Trapezoidal Earth Dam with Drainage Toe
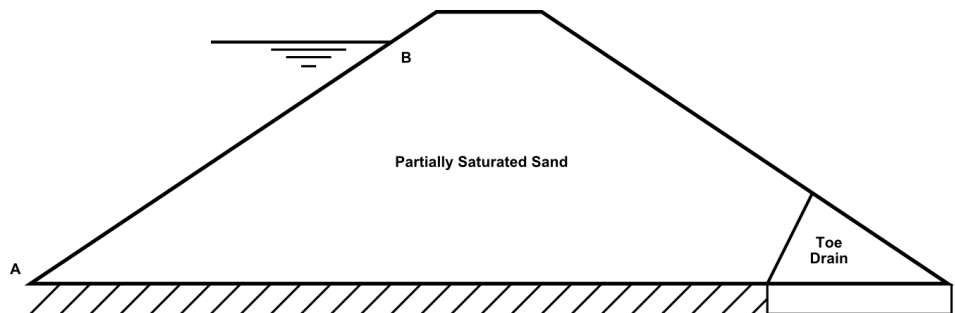
For LUSAS version:	17.0
For software product(s):	Any Plus version
With product option(s):	Nonlinear, Dynamic, Thermal

Description

This example illustrates water seepage through a trapezoidal earth dam to drain in the granular filter in the downstream toe. The flow unconfined, with the dam built on an impermeable foundation.

On the upstream side, the water is at the reservoir level of 12m which results in equipotential line along AB with maximum head of 12m. The filter drain on the downstream side is considered fully effective.

The modelling units used are N, m, kg, s, C throughout.

Objectives

The required output from the analysis consists of:

❑ The calculation of steady-state seepage flow through the body of the dam.

❑ Contours of pore water pressures.

Keywords

2D, Inplane, Dam, Pore Pressure, Two Phase Material, Consolidation, Displacement reset (Geostatic) step, Transient, Seepage

Associated Files

❑ **earth_dam_modelling.vbs** carries out the modelling of the example.

Discussion

Not all the flow networks of an earth dam are known from the beginning. The free flow surface, or top flow line (where the pressure is atmospheric), can only be approximated when using hand methods. The uppermost flow line inside an earth dam follows a parabolic path, which is corrected to be tangential to the upstream slope and discharge surface, as well as at different material boundaries. Earth dams almost always include drainage filters (having a large value of permeability) positioned in the downstream slope to keep seepage entirely within the dam, and prevent erosion of the downstream surface.

There are two basic assumptions in such analyses:

a) Between two consecutive points of equipotential lines in contact with the free surface there is a constant piezometric difference

b) At the point that the flow from the upstream side to the downstream side begins, the upper most flow line is positioned perpendicular to the upstream slope of the dam

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller see the heading *Running LUSAS Modeller* in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command

File>New to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.
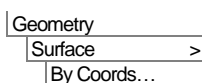
Creating a new model

- Enter a File name of **Earth_dam**

- Use the default User-defined working folder.

- Ensure an Analysis type of **Structural** is set.

- Select an Analysis Category of **2D Inplane**

- Set Model units of **N,m,t,s,C**

- Leave the Timescale units as **Days**

- Select a Startup template of **None**.

- Ensure the Layout grid is set as **None**

- Enter a Title of **Trapezoidal earth dam with drainage toe**
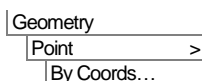
- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.
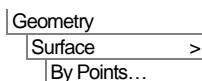
Defining the Geometry

Geometry	
Surface	>
By Coords…	

Enter the following coordinates to define the partially saturated sand part of the dam.

X	Y
0.0	0.0
17.8064	12.0
19.0345	12.8276
23.5862	12.8276
36.2038	4.34484
33.5172	0.0

Geometry	
Point	>
By Coords…	

Enter the following coordinates (**42.6207, 0.0**) to define the downstream toe point.

