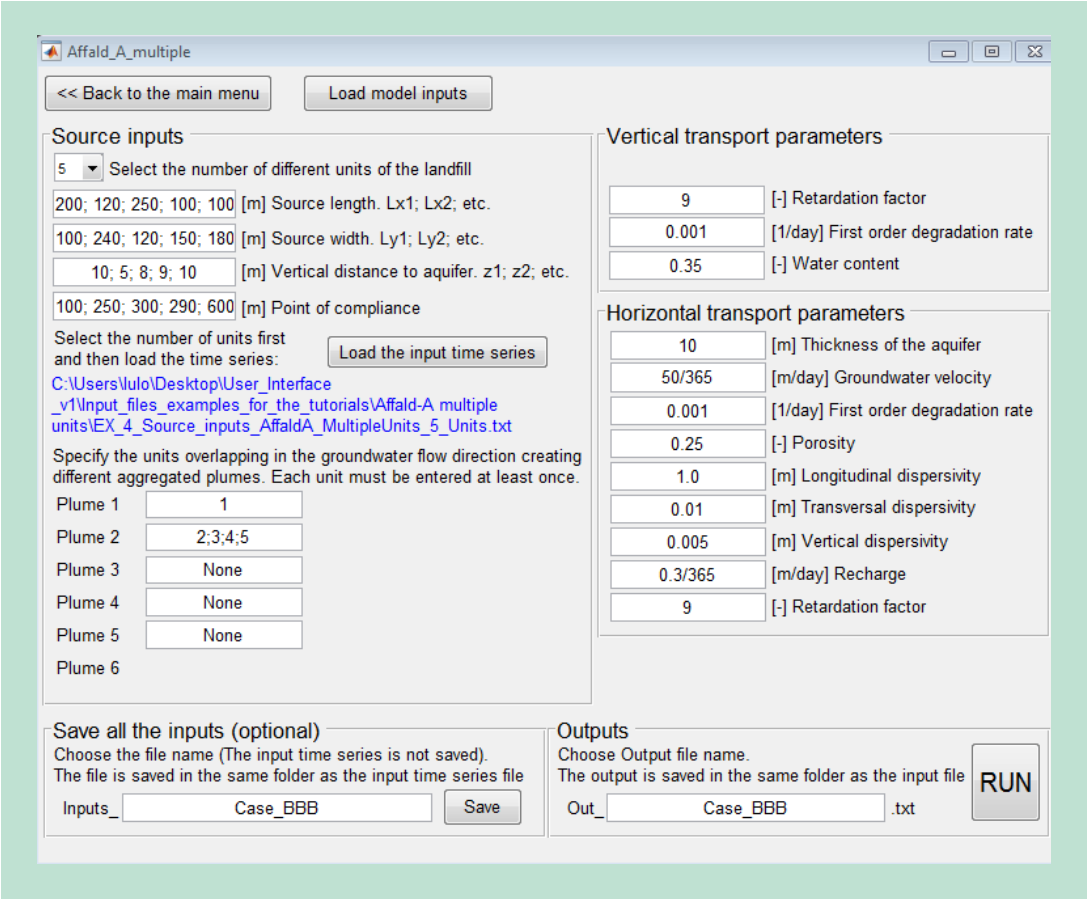


FIGURE 10. Screenshot for the step by step tutorial of the model ”Affald-A multiple units”. The user has called the case study “CASE_BBB”.

6) The user can save the inputs (this is optional) by pressing the “Save” button. Saved inputs can then be loaded with the “Load model inputs” button.
7) The user can change the input parameters (remember to use a dot as decimal separator and not a comma).
8) Run the model by pressing the button ‘RUN’ and wait until the results pop-up. The result window that pops up is shown in the following.

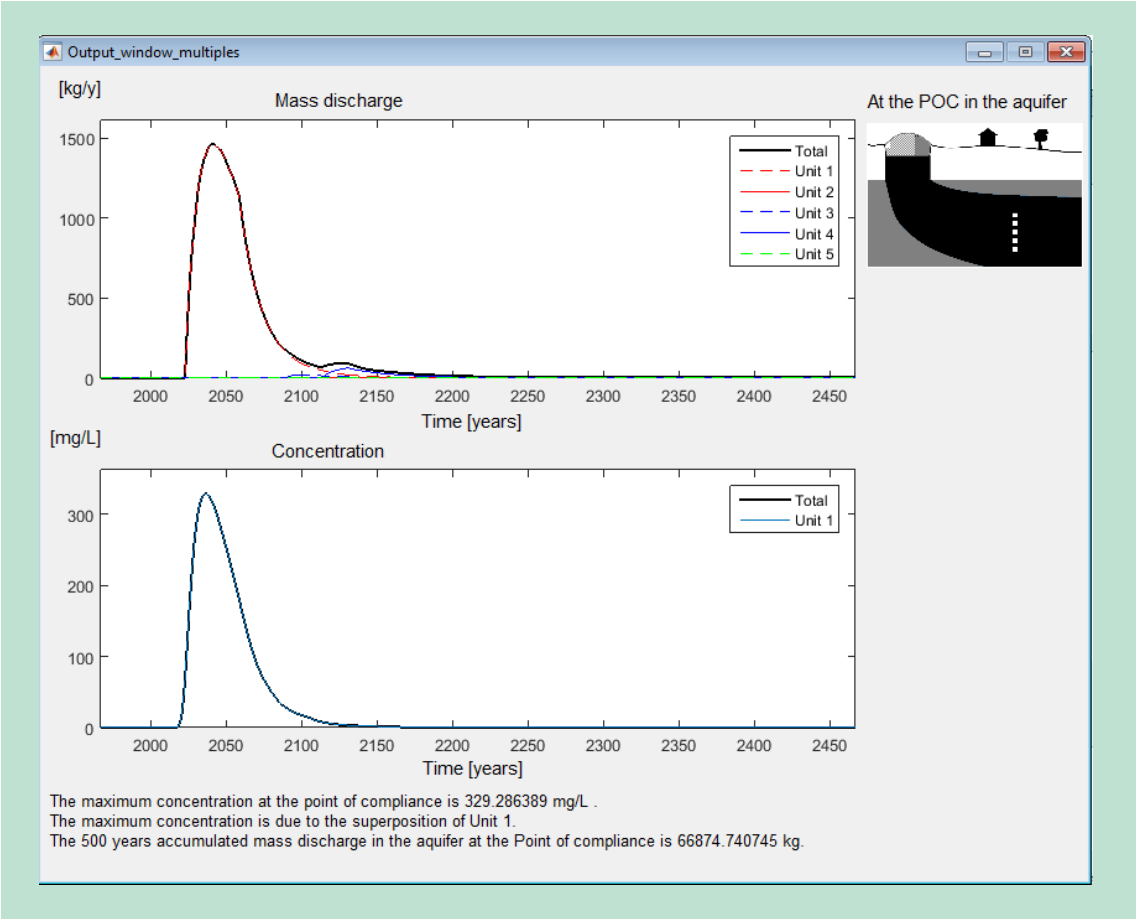


FIGURE 11. Example of a result window of “Affald-A multiple units”.

