Funded by

Bundesministerium für Bildung und Forschung

TECHNISCHE UNIVERSITÄT DRESDEN

Numerical flow and transport modeling using the INOWAS platform

Tutorial 1: Set up of steady state groundwater flow model

INOWAS

MAR Junior Research Group

The INOWAS Platform

The INOWAS platform is a free web-based platform to provide a collection of simple, practical and reliable tools to solve groundwater related issues.

This tutorial uses the Tool 3 - “Numerical groundwater modelling and optimization” to create a numerical groundwater flow model on the platform.

The benefits of creating numerical models on the platform:

- ✓ **1. User-friendly** (easy to access, free)
- ✓ **2. Shareable-models** (models can be shared via the platform)
- ✓ **3. Intuitive results visualization**

Introduction

This tutorial provides an overview of the “Numerical groundwater modelling and optimization” tool on the INOWAS platform and guides users to create a simple **steady-state groundwater flow model** on the platform.

More information about the tool can be found on the respective documentation page:

https://inowas.com/tools/t03-modflow-model-setup-and-editor/

The tutorial takes about 30 min for completion.

Before you start, please register your user account here: https://inowas.com/

Example background

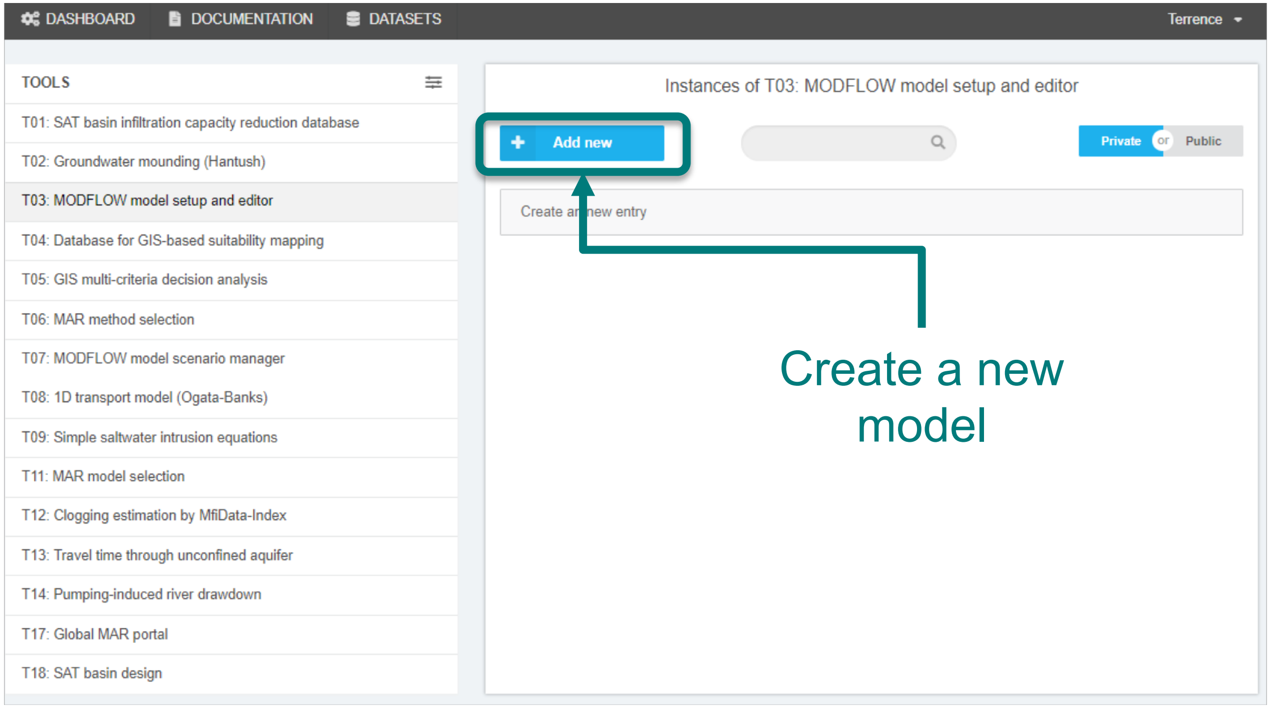
The groundwater flow model which will be build during this tutorial is a **confined steady state groundwater flow model.**

The model only has one steady-state stress period. For discretization, the model has 1 layer with 50 m depth and 10 x 10 columns & rows with 100 m width & height.

There are two constant boundaries on the left and right side of the model domain. In addition, the aquifer is assumed to be homogenous and isotropic.

Create a new model on the platform

Step 1. Log in to the web-based INOWAS platform
Step 2. Navigate to the dashboard and select "T03. Numerical groundwater modelling and optimization"
Step 3. Create a new model by clicking on the "Add new" button