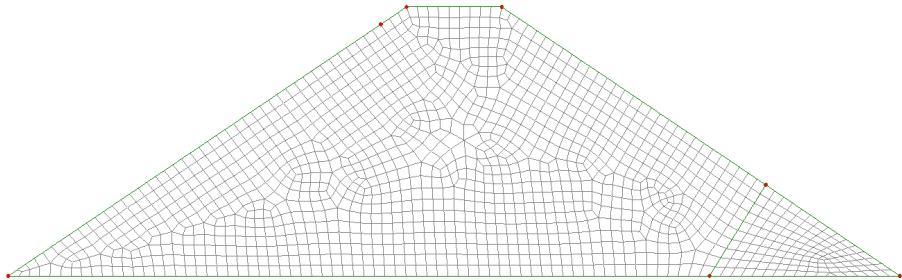| Geometry	
Surface	>
By Points…	

Holding down the **P** key to ensure only points are selected, box-select the three points that form the bottom-right corner (Points 7, 6 and 5) in order to create the filter discharge surface.

Defining and Assigning Mesh Attributes

Attributes
Mesh   >
  Surface…

- Select **Plane strain two phase** elements types with **Quadrilateral** shape and **Quadratic** interpolation order (QPN8P elements).

- Unselect **Automatic** mesh, tick **Element size** and enter **0.5**

- Enter the attribute name as **QPN8P 0.5m** and click **OK**

- Select the two surfaces on the model and drag and drop the **QPN8P 0.5m** mesh from the Treeview onto both surfaces.

Material Properties

Attributes
Material   >
  Isotropic...

Two different materials need to be defined; one for the partially saturated sand and one for the filter. For this example both materials need two phase properties defined.

Two-phase material properties are required when performing an analysis in which two-phase elements are used to define a drained and undrained state for soil.

To define partially saturated sand:

- With the **Elastic** tab selected enter a Young's modulus **1.0E9**, Poisson's ratio **0.2** and Mass density **2.0E3**.

- Select (tick) the **Two phase** option. On the Two Phase page ensure the **Partially saturated** option is selected, and enter the Bulk modulus of fluid phase as **1.0e9**, the Porosity of medium as **0.333333**, the Unit weight of fluid as **10.0e3** the Hydraulic conductivity in every global direction as **0.152e-3**, the Density of the fluid as **1.0e3** and the Degree of saturation to be considered as fully saturated as **1.0**.

Isotropic						
☐ Plastic	☐ Creep	☐ Damage	☐ Shrinkage	☐ Viscous	☑ Two phase	

Elastic | Two Phase |

○ Fully saturated
● Partially saturated
○ Piecewise linear partially saturated

Define curves...

☐ Incompressible solids

	Value
Bulk modulus of solid phase	0
Bulk modulus of fluid phase	1e9
Porosity of medium	0.333333
Unit weight of fluid	10e3
Hydraulic conductivity in global X direction	0.152e-3
Hydraulic conductivity in global Y direction	0.152e-3
Hydraulic conductivity in global Z direction	0.152e-3
Density of fluid	1e3
Irreducible saturation	0
Degree of saturation to be considered as fully saturated	1

Name [   ] ▼ ⬍ (new)

[OK] [Cancel] [Apply] [Help]

Note: Usually the value of Bulk modulus during the solid phase is substantially larger than the fluid phase, and therefore does not typically form part of the geotechnical testing. If the bulk modulus of solid phase is left as **0**, LUSAS assumes an incompressible solid phase.

Note: All time dependent material units (e.g. Hydraulic conductivity) are entered as model units (i.e. seconds). The timescale units are only used when defining analysis properties, such as nonlinear timesteps. The units expected by each input are shown as a tooltip.

- Press **Define Curves** to define a new pressure-effective saturation curve.

- Enter the Rate of water extraction as **1.3774**, the Weight factor as **1.0** and the Air entry to **-0.261097**.

- Press **OK** to leave the curve definition dialog.

- Enter the **Name** of the newly defined material as **Partially Saturated Sand** and press **Apply**

To define granular fill material:

- To define the granular filter material; in the Two Phase tab, change the Hydraulic conductivity to **1.0** in all three global (X, Y and Z) directions.

- Change the attribute name to **Filter** and press **OK**

- Assign the two attributes to their corresponding surfaces by dragging and dropping them onto the relevant features.