9) Reading the results from the pop-up window.
   The top graph shows the mass discharge in the aquifer at an infinite plane located at the Point of compliance. Both the total mass discharge and the contribution from each unit are shown.
   The bottom graph shows the highest concentration (over a 2 m long screen) in the aquifer. Both the total concentration and the contribution from each unit are shown. Nevertheless, it this case the maximum concentrations are given by Unit 1 (the overlapping of the other units produced smaller maximum concentrations).
   Note that the total mass discharge always results from all the units contributions, whereas the maximum concentrations can result from fewer units.
   At the bottom of the result window there are 3 sentences reporting the maximum concentration at the point of compliance, the units that produce the maximum concentrations and the 500 year accumulated contaminant mass discharge.
10) A result file called “Out_CASE_BBB.txt” is automatically created in the same folder as the input time series file. The output file contains all the output time series and it is shown in the following (the file can also be opened in Excel). The output time series (mass discharge and concentration) for each unit are found by scrolling down in the same file.

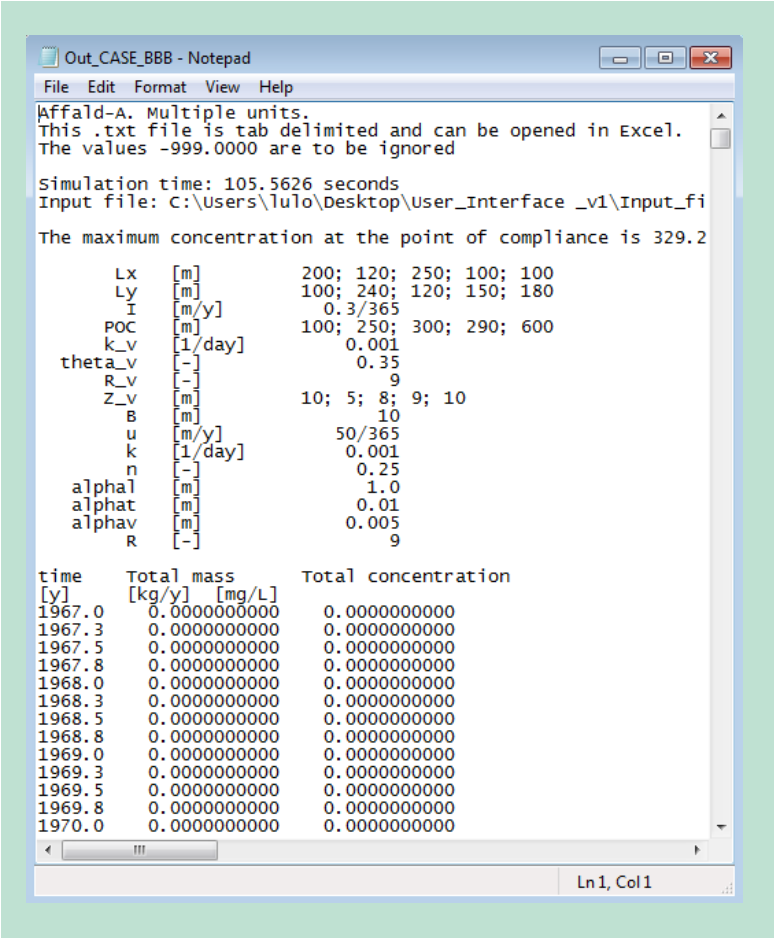


FIGURE 12. Example of an output .txt file from the model ”Affald-A mulitple units”

8. Affald-B single unit

The model “Affald-B single unit” (Figure 2) can be used to simulate concentration and mass discharge downstream a single landfill unit that is partially submerged in the aquifer (the bottom of the unit is below the top of the aquifer).

The interface of the model “Affald-B single unit” is shown in the following figure. The user has different options (Back to the main menu; Save all the inputs; Load model inputs).

First of all the user must enter the input time series. The user needs to press “Browse” within the “Source inputs” box. A new window pops up where the user needs to select the input .txt file. The input file must be created correctly according to the instructions given in the section “How to create the input .txt file”. If the input .txt file is made correctly the user will be able to see the 2 graphs of the input time series (water discharge and concentration) in the user interface.
Once the input time series is included then the user can save (this is optional) the input parameters. When all the input parameters are given the user can select the result file name and run the model by clicking on the “RUN” button. The model will run and automatically pop-up a result window and write the output .txt file in the same location as the selected .txt input file.

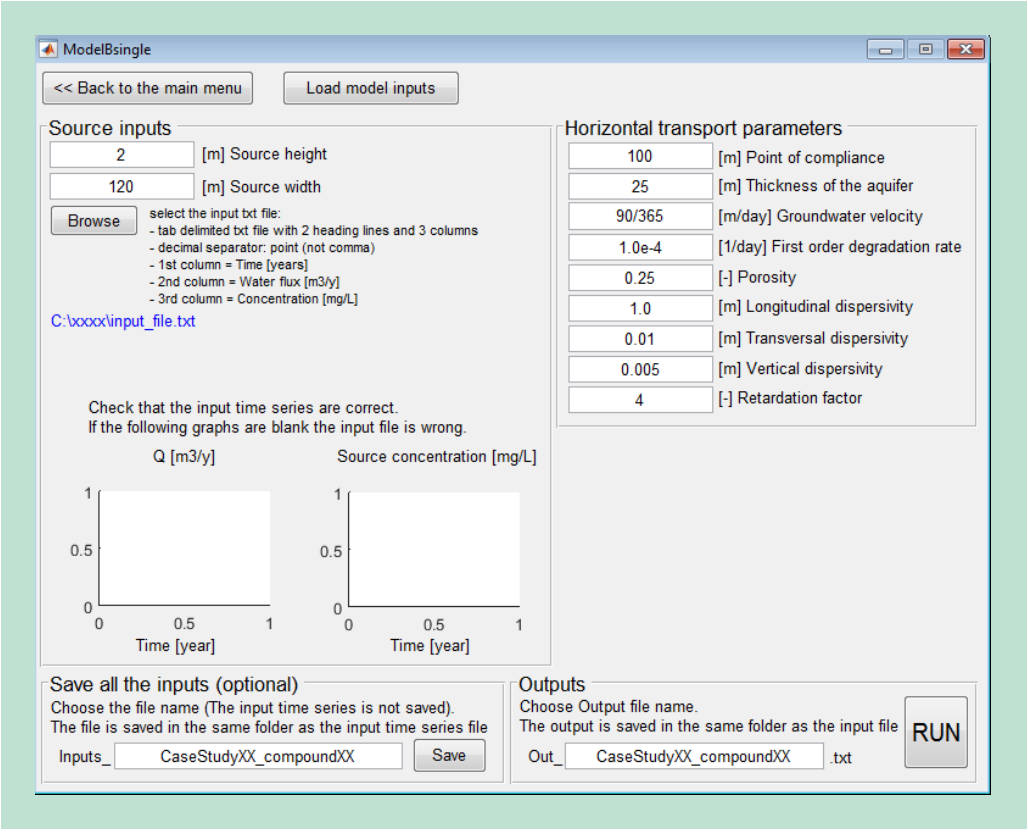


FIGURE 13. Screenshot of the model ”Affald-B single unit”.

c. Affald-B single unit. Step by step tutorial

In the following we show a step by step example of how to use the model:

9) Open the “Affald-B single unit” window.