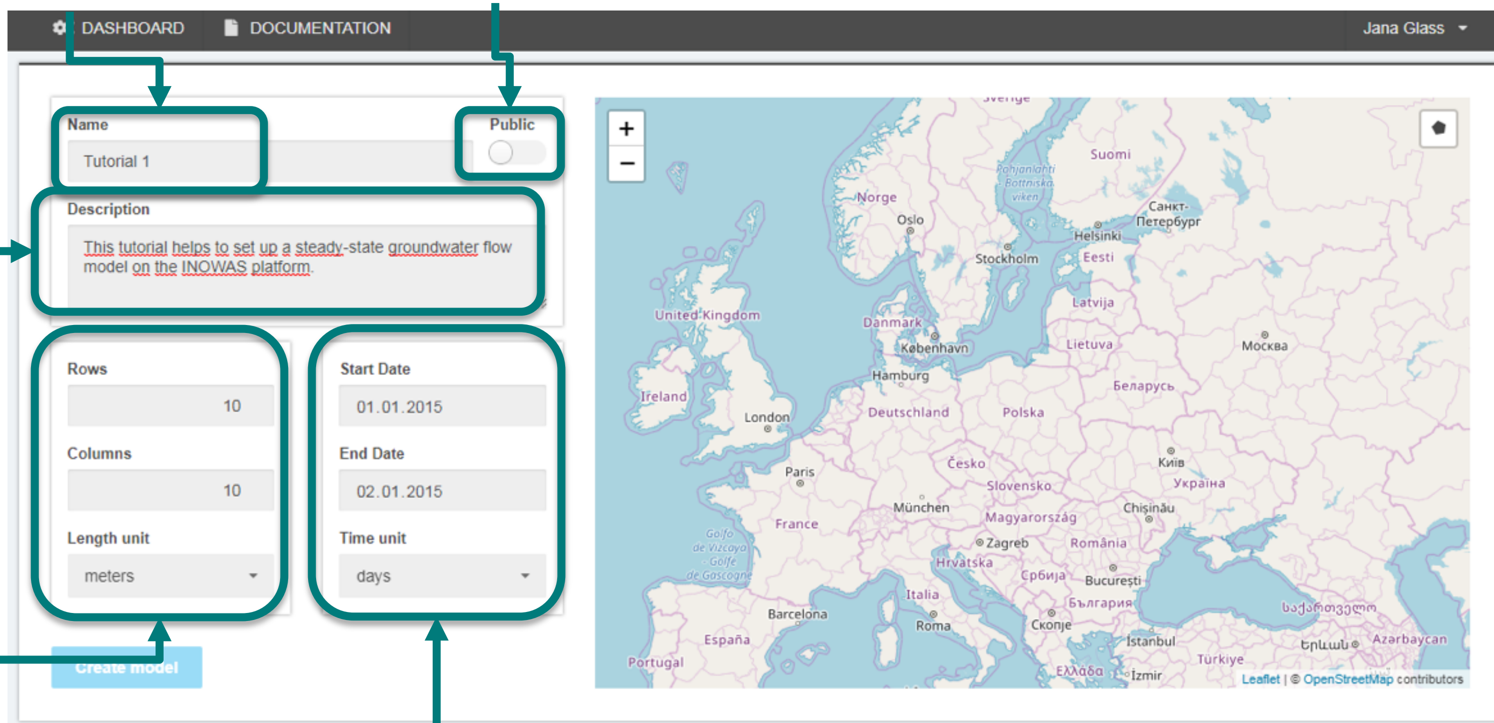
Set general model properties

Step 4. Fill the general information of the new model

Define the model name

Set the model as Public or Private

Add description of the new model



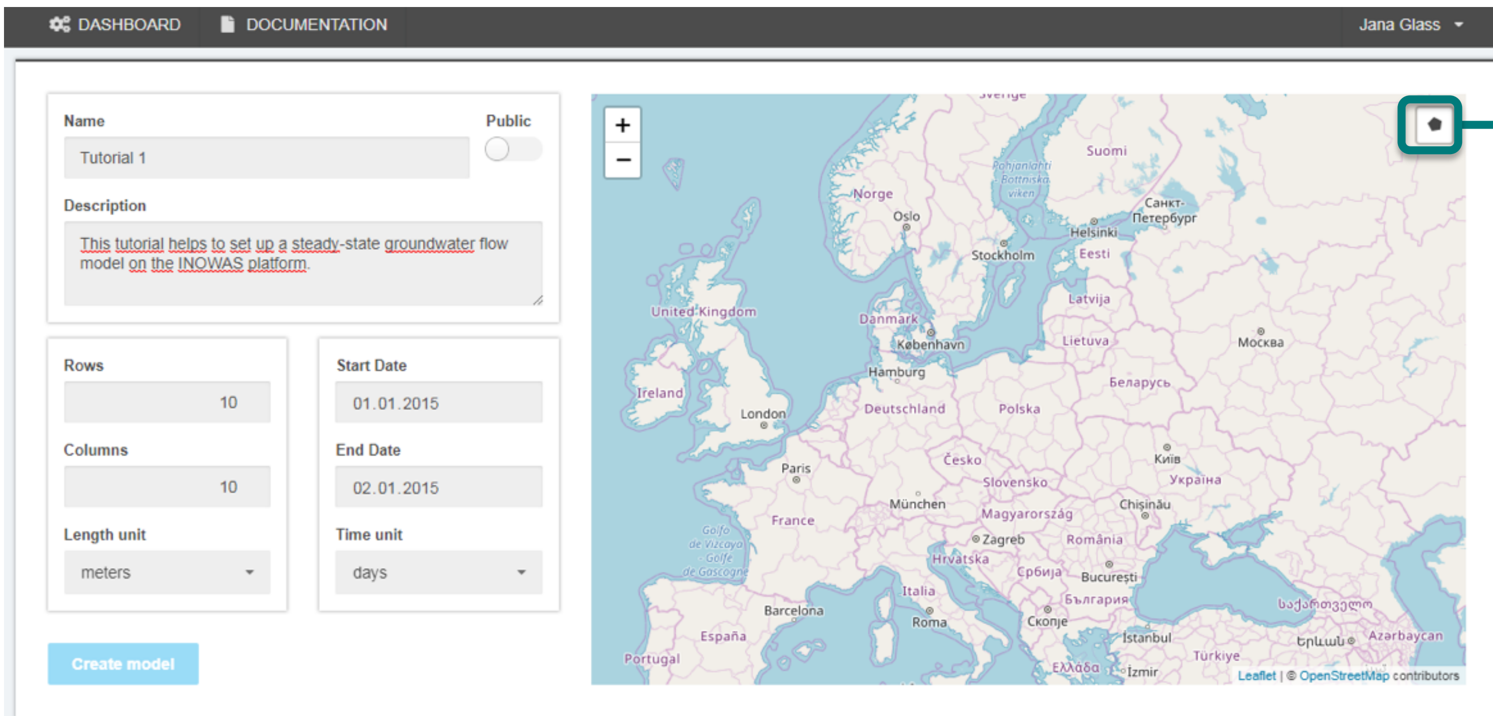
Define the number of rows and columns, and length unit

Set the start date, end date, and time unit

Set general model properties (2)

Step 5. Define the spatial discretization and the model area. The cell size is determined by the bounding box. The bounding box can be set by creating a polygone.

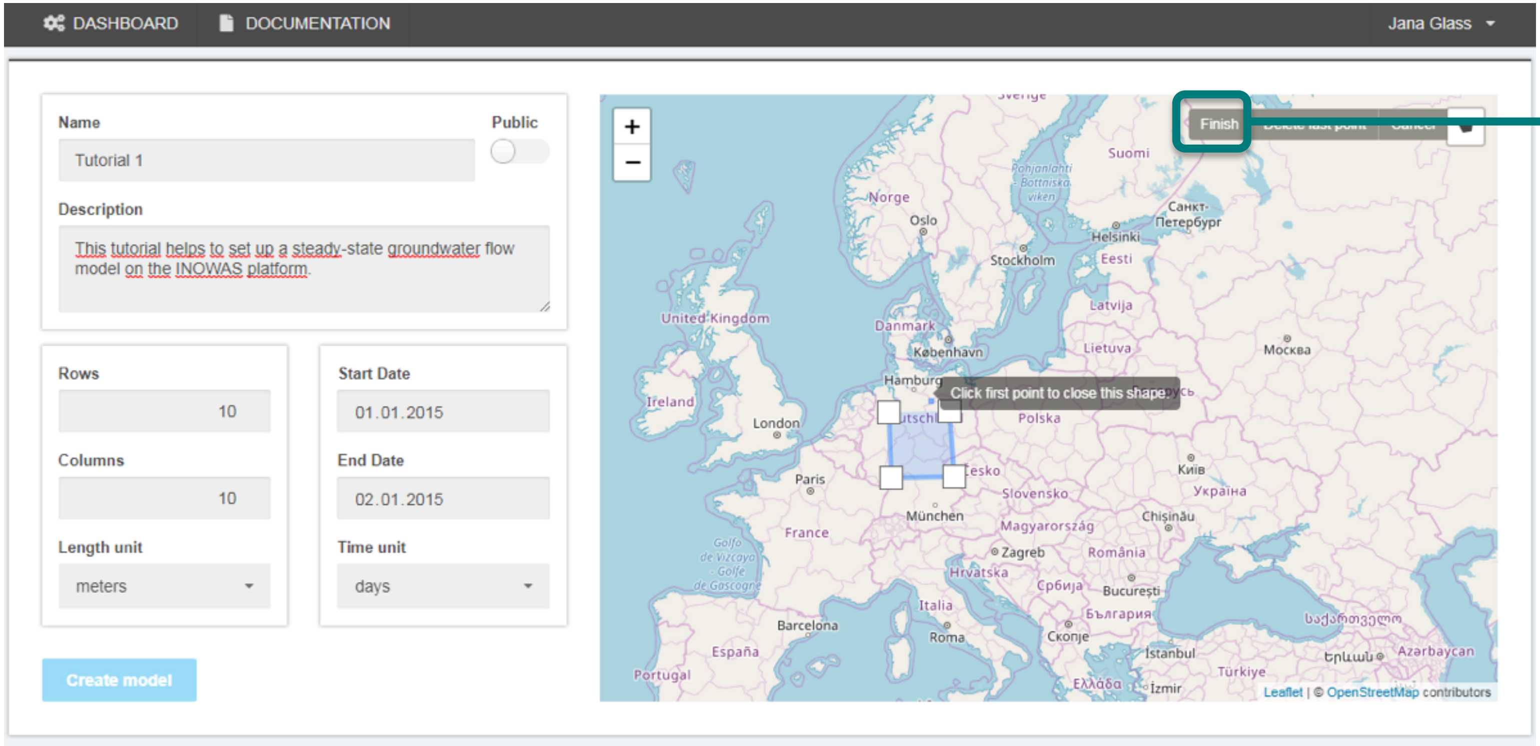
Alternatively, the model area can be uploaded (shown later).



Click this button to start create the bounding box

Set general model properties (3)

Step 5. Define the spatial discretization and the model area. Draw the model area by click on various points on the map.