Loadcases

The geometry specified for the dam represents the as-constructed state, prior to any settlement and the filling of the reservoir. A geostatic loadcase will be used to model the initial stresses in the earth structure (due to self-weight) without any deformation. A second loadcase will then be used to apply the hydrostatic water pressure to the 'as-constructed' dam.

- In the 🕒 Treeview right-click on **Loadcase 1** and click rename. Change the name to **Initial Condition**

Analyses
  Loadcase...

- Add a second loadcase, changing the name to **Seepage** and press **OK**.

Note: Gravity is not assigned to this loadcase. Gravity will be assigned later as a property of the Analysis.

Supports

Using two consecutive loadcases requires support attributes to be defined to both in order to add and remove the restraint to the model for each stage.

Initial condition

During the Initial Condition pore-water will be allowed to flow freely through the dam to prevent the build-up of pore pressure during the construction stage.

Attributes
  Supports…

- Leaving all other degrees of freedom **Free**, set Pore pressure to **Fixed**. Enter the name of the support as **Free Flow** then press **Apply** to define a constant (zero) pressure attribute.

- Switch the X and Y translation freedoms to **Fixed**. Change the attribute name to **Fixed XY (Free Flow)** and press **OK** to define a fixed structure support.

Once the support attributes have been defined, they must be assigned to define the initial condition.

- In the 🕒 Treeview right-click on **Initial Condition** and press **Set active**

- Select both surfaces and, from the 🔴 Treeview, drag and drop the support attribute **Free Flow** onto the model. Press **OK** to assign the support.

- Now select only the two bottom lines of the dam, and assign the **Fixed XY (Free Flow)** support attribute. Press **OK** to assign the support.

Seepage stage

During the seepage stage the pore pressure will be allowed to build-up within the structure, with flow restricted to the drainage filter material in the downstream toe.

Attributes
  Supports…

- Leaving all degrees of freedom **Free** (including Pore pressure). Enter the name of the support as **Seepage Flow** and press **Apply** to define a 'freeing' support.

- Change the X and Y translation freedoms to **Fixed** and enter the name as **Fixed XY (Seepage Flow)**. Press **OK** to define a structural support that restricts flow.

Nonlinear analyses inherit support conditions from the previous stage. These new support attributes are therefore used to free the Pore pressure fixity where required.

- In the 🕒 Treeview right-click on **Seepage** and press **Set active**

- Select both surfaces and, from the 🔴 Treeview, drag and drop the support attribute **Seepage Flow** onto the model.

- In the Assign support dialog choose the **From loadcase (Nonlinear and Transient Analysis)** option, and select the **Seepage** loadcase. Press **OK** to assign.

- Now select only the left-hand bottom line of the dam (i.e. not within the filter), and assign the **Fixed XY (Seepage Flow)** support attribute, the ensuring **From loadcase (Nonlinear and Transient Analysis)** option and **Seepage** loadcase are selected in the Assign support dialog.

There is no need to assign a new support attribute to the line at the bottom of the filter material, as the required support, assigned in the previous loadcase (structurally fixed with free flow), will carry forward.
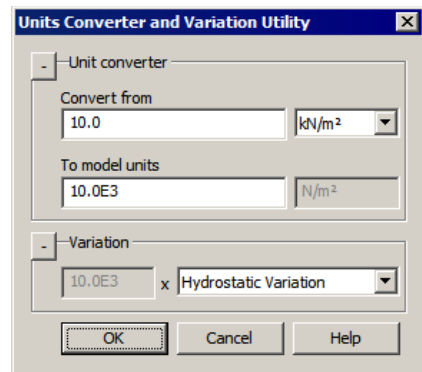
Structural Loading

Gravity

The effects of the self-weight of the dam will be modelled throughout this analysis and can be applied as a property of the analysis.
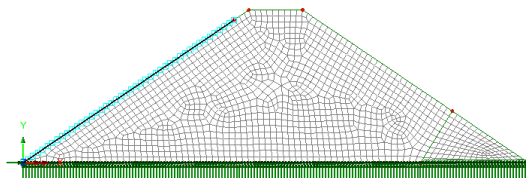
- In the 🕒 Treeview right-click on **Analysis 1** and select **Gravity** (The loadcase icons will change to show that gravity has been assigned)

Pore pressure

For this analysis it will be assumed that the reservoir retained by the dam does not drain significantly due to seepage, or emptying, and therefore the total pressure head can remain fixed at 12m.