10) Load the provided input file “EX_1_Source_inputs_AffaldB_SingleUnite_varyingQ_varyingC” (4 different examples are provided and the user can choose among them).
11) Let us call the case study “CASE_CCC” as shown in the following figure

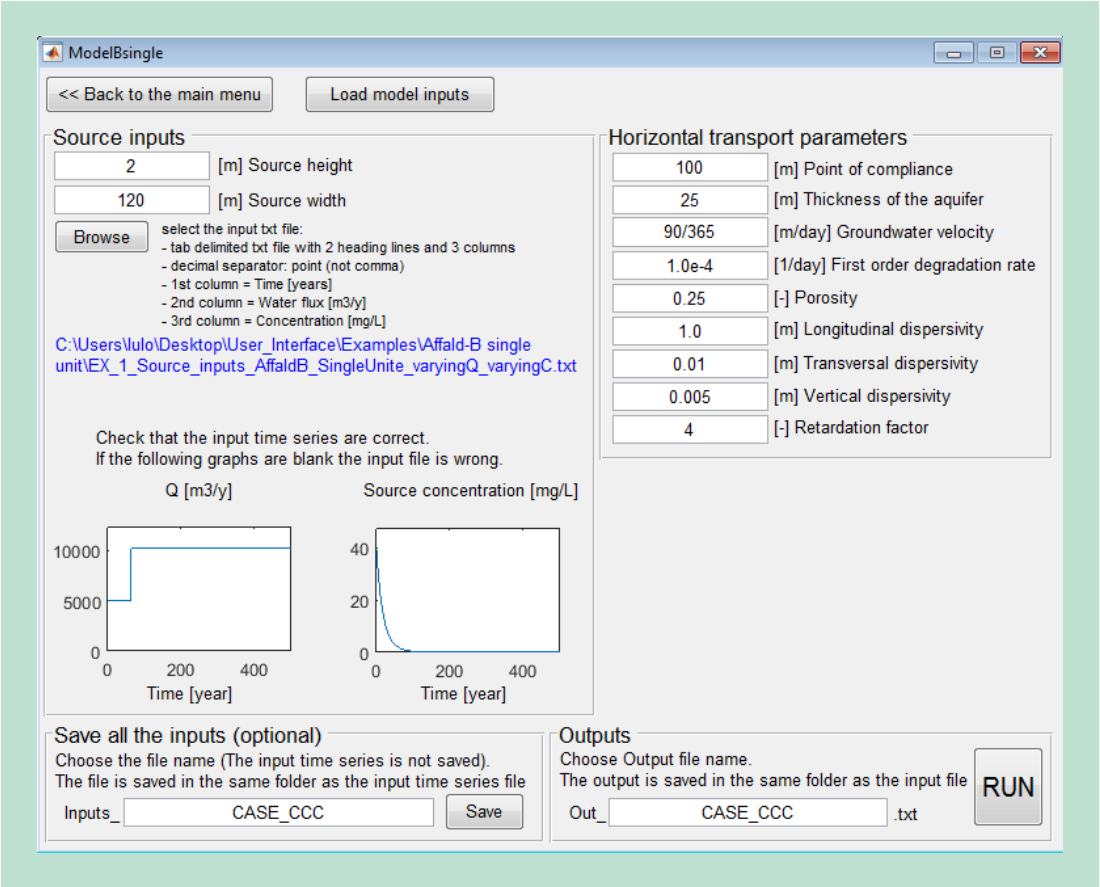


FIGURE 14. Screenshot for the step by step tutorial of model ”Affald-B single unit”. The input time series are properly displayed and the user has called the case study “CASE_AAA”.

12) The user can save the inputs (this is optional) by pressing the “Save” button. Saved inputs can be loaded with the “Load model inputs” button.
13) The user can change the input parameters (remember to use a dot as decimal separator and not a comma).
14) Run the model by pressing the button ‘RUN’ and wait until the results pop-up. The result window that pops up is shown in the following.

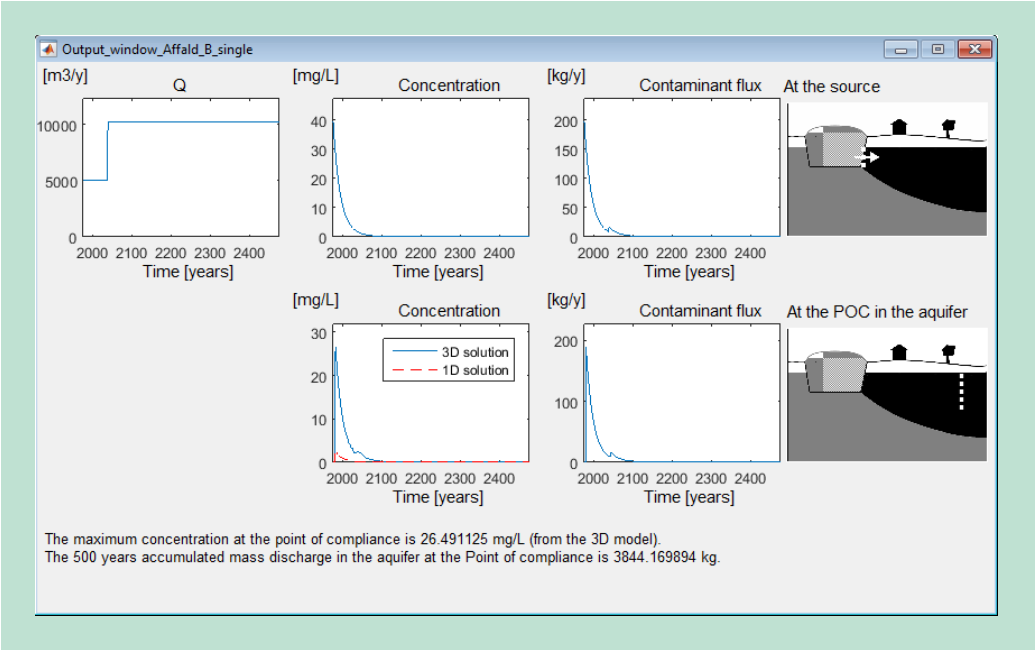


FIGURE 15. Example of a result window of Affald-B single unit.

15) Reading the results from the pop-up window.
The top 3 graphs show the water discharge Q, the concentration and the contaminant mass discharge at the source.
The 2 bottom graphs show the concentration over a 2 m well screen at the point of compliance and the contaminant mass discharge over an infinite plane (perpendicular to the groundwater flow direction) located at the point of compliance. The concentration at the POC is always computed using both a 3D model and a 1D model. The model giving the highest concentrations is the one to be used. The idea is that the 3D model is suitable for thick aquifers whereas in the case of thin aquifers the 1D model is more accurate (because the 3D model is designed for aquifers of infinite extent). Further details can be found in the main report.
At the bottom of the result window there are 2 sentences reporting the maximum concentration at the point of compliance and the 500 year accumulated contaminant mass discharge.
16) A result file called “Out_CASE_AAA.txt” is automatically created in the same folder as the input time series file. The output file contains all the output time series and it is shown in the following (the tab delimited .txt file can also be opened in Excel). Each column of the output file include a time series:
Column 1. PoC_t_3D. [y]. Time of the 3D model outputs of the horizontal transport.
Column 2. PoC_c_3D. [mg/L]. Concentration at the Point of compliance from the 3D model (the related time is in column 1).
Column 3. PoC_t_1D. [y]. Time of the 1D model outputs of the horizontal transport.
Column 4. PoC_c_1D. [mg/L]. Concentration at the Point of compliance from the 1D model (the related time is in column 3).
Column 5. PoC_Mass_1D. [kg/y]. Mass discharge at a plane at the Point of compliance distance from the 1D model (the related time is in column 3).