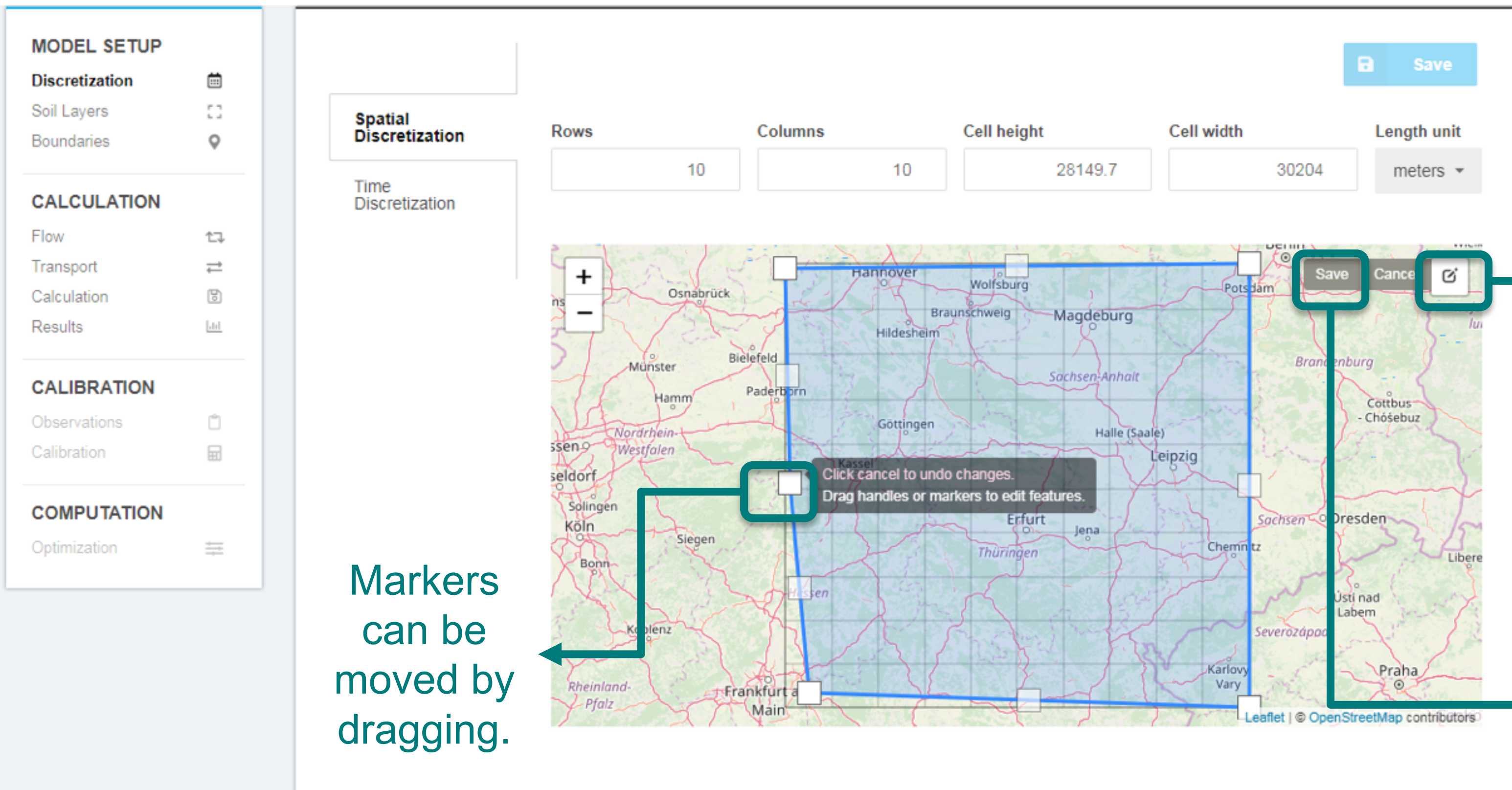


Finish editing bounding box

After defining the bounding box, the model can now be created by clicking the "Create model" button.

Set general model properties (4)

After creating the model, the bounding box can be modified.



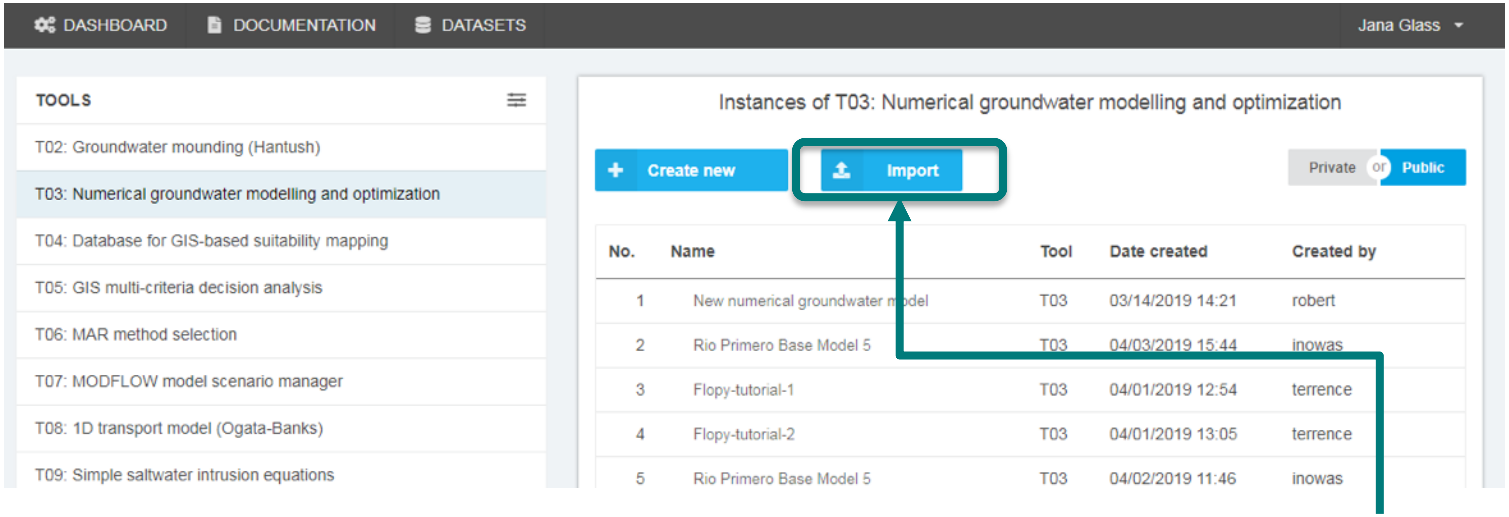
Markers can be moved by dragging.

Start to modify the model area

Push the "Save" button to save changes.

Import models/ model area

The second option to create the bounding box is to upload the model area.
Existing models or parts of a model (such as the model area in our case) can be uploaded by using the import function.



Import existing model files

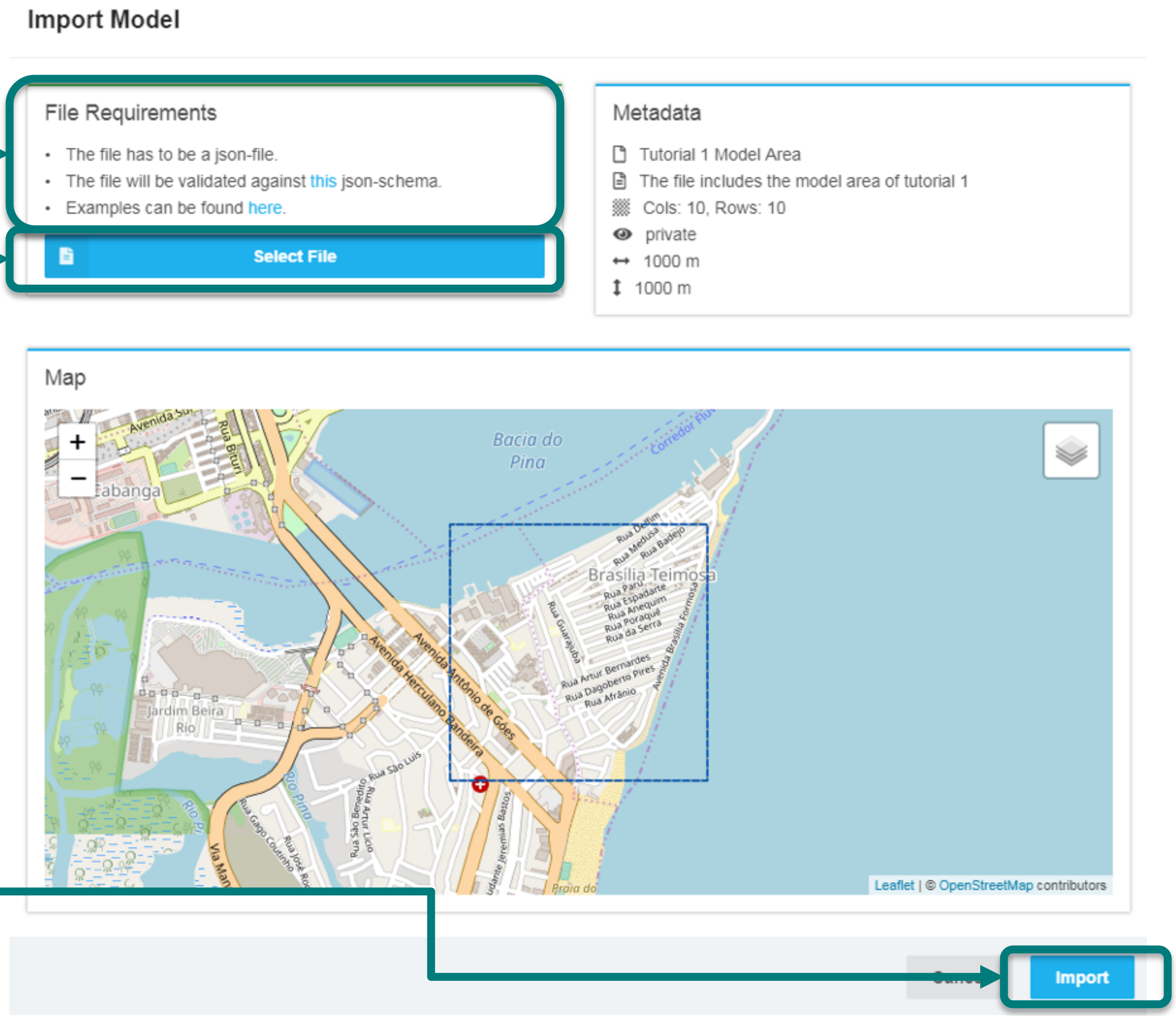
Please go back to the Dashboard and click on the “Import” button.