Attributes
   Loading…

- On the structural loading dialog select **Prescribed Displacement**, click **Next** and for **Pore pressure** select the **Fixed** radio button, and click the 🔽 button in the **Displacement** field.

- In the Unit Converter and Variation Utility dialog, select **New...** from the Variation drop-down menu to open the Variation dialog.

- Select **General Field Variation**, click **Next** and enter **12-y** in the function field.

- Click the **Function limits** button, then tick **Max. x coordinate** and enter a value of **17.8065**

- Click **OK**, ensure **Global Coordinates** is selected, enter **Hydrostatic Variation** as the name and click **Finish** to exit the dialog.

- To convert the variation, defined in m-head, to hydrostatic pressure, select **kN/m^2** from the Unit converter options. Then enter a value of **10** in the Convert from box. and click **OK**

- Back on the Prescribed displacement dialog, name the attribute **Hydrostatic Pressure** and click **Finish**

Select the lower inclined line of the upstream slope of the dam (Line 1 in the model) and from the 🕸 Treeview and drag and drop the loading attribute **Hydrostatic Pressure** onto the selected line, ensure the loadcase **Seepage** is selected, then click **OK**

Nonlinear Control

The use of two-phase consolidation materials and the requirement to reset the initial displacements require nonlinear settings to be used.

- In the 🕒 Treeview right-click on the loadcase **Initial Condition** and select the **Nonlinear & Transient** option from the **Controls** menu.

- In the Nonlinear & Transient dialog select the **Nonlinear** option in the top-left hand corner and leave the incrementation type as **Manual**

- Select the **Displacement reset step** option. This is effectively setting a Geostatic step.

- In the solution strategy panel change the Incremental displacement norm to **0**. This leads to convergence being assessed based on the (total) Displacement norm. Click **OK** to exit the dialog.

A 🐞 Nonlinear and Transient entry will be added to the Treeview. Double-clicking on this entry will allow any changes to be made to the control properties.

Note: Most geotechnical problems begin from a geostatic state where the undisturbed soil or rock body remains in equilibrium with the prescribed boundary conditions and geostatic loads, including gravity. The use of the 'Displacement reset step' produces zero deformations, but establishes the initial stress field that can be used in a subsequent static or coupled field diffusion/stress analysis.

- In the 🕓 Treeview right-click on **Seepage** and select the **Nonlinear & Transient** option from the **Controls** menu.

- In the Nonlinear & Transient dialog select the **Nonlinear** option and leave the incrementation type as **Manual**

- Select the **Time domain** option and select a **Consolidation** time domain. Enter the initial time step as **0.01**, the total response time as **30** and the maximum number of time steps as **30**. Note that analysis time is measured in days, which was the timescale option set when creating the model.

- Tick the **Automatic time stepping** option, press the **Advanced** button, and change the Time step increment restriction factor to **5.0** and the Maximum time step to **5.0.** As consolidation/seepage effects decay exponentially this will allow the time interval to grow accordingly. Press **OK** to exit the dialog.

- Ensure the Solution strategy is set to **Same as previous loadcase**

- Having set all the Nonlinear & Transient options select **OK** to return to the Modeller graphics window.

Running the Analysis

Open the Solve Now dialog, ensure **Analysis 1** is selected and press **OK** to solve.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the 🕓 Treeview.

In addition, solution files will be created in the   **Associated Model Data\Earth dam** folder.

- ❑ **Earth_dam.dat** this data file contains details of model data and assigned attributes.

❑ **Earth_dam.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

❑ **Earth_dam.mys** this is the LUSAS results file from the analysis which is loaded automatically into the 🕒 Treeview to allow results to be viewed.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered is written to the output files in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding the Model

If errors have been made in the modelling of this example that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully.

❑ **earth_dam_modelling.vbs** carries out the modelling of the example.