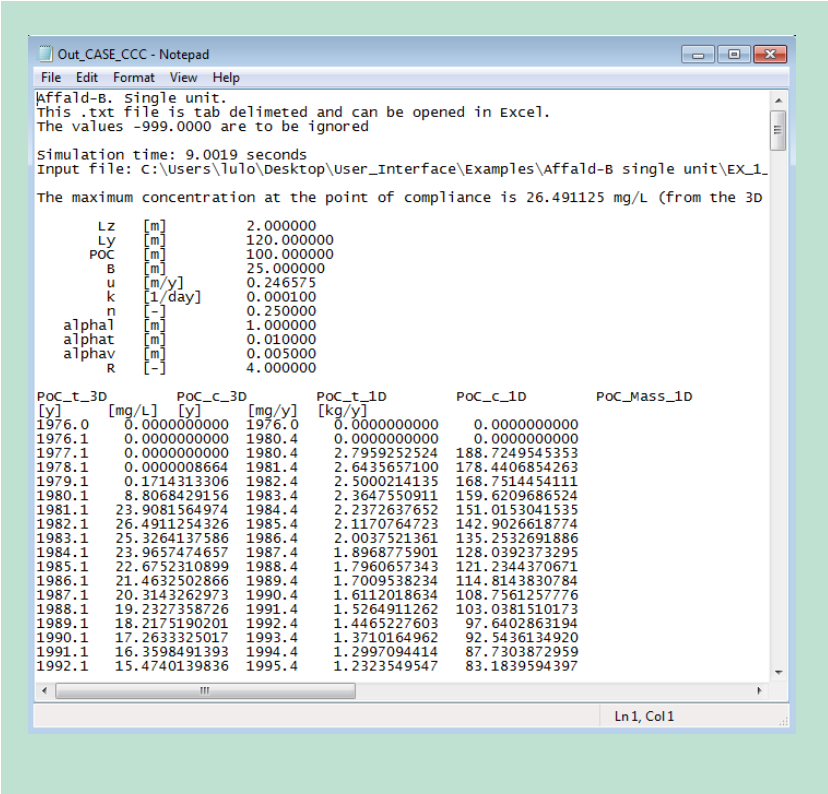


FIGURE 16. Example of an output .txt file from the model ”Affald-B single unit”.

9. Affald-B multiple units

The model “Affald-B multiple units” (Figure 2) can be used to simulate concentration and mass discharge downstream a landfill made of several units that are partially submerged in the aquifer (the bottom of the landfill is below the top of the aquifer).
The interface of the model “Affald-B multiple units” is shown in the following figure. The user has different options (Back to the main menu; Save all the inputs; Load model inputs).

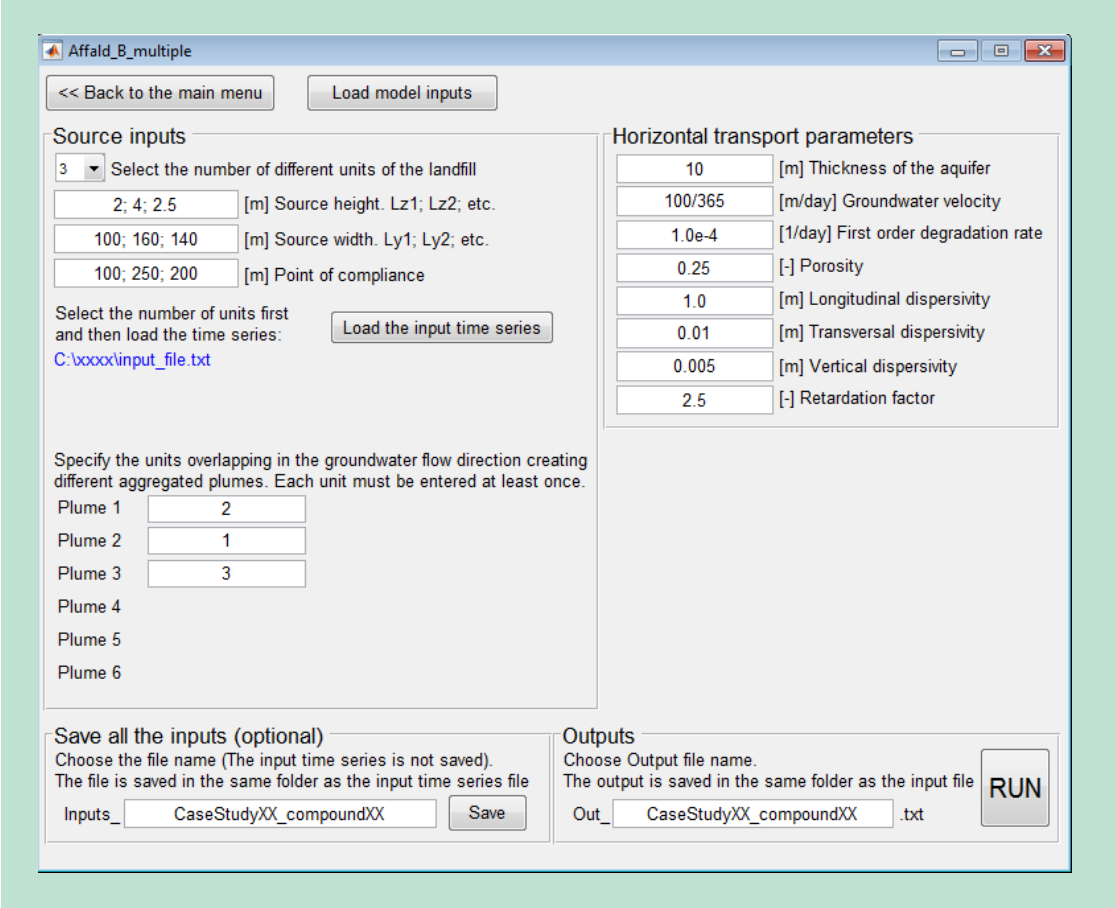


FIGURE 17. Screenshot of the model ”Affald-B multiple units”.

First of all the user must enter the right number of units to be simulated using the drop-down menu in the “Source inputs” window. Then the fields “Source height”, “Source width” and “Point of compliance” must be entered. Note that there must be the same number of entries as the number of units (i.e. if there are 4 units then there must be 4 numbers in the field “Source length” and in the other fields). The number of entries must be separated by a semicolon. For instance, the landfill of Faaborg (one of the examples included Miljøstyrelsen, 2018) has 2 different units and the following figure shows how to input the 2 unit geometries, distances to the PoC, etc.