Import models/ model area

The file has to be in json-format and needs to have a specific json-schema.

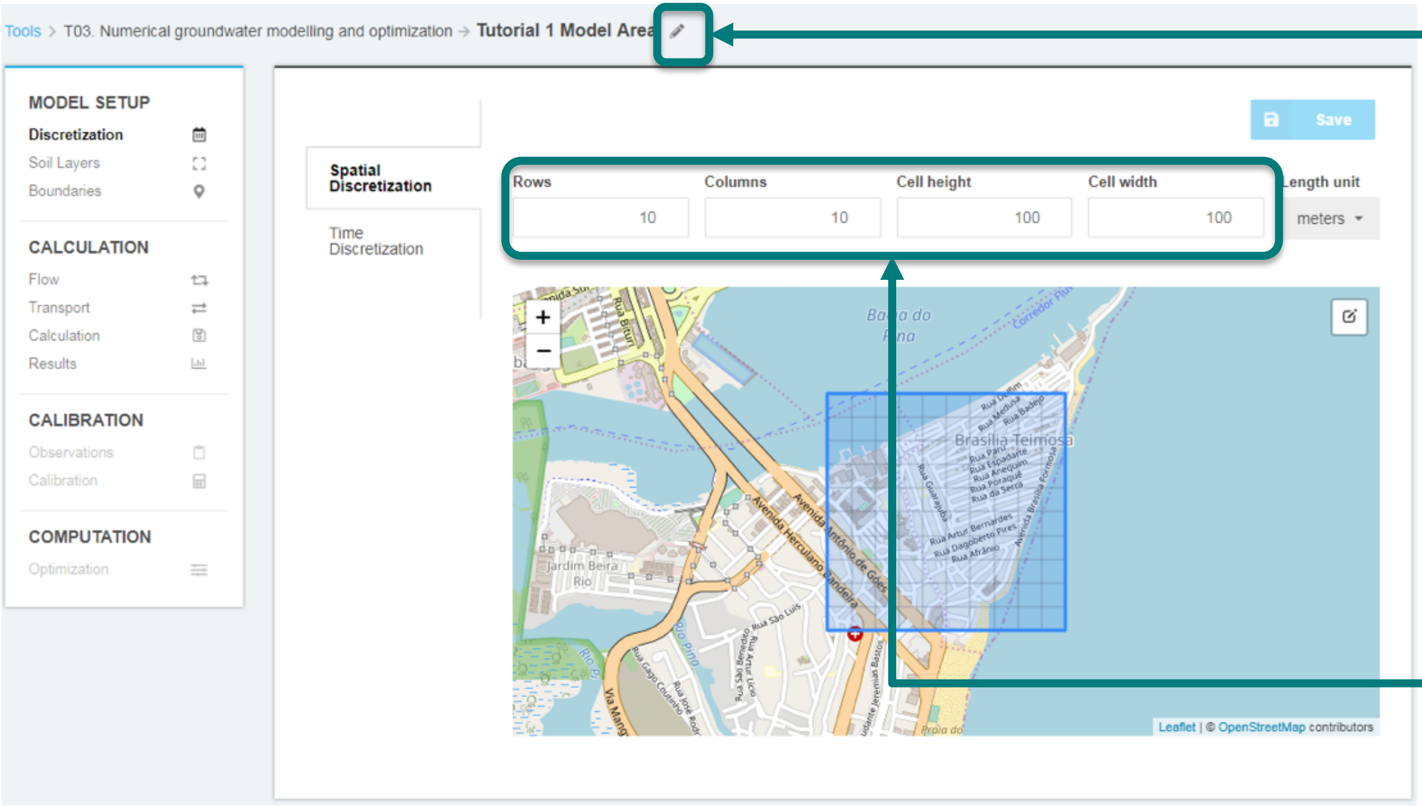
Now select the file “Tutorial 1_model_area.json” which was included in the tutorial material.
(or download it from inowas.com)

Click on “Import” to finish the model or model area upload.



General model name and spatial discretization

After creating the model, the spatial discretization is visualised. General model properties such as the name, description and public availability can be modified.



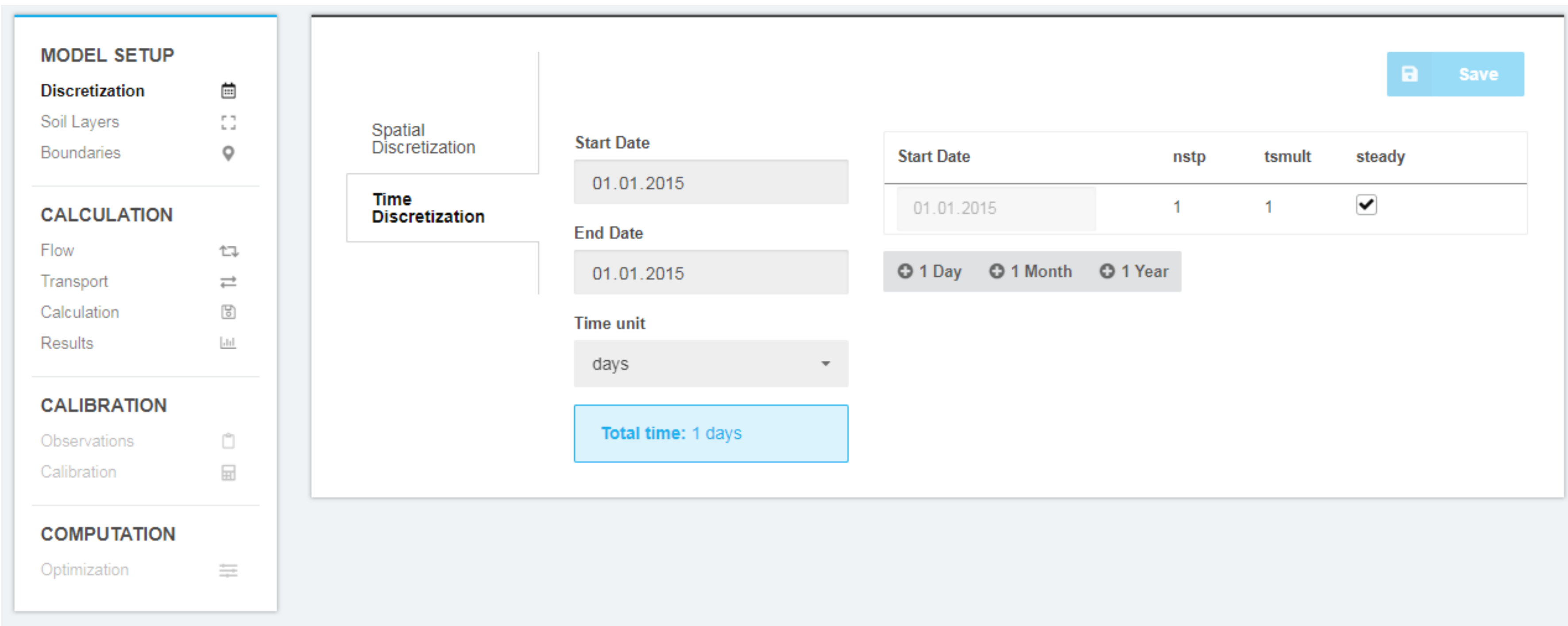
edit the model name, description and private/public status of the model

Visualize and edit the spatial discretization

Time discretization

In addition to the spatial discretization, also the time discretization can be visualised and edited.

Please make sure, you have set one steady state period with the length of one day.