File
  New…

Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

• Enter the file name as **earth_dam**

File
  Script   >
    Run Script...

• To recreate the model, select the file **earth_dam_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Run the analysis to generate the results.

Viewing the Results

Analysis loadcase results for each time step are present in the 🕒 Treeview.
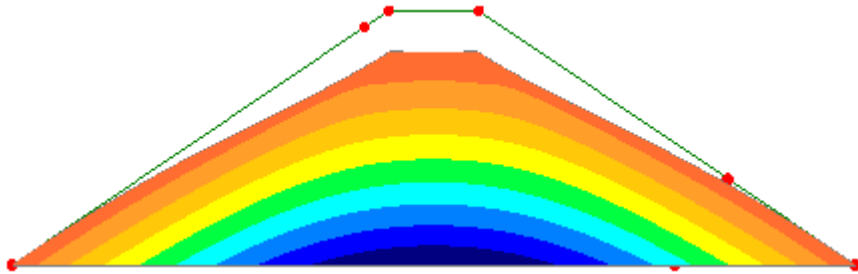
Viewing the Initial Stress Distribution

Two steps take place in the **Initial Condition** loadcase; a first increment and then the Displacement reset (Geostatic) step.

• In the 🕒 Treeview right-click on the Initial Condition loadcase results for **Increment 1** and **Set Active**.

• In the 🔲 Treeview turn-off the **Mesh** and **Attributes** layers.

- In the panel at the bottom of the ⬓ Treeview press the **Deformations...** button, select **Specify factor** enter a factor of **2.0e3** and press **OK**.

- With no features selected right click in a blank part of the Graphics window and select the **Contours** option to add the **Contours** layer to the ⬓ Treeview.

- On the dialog set the results entity to **Stress - Two phase plane strain** and component **SY**, then press **OK**.

The results from Increment 1 show how the structure would behave if gravity was applied to the structure immediately after a wished-in place construction. This is used to get the 'in-situ' stresses then used for the Geostatic step.

- In the 🕓 Treeview right-click on the Initial Condition loadcase results for **Displacement Reset Step** and **Set Active**.
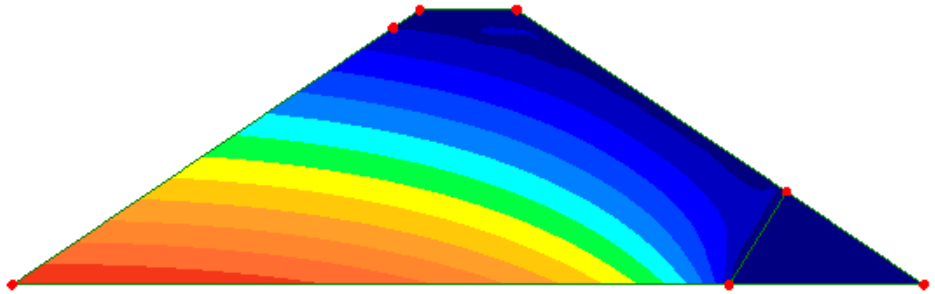

In the Displacement Reset Step loadcase, the deformations are set to zero (to match the as-built geometry) with in-situ stresses maintained
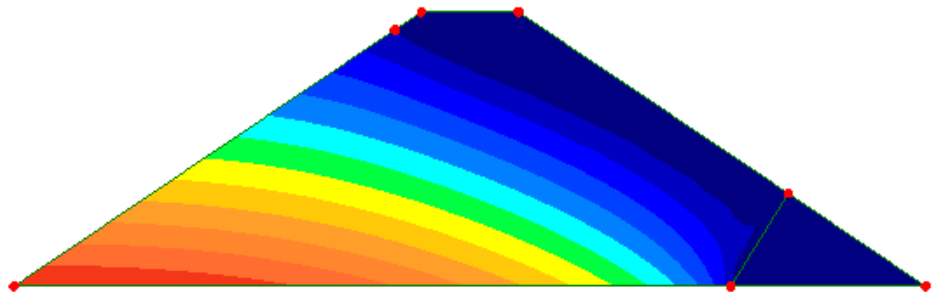
Viewing the Phreatic Line Development

- In the 🕓 Treeview right-click on the Seepage loadcase results for **Time Step 1**, and **Set Active**.