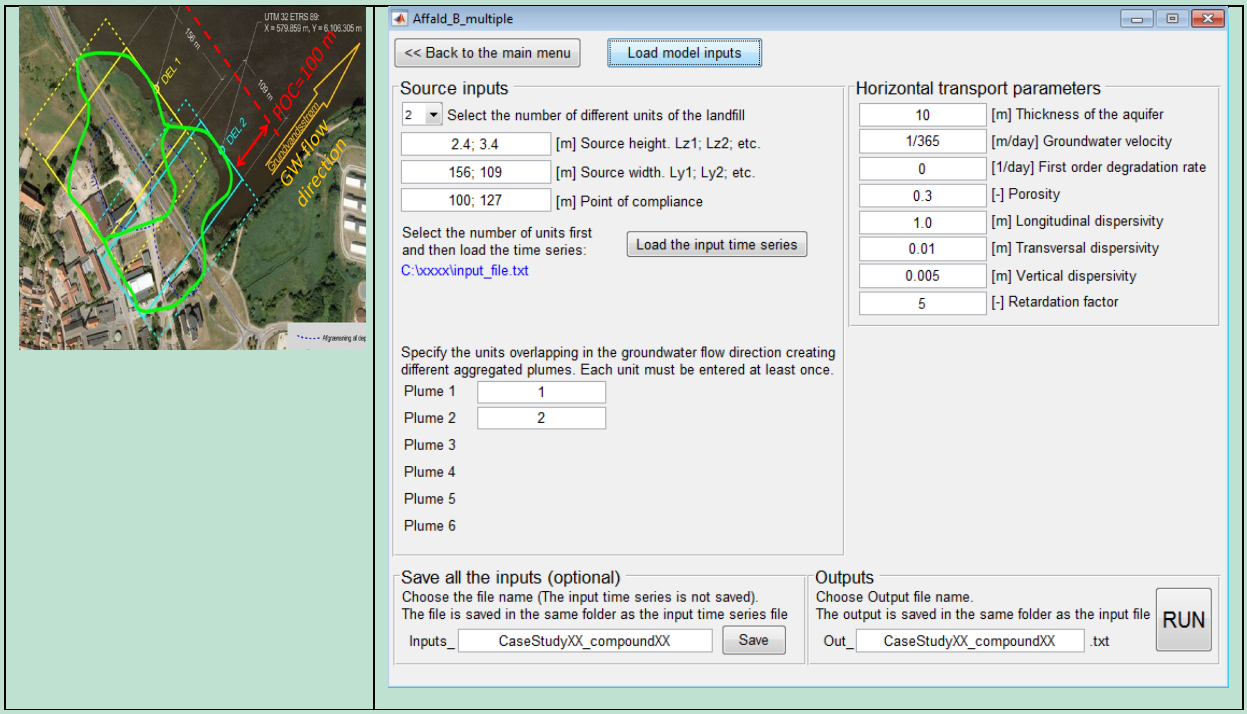


FIGURE 18. Example of how to enter the 2 units of the landfill of Faaborg. The input data are provided together with the model interface and are located in the folder "Input_files_examples_for_the_tutorials"'

Now the user must enter the input time series by pressing the "Browse" button within the "Source inputs" box. A new window will open where the user needs to select the input .txt file. The input file must be created correctly according to the instructions given in the section "How to create the input .txt file".
Afterwards the user needs to specify how the different source units overlap in the groundwater flow direction. A part of Section 7 shows how to do it.
Once the input time series is included then the user can save (this is optional) the input parameters. When all the input parameters are given the user can select the result file name and run the model by clicking on the "RUN" button. The model will run and automatically pop-up a result window and write the output .txt file in the same location as the selected .txt input file.

d. Affald-B multiple units. Step by step tutorial

In the following we show a step by step example of how to use the model:

11) Open the "Affald-B multiple unit" window.
12) Under the field "Select the number of different units of the landfill" choose "3".
13) Click on "Load model inputs" and select the file "EX_2_inputs_AffaldB_MultipleUnites_3_Units". This example contains 3 different units (several examples are provided and the user can choose among them).
14) Click on "Load the input time series" and load the provided input file "EX_2_Source_inputs_AffaldB_MultipleUnites_3_Units.txt".
15) Let us call the case study "CASE_DDD" as shown in the following figure.

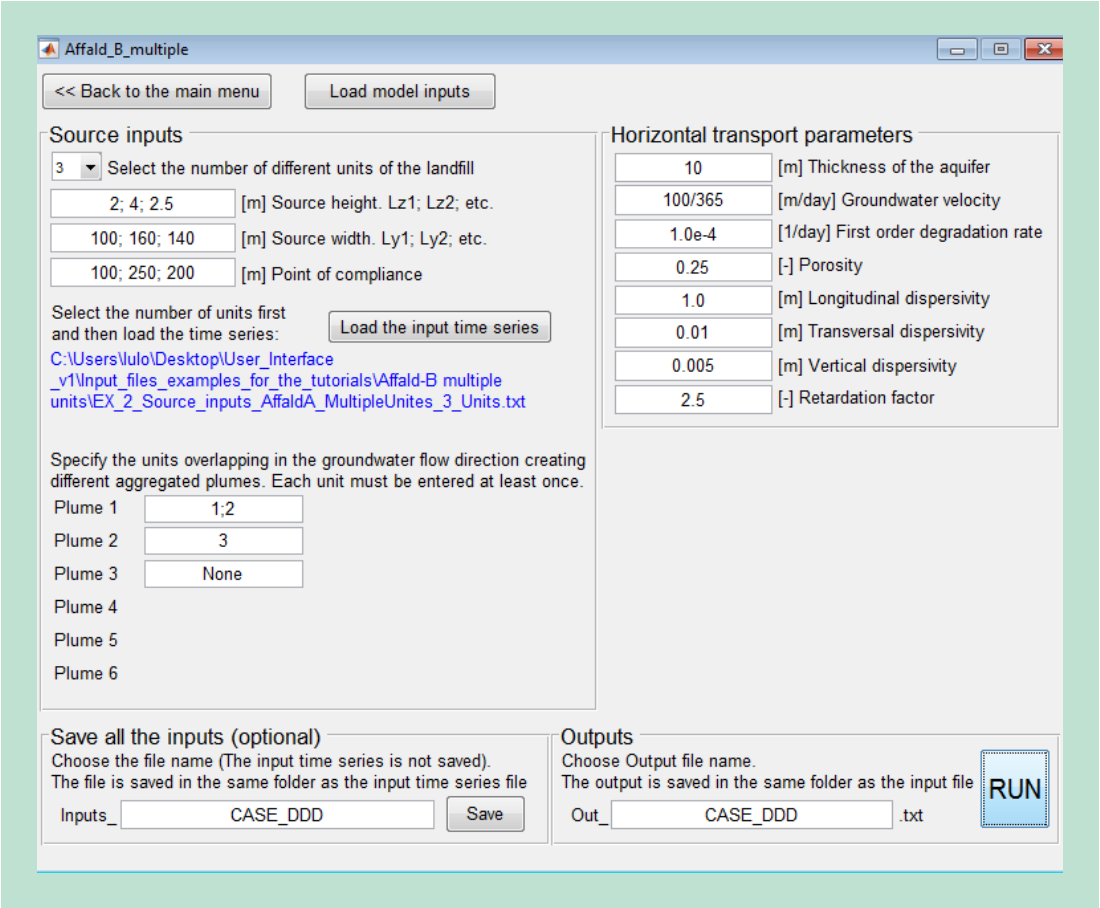


FIGURE 19. Screenshot for the step by step tutorial of model ”Affald-B multiple units”. The user has called the case study “CASE_DDD”.

16) The user can save the inputs (this is optional) by pressing the “Save” button. Saved inputs can then be loaded with the “Load model inputs” button.
17) The user can change the input parameters (remember to use a dot as decimal separator and not a comma).
18) Run the model by pressing the button ‘RUN’ and wait until the results pop-up. The result window that pops up is shown in the following.