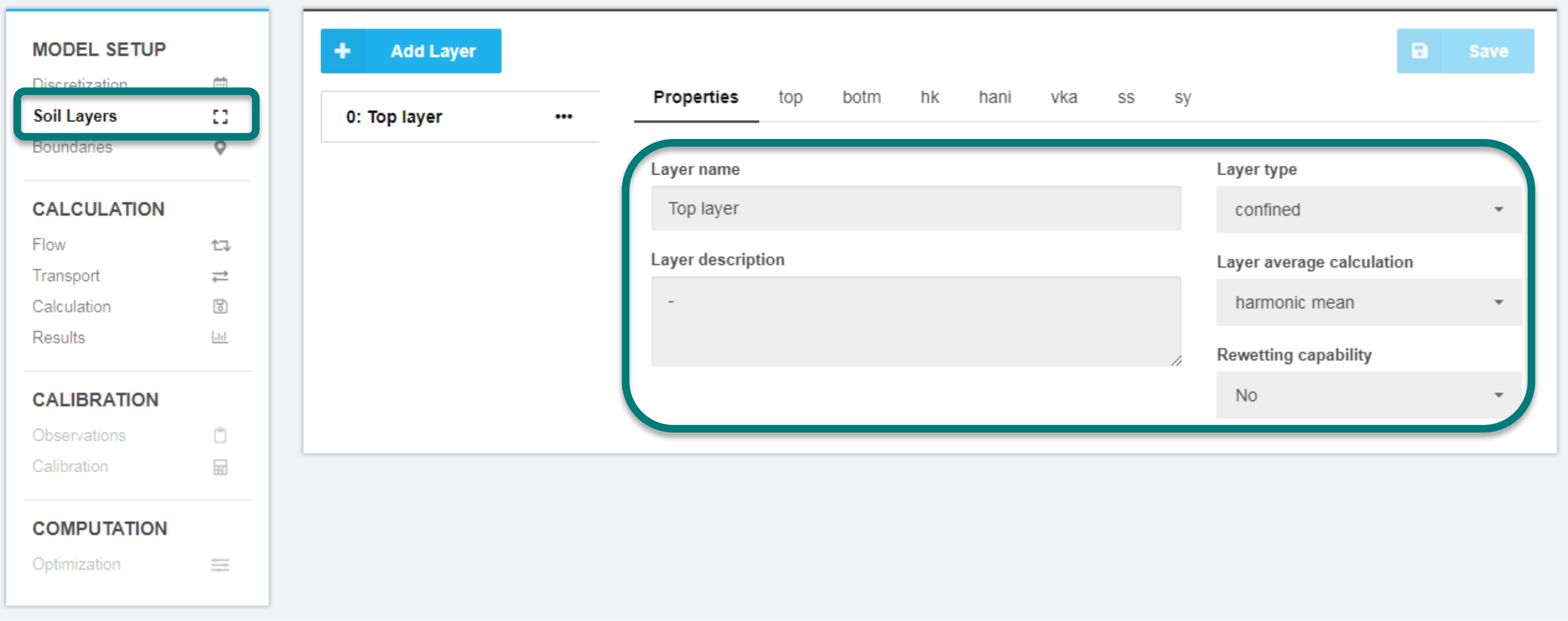


INOWAS

Soil layers

Now, the soil layer needs to get defined. By default, one soil layer exists.

1. Click ‘Soil Layers’ section to edit the default layer



2. Add the name of the layer, set the layer type as confined. Keep the rest of the features as the default setting.

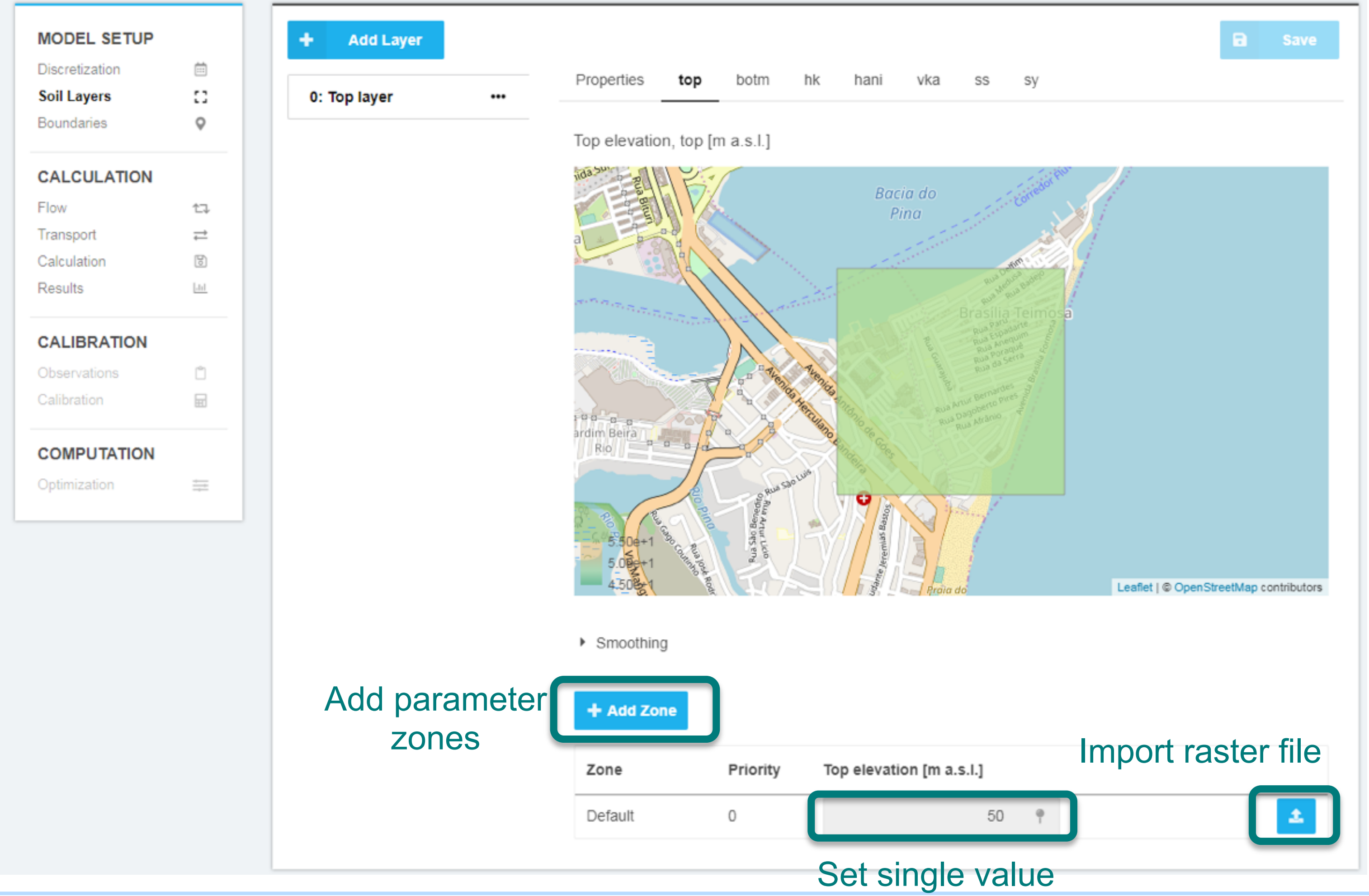
Soil layer properties

There are currently three possibilities available to define layer properties such as layer extents, hydraulic conductivity and storage parameters.

1) Set a single value,
2) Define zones or
3) Import a raster file containing the respective values

Soil layer properties (2)

The screenshot visualizes the three options to set soil parameters:



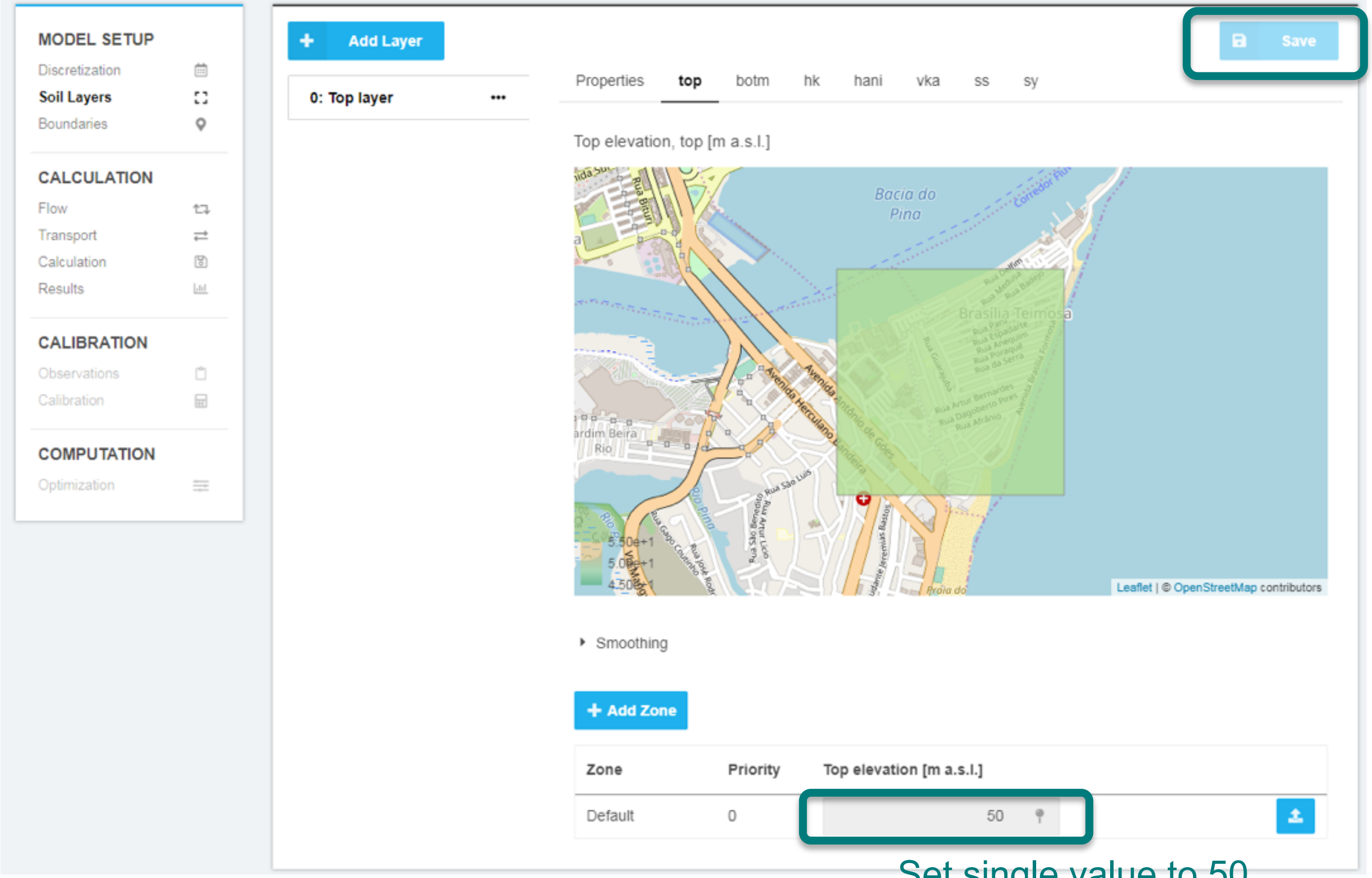
INOWAS

Soil layer properties (3)

In this tutorial, we will define layer parameters by setting a single value for the whole model area.

Don´t forget to save the changes!

Set the top layer elevation to 50 m as shown on the screenshot.



Set single value to 50

INOWAS

Soil layer properties (4)

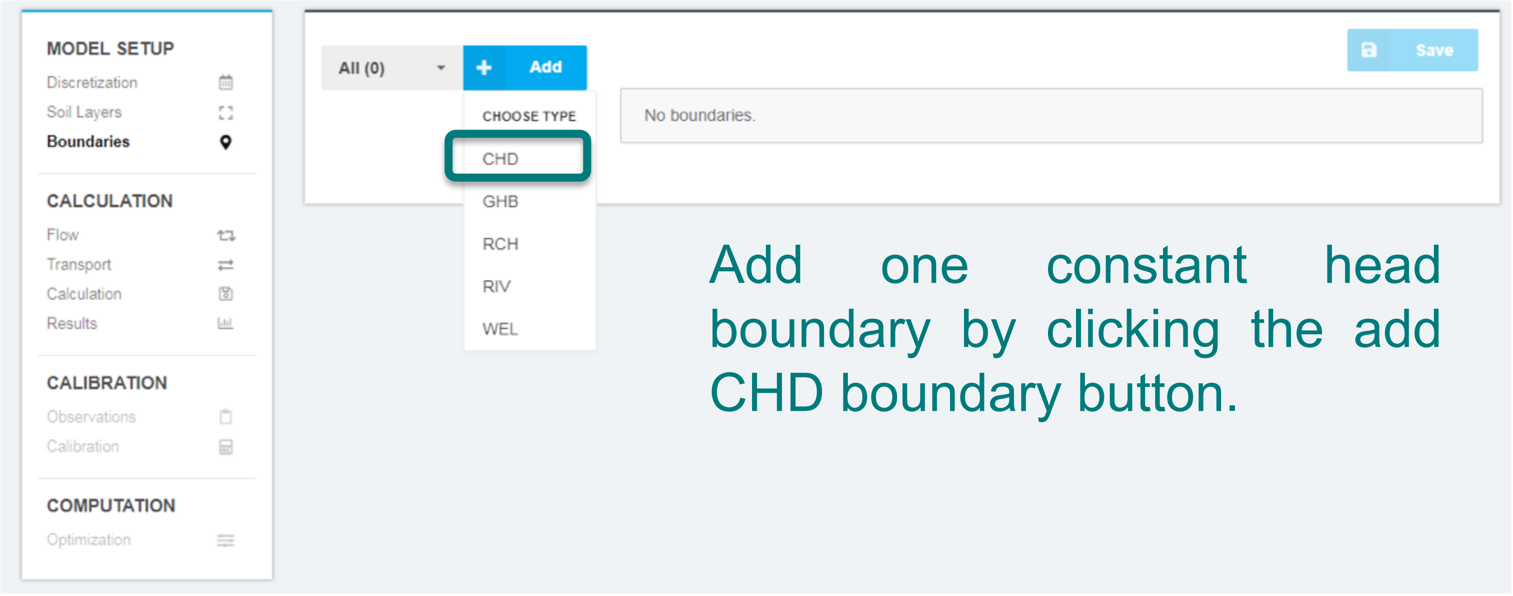
Now also define the following soil parameters as previously described for the top layer elevation.

• Bottom of the layer	**Botm: 0**
• Horizontal hydraulic conductivity	**Hk: 10**
• Horizontal anisotropy	**Hani: 1**
• Vertical hydraulic conductivity	**Vka: 10**
• Specific storage	**Ss: 0.00001**
• Specific yield	**Sy: 0.15**

Boundary conditions

Boundary conditions can be edited in the section “Boundaries”. At the moment the following MODFLOW boundary conditions can be added: Constant head boundary condition (CHD), general head (GHB), recharge (RCH), river (RIV) and well (WEL).

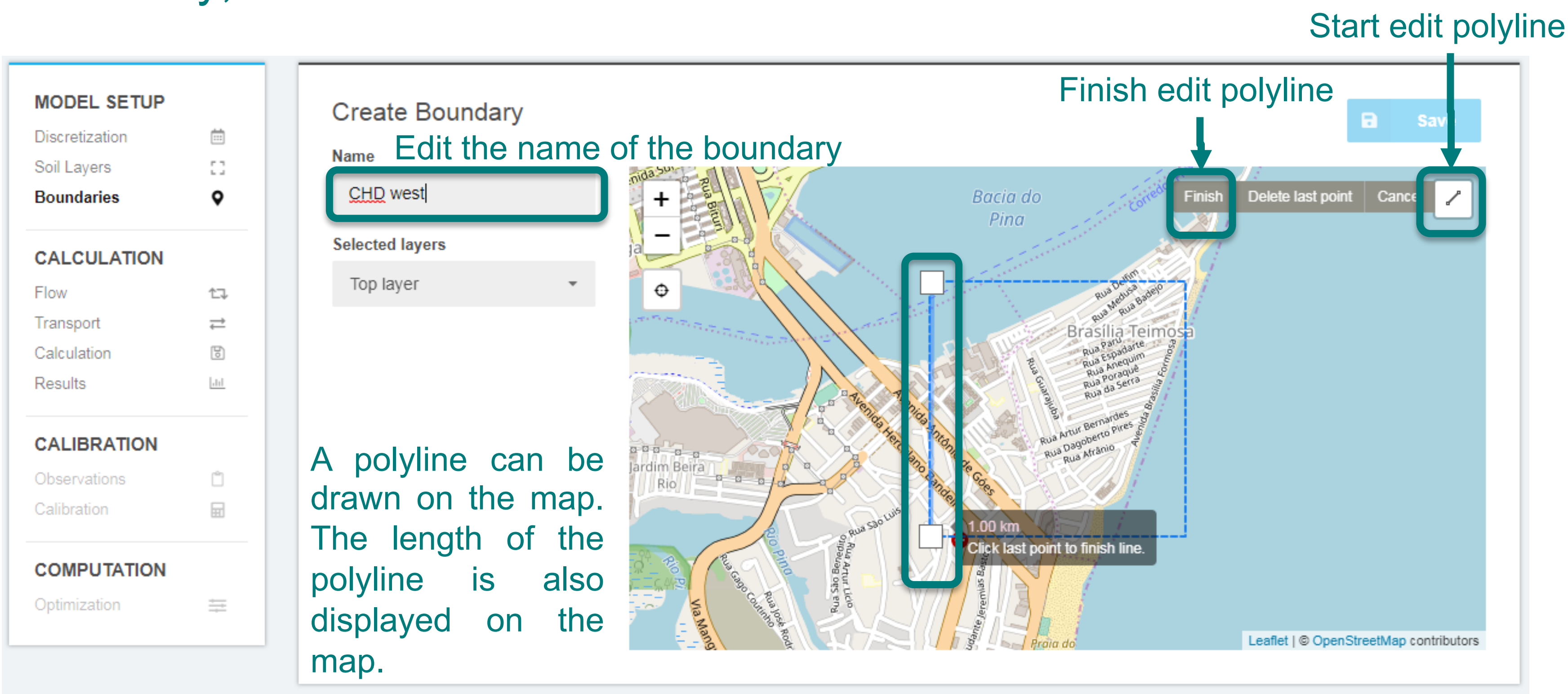
For the tutorial, we will add in total 2 CHD boundaries on the eastern and western side of the model domain.