- In the ⬓ Treeview switch off the **Deformed Mesh** layer.

- Double-click the **Contours** layer in the ⬚ Treeview. On the dialog, set the results entity to **Displacement** and component to **PRES**.

- Select the **Contour Display** tab and deselect **Deform**.

- Select the **Contour Range** tab and set an **Automatic** contour range using an Interval of **10e3**. Enter a Maximum value of **120e3** (12m head) a Minimum value of **0**. Click **OK.**

- By setting each time-step active in turn (right-click the time-step in the ⬚ Treeview and **Set Active**) the gradual development of a semi-steady state seepage flow can be seen. During the first three time-steps (Time = 0.31 days =~7.5 hours) the effects of primary consolidation can be observed. From the fourth time-step onwards, however, there is very little change between results.

- In the ⬚ Treeview right-click on the Seepage result-set for **Time Step 10**, and **Set Active**.

From the basic parabola, the top flow (zero pressure) line curves correctly upward on the upstream side, and downward reaching the filter toe.

Dam Discharge

To calculate the seepage flowrate into the filter (i.e. the rate of water draining through it the body of the dam), the flow reaction of the filter bed is going to be used (as water is being drained only from this location).

- Select the bottom line of the toe-drain filter and press [icon] to define a new Group. Enter the name as **Filter** and press **OK** to create the new group.

- Ensure that the last time-step increment of the Seepage results (**Time Step 10 Time = 30.0000**) is set active in the [icon] Treeview:

- The chosen loadcase should be set as (**Time Step 10 Time = 30.0000**)

- Select the entity **Reaction**.

- For the extent select the previously defined group named **Filter**.

- Press **Finish** to view the tabulated results summary.

	Feature ▲	Node	FX[N]	FY[N]	VFlo[N]	RSLT[N]
1	Feature	Node	FX[N]	FY[N]	VFlo[N]	RSLT[N]
2	Line 6	135	-6.02573E3	17.1624E3	N/A	18.1895E3
3	Line 6	134	-12.7967E3	36.2529E3	N/A	38.4452E3
4	Line 6	133	-6.20132E3	18.9331E3	N/A	19.9228E3
5	Line 6	132	-12.2528E3	38.7486E3	N/A	40.6397E3
6	Line 6	131	-6.17924E3	19.9308E3	N/A	20.8668E3
7	Line 6	130	-12.0894E3	40.7204E3	N/A	42.4771E3
8	Line 6	129	-6.04933E3	20.8163E3	N/A	21.6775E3
9	Line 6	128	-11.951E3	42.4641E3	N/A	44.1138E3
10	Line 6	127	-5.95316E3	21.6493E3	N/A	22.4528E3
11	Line 6	126	-11.8034E3	44.0876E3	N/A	45.6403E3
12	Line 6	125	-5.86751E3	22.4376E3	N/A	23.1921E3
13	Line 6	124	-11.6381E3	45.6256E3	N/A	47.0866E3
14	Line 6	123	-5.77922E3	23.1877E3	N/A	23.897E3
15	Line 6	122	-11.4518E3	47.0907E3	N/A	48.4632E3

Model info \ 2:Seepage - 12:Time Step 10 Time = 30.0000 \ 2:Seepage - 12:Time Step 10 Time = 30.0000(Summary)

The value of **VFlo** is the volume of flow during the selected time-step.

Calculations

Considering the semi-steady flow achieved by 30 days, the average flow-rate into the filter drain can be calculated from the total discharge shown in these results over the duration of the time-step between the last and second-to-last increments.

- Save the results in a spreadsheet by right-clicking on the Print Results Wizard output and selecting **Save as Microsoft Excel**.

- In the saved spreadsheet, sum the **VFlo** column to get the total discharge over the time-step ($193m^3$).

- Identify the time of the last and second-to-last timesteps from the ⬕ Treeview (4.2 days)

- Divide the total discharge over the final interval to calculate the average flow rate into the filter dam (45.9m^3/day/m)

This completes the example.