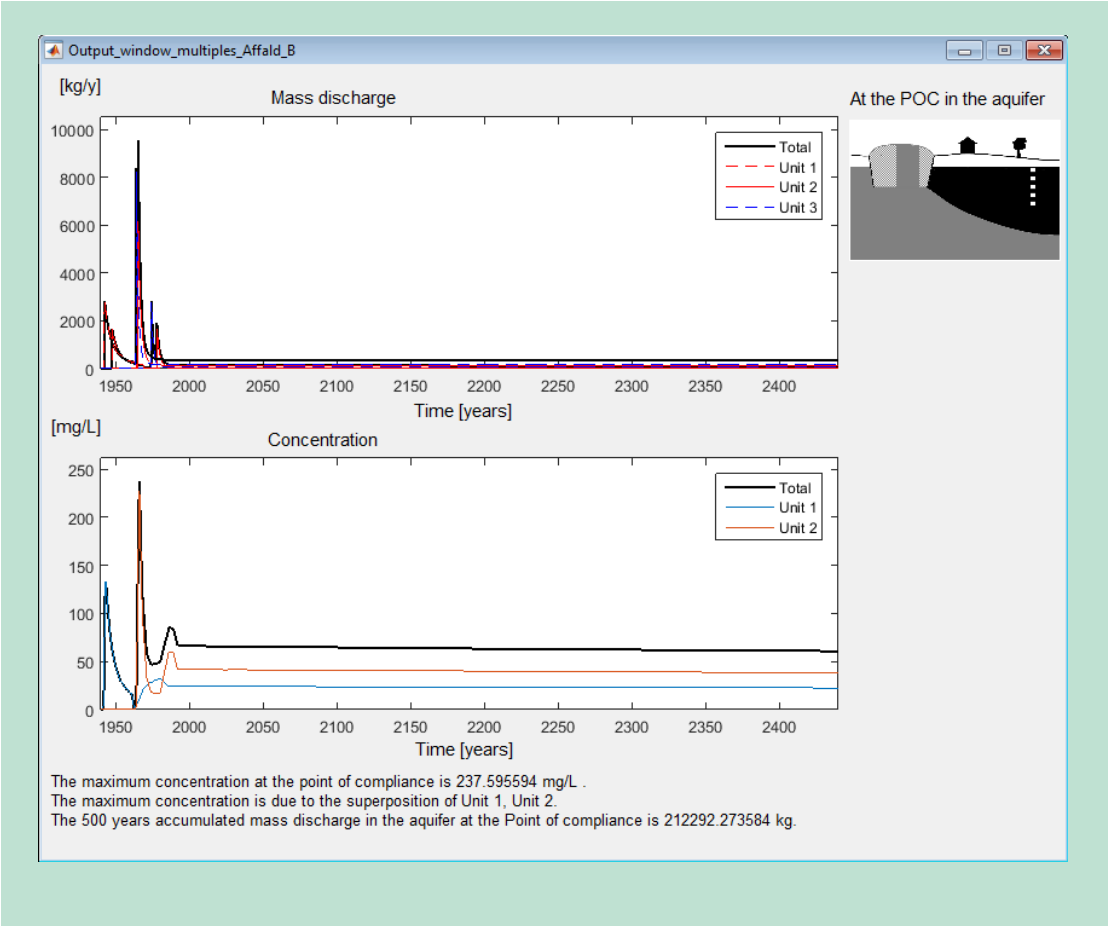


FIGURE 20. Example of a result window of “Affald-B multiple units”.

19) Reading the results from the pop-up window.
The top graph shows the mass discharge in the aquifer at an infinite plane located at the Point of compliance. Both the total mass discharge and the contribution from each unit are shown.
The bottom graph shows the highest concentration (over a 2 m long screen) in the aquifer. Both the total concentration and the contribution from each unit are shown. Nevertheless, in this case the maximum concentrations are given by the superposition of Unit 1 and Unit 2.
Note that the total mass discharge always results from all the units contributions, whereas the maximum concentrations can result from fewer units.
At the bottom of the result window there are 3 sentences reporting the maximum concentration at the point of compliance, the units that produce the maximum concentrations and the 500 year accumulated contaminant mass discharge.
20) A result file called “Out_CASE_DDD.txt” is automatically created in the same folder as the input time series file. The output file contains all the output time series and it is shown in the following (the file can also be opened in Excel). The output time series (mass discharge and concentration) for each unit are found by scrolling down in the same file.

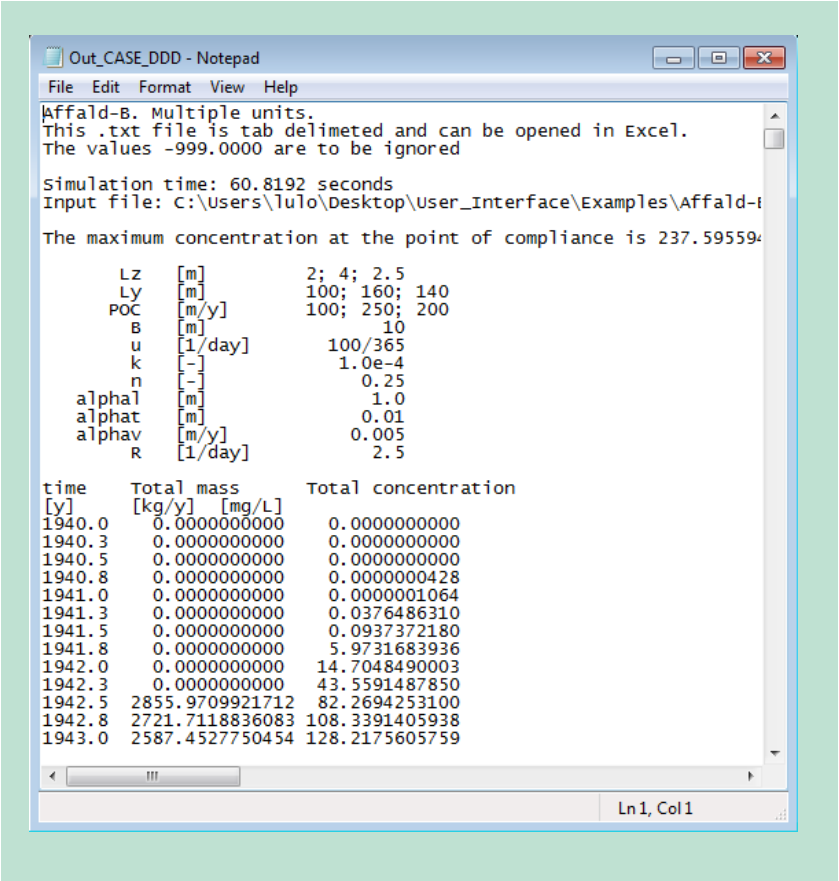


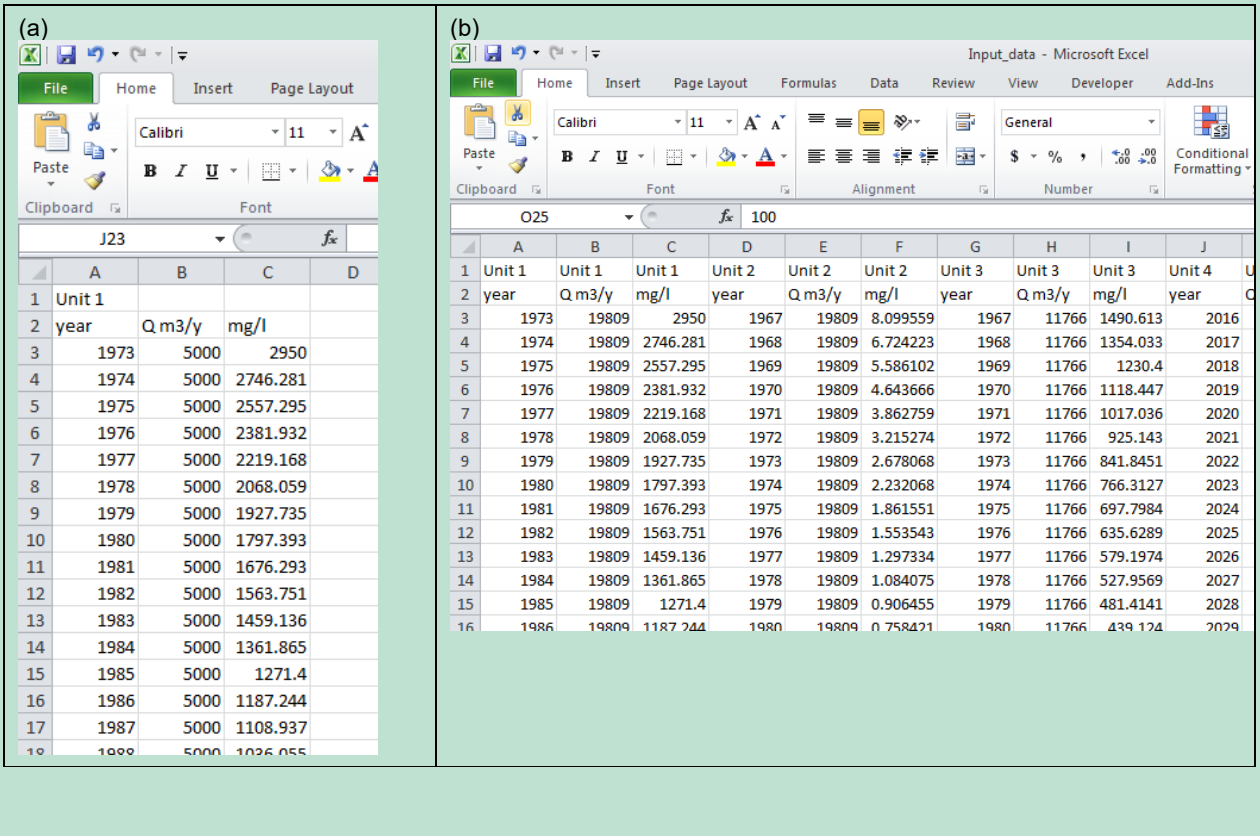
FIGURE 21. Example of an output .txt file from the model ”Affald-B mulitple units”.