Boundary conditions (2)

The CHD boundary can be added by drawing a polyline on the map.

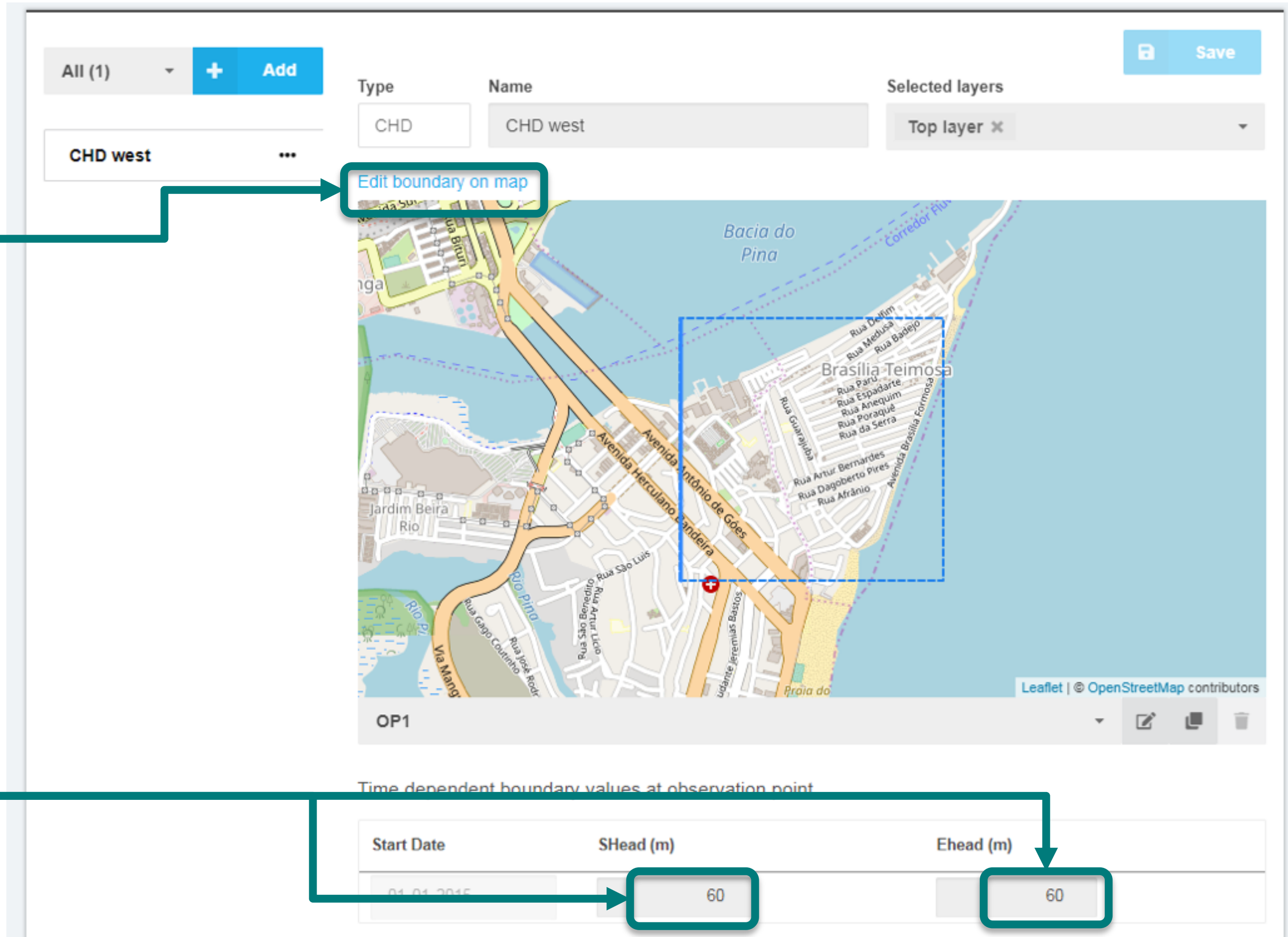
Add a polyline. After drawing the polyline along the western model boundary, click on Finish to return to the Model Editor.



Don´t forget to save the changes!

Boundary conditions (3)

Set the starting and ending head of the western boundary to 60 m.

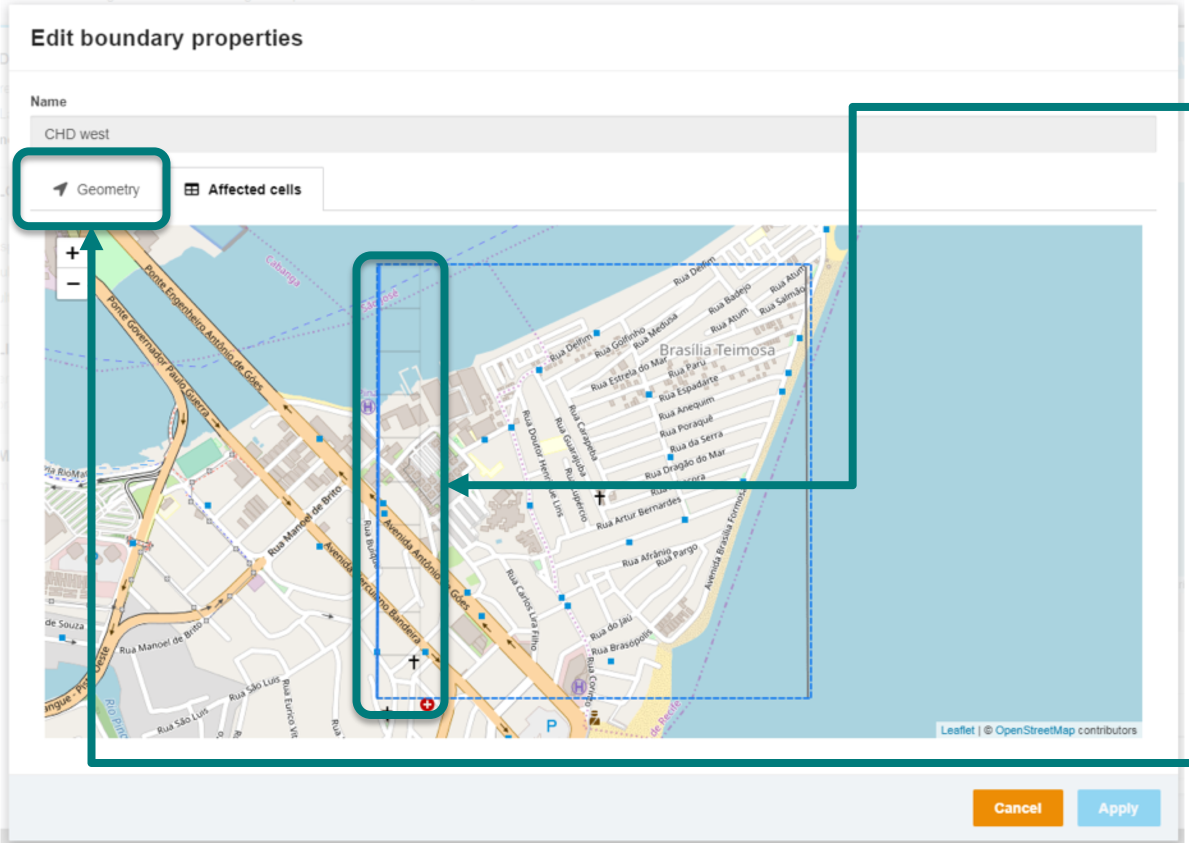


The location of the boundary can be edited here after it has been created

Starting head and ending head of the CHD boundary. Here, the values are the same: 60 m

Boundary conditions (4)

Click on the “Edit boundary on map” to visualize the location of the polyline and the affected cells.