10. How to create the input .txt file

The models require the input time series of water discharge and contaminant concentration from the landfill that are obtained with the Kildestyrkemodel (2017). The models Affald-A and Affald-B require a .txt file with these time series. The .txt file must be created respecting the following requirements:

- There must be 2 heading lines and 3 columns for each contaminant unit
- The decimal separator must be a dot (not a comma)
- The 1st column is the time [years]
- The 2nd column is the water discharge [m^3/y]
- The 3rd column is the concentration [mg/L]
- The input time series must be 500-year long and with 1 year time step (501 lines in total)

The user can open with Excel the tab delimited input .txt files provided with the tutorials and see how the different time series are entered.
The following steps show an example of how to create the input file:
1) Copy the output time series of Kildestyrkemodel into a new Excel spreadsheets following the requirements stated above. The following figure shows 2 examples of how a spreadsheet should look like in the case of a single unit source and in the case of multiple units.



(a)

	A	B	C	D
1	Unit 1			
2	year	Q m3/y	mg/l	
3	1973	5000	2950	
4	1974	5000	2746.281	
5	1975	5000	2557.295	
6	1976	5000	2381.932	
7	1977	5000	2219.168	
8	1978	5000	2068.059	
9	1979	5000	1927.735	
10	1980	5000	1797.393	
11	1981	5000	1676.293	
12	1982	5000	1563.751	
13	1983	5000	1459.136	
14	1984	5000	1361.865	
15	1985	5000	1271.4	
16	1986	5000	1187.244	
17	1987	5000	1108.937	
18	1988	5000	1026.055	

(b)

	A	B	C	D	E	F	G	H	I	J
1	Unit 1	Unit 1	Unit 1	Unit 2	Unit 2	Unit 2	Unit 3	Unit 3	Unit 3	Unit 4
2	year	Q m3/y	mg/l	year	Q m3/y	mg/l	year	Q m3/y	mg/l	year
3	1973	19809	2950	1967	19809	8.099559	1967	11766	1490.613	2016
4	1974	19809	2746.281	1968	19809	6.724223	1968	11766	1354.033	2017
5	1975	19809	2557.295	1969	19809	5.586102	1969	11766	1230.4	2018
6	1976	19809	2381.932	1970	19809	4.643666	1970	11766	1118.447	2019
7	1977	19809	2219.168	1971	19809	3.862759	1971	11766	1017.036	2020
8	1978	19809	2068.059	1972	19809	3.215274	1972	11766	925.143	2021
9	1979	19809	1927.735	1973	19809	2.678068	1973	11766	841.8451	2022
10	1980	19809	1797.393	1974	19809	2.232068	1974	11766	766.3127	2023
11	1981	19809	1676.293	1975	19809	1.861551	1975	11766	697.7984	2024
12	1982	19809	1563.751	1976	19809	1.553543	1976	11766	635.6289	2025
13	1983	19809	1459.136	1977	19809	1.297334	1977	11766	579.1974	2026
14	1984	19809	1361.865	1978	19809	1.084075	1978	11766	527.9569	2027
15	1985	19809	1271.4	1979	19809	0.906455	1979	11766	481.4141	2028
16	1986	19809	1187.244	1980	19809	0.758421	1980	11766	439.124	2029

FIGURE 22. Examples of how to create the input time series for the models Affald-A and Affald-B in Excel. (a) The case of a single unit source. (b) The case of multiple unit sources.

1) Save the Excel file as “Text (Tab delimited)”. See the following figure.

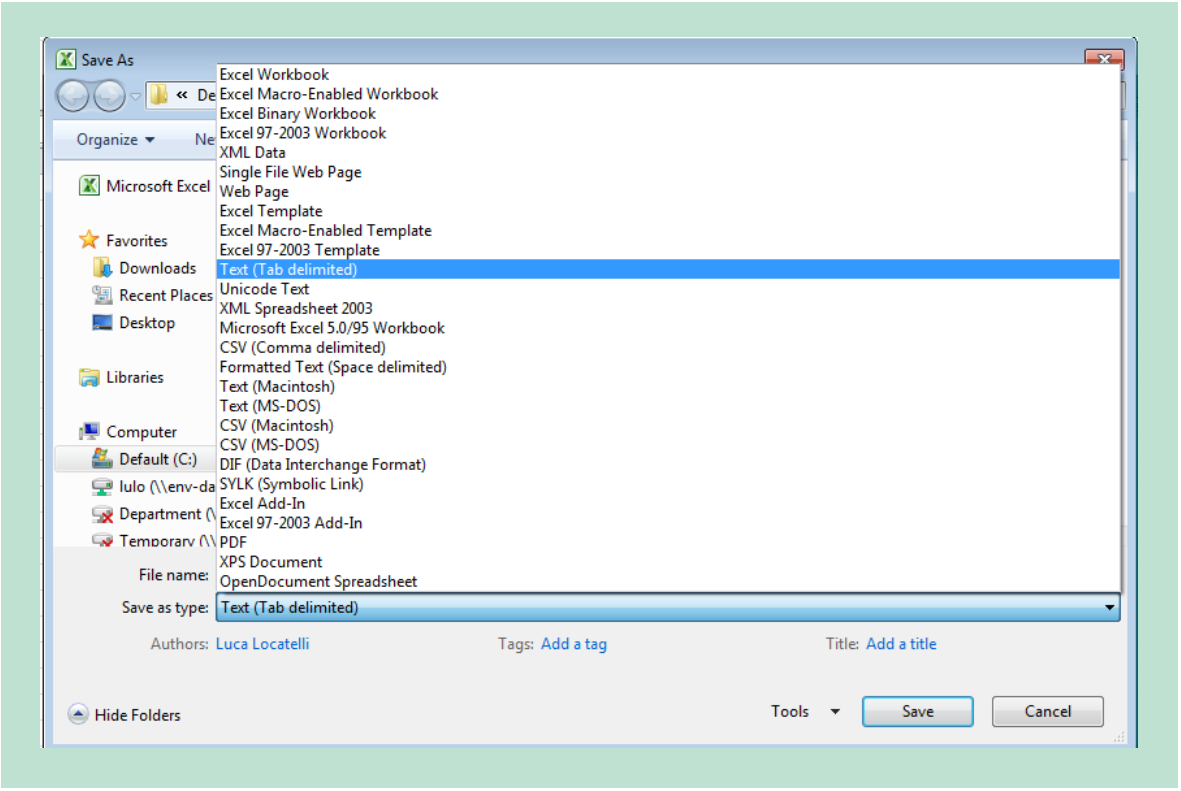


FIGURE 23. Example of how to save the Excel file as “Text (Tab delimited)”.

2) Check that the .txt file looks like the one shown in the following figure.

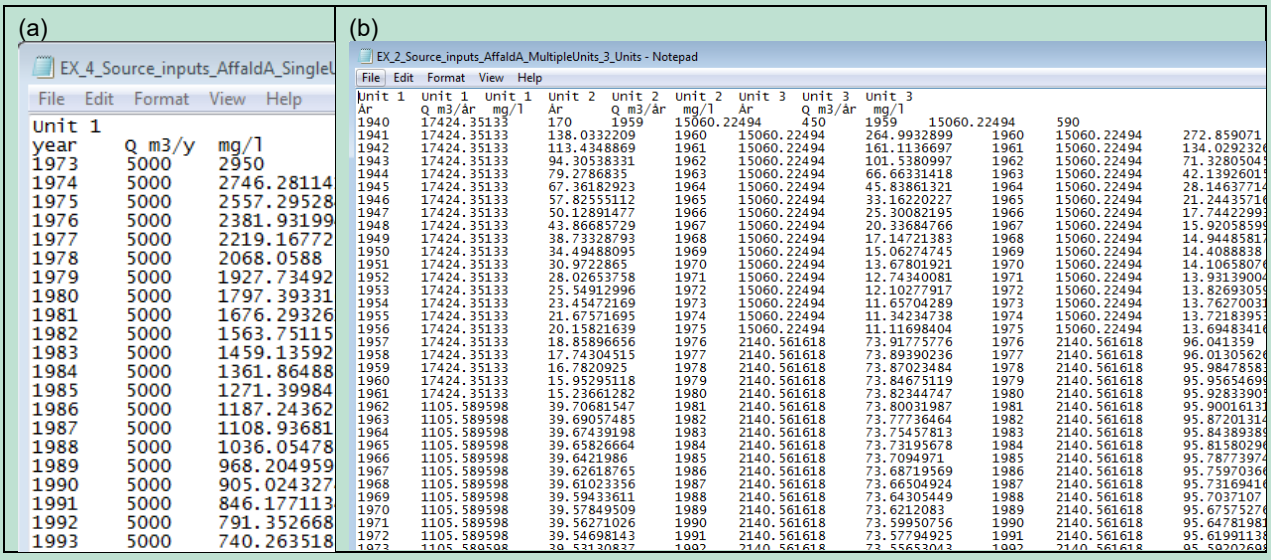


FIGURE 24. Examples of how the .txt input file should look like. (a) The case of a single unit source. (b) The case of multiple unit sources.

11. Frequently asked questions.

Here is a list of questions/issues/discussions that the users might find useful.

1) QUESTION. The output concentration is not smooth. Instead, it is made of several small steps like the one shown below.

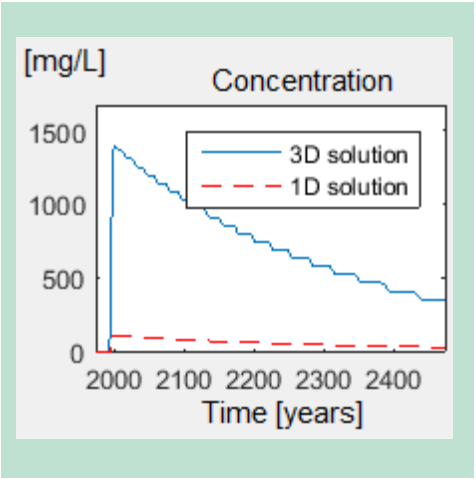


FIGURE 25. Examples of an output concentration that is not smooth.

ANSWER. The program in order to run fast discretizes the input source functions. It can happen in some cases that the input function discretization is somewhat rough and the output looks like the one in the figure. The error introduced by the rough discretization is generally insignificant.

2) QUESTION. The output concentration from the 3D model has a flat peak like the one shown in the figure below.

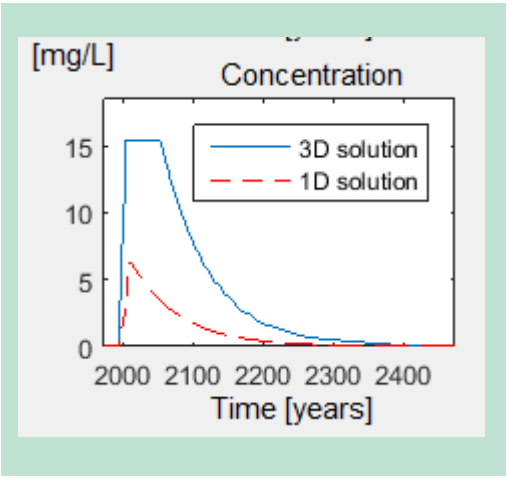


FIGURE 26. Examples of an output concentration from the 3D model that shows a flat peak.

ANSWER. The 3D model assumes that the input water from the landfill do not affect the water balance of the aquifer (Nevertheless, landfills are generally large and can affect the water balance of an aquifer). Therefore, the model computes the concentrations in the aquifer excluding the dilution given by the additional water coming from the landfill. In some cases the model gives concentrations that are higher than the input concentrations and this is physically unacceptable. Concentrations in the aquifer that are higher than the input concentrations are excluded. The contaminant mass discharge in the aquifer is correct since it is computed using a different model. See the main report (DEPONIER RAPPORT 2017) for further details and discussion about the models.

3) QUESTION. How is it possible that the 1D model results in much lower concentrations compared to the 3D model?
ANSWER. The 1D model is based on different assumptions compared to the 3D model, i.e. it assumes that the contaminant is uniformly distributed over the depth of the aquifer. This assumption is valid only in the case of “thin” aquifers. Therefore if the aquifer is “thick” the 1D model can give much smaller concentrations compared to the 3D model. See the main report (Miljøstyrelsen, 2018) for further details and discussion about the models.

4) QUESTION. Why is the output concentration from the model constant as a function of the distance downstream the landfill?
ANSWER. The user might experience that the output concentration of the 1D model does not change even if different PoC distances are entered (i.e. the concentration is the same at PoC=100m and PoC= 600m). This happens in the case of thin aquifers (1D model) and when there is no degradation. This is because the 1D model does not include dispersion. Conceptually, this is reasonable for landfills discharging in thin aquifers, in fact it will require very large distances before dispersion will have an effect and in most of the cases the PoC distances are relatively small.

References

Udvikling af metodik til risikovurdering ved deponering af affald. Delopgave 2 – Stoftransport i jord og grundvand: Retningslinjer for opstilling af numerisk model. Lizzi Andersen, Erik Aagaard Hansen, Ole Hjelmar, René Møller Rosendal, Steen Stentsøe. Miljøstyrelsen, Miljøprojekt nr. 2117, Oktober 2021

GrundRisk Landfill. Transport of contaminants released from landfills – a part of a risk assessment tool. Miljøstyrelsen, Environmental Project no. 2181, October 2021

GrundRisk Landfill - User Guideline

The Danish Environmental
Protection Agency
Tolderlundsvej 5
DK - 5000 Odense C

www.mst.dk