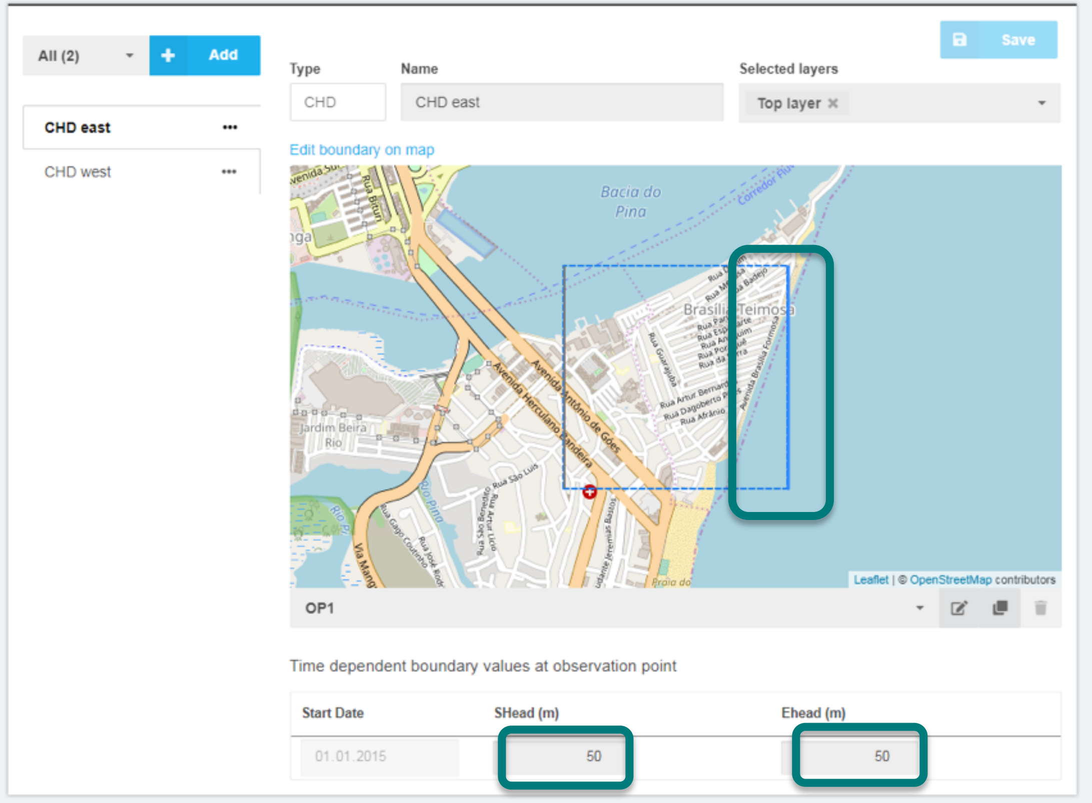
Make sure that at the western side of the boundary, all 10 cells are visible.

If this is not the case, you need to modify the location of the boundary in the “Geometry” section as previously described for the bounding box (p.9).

INOWAS

Boundary conditions (5)

Now add a second CHD boundary on the eastern side of your model domain. The starting and ending head is 50 m. Also make sure, that all 10 cells on the eastern model boundary are affected by the drawn polyline.



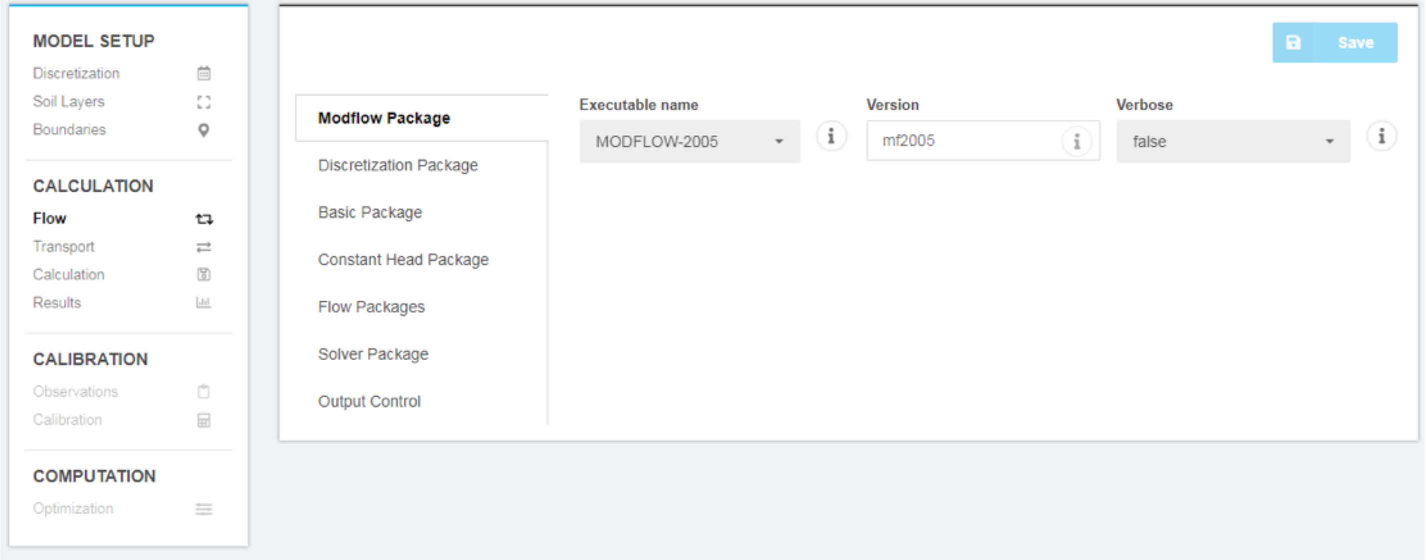
Don´t forget to save the changes!

INOWAS

Calculation: flow parameters

In the Flow part of the Calculation section, MODFLOW-specific parameters such as the executable and parameters of the solver, flow and basic package can be edited. The Basic package includes e.g. the visualisation of the starting head and active cells.

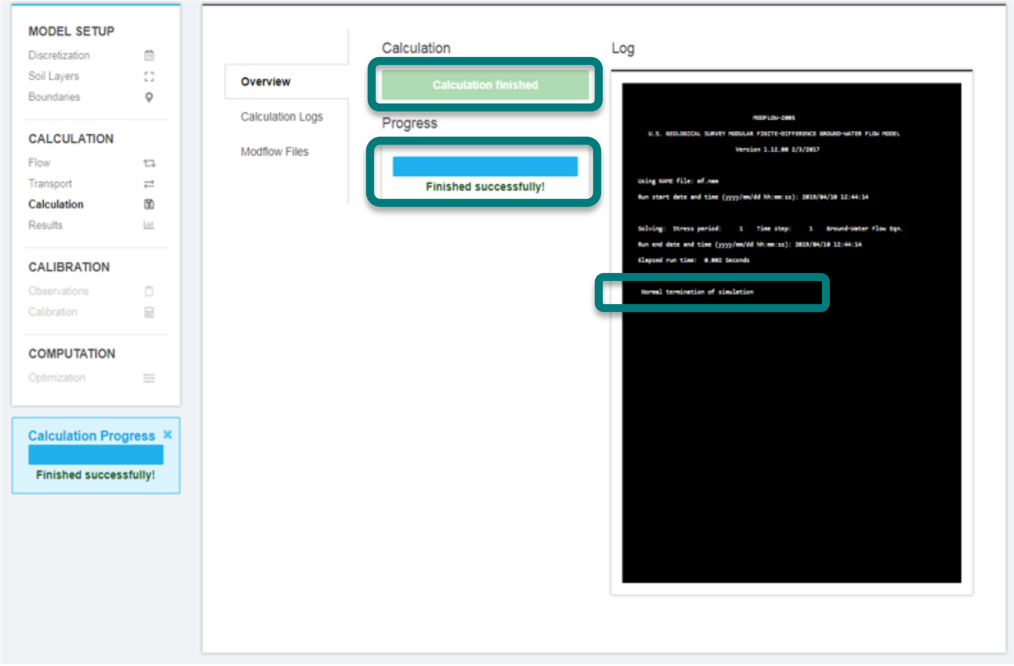
The peconditioned conjugate-gradient Package (**PCG**) as solver and the Layer-Property Flow Package (**LPF**) as flow package are utilized. All default parameters can be kept constant.



INOWAS

Model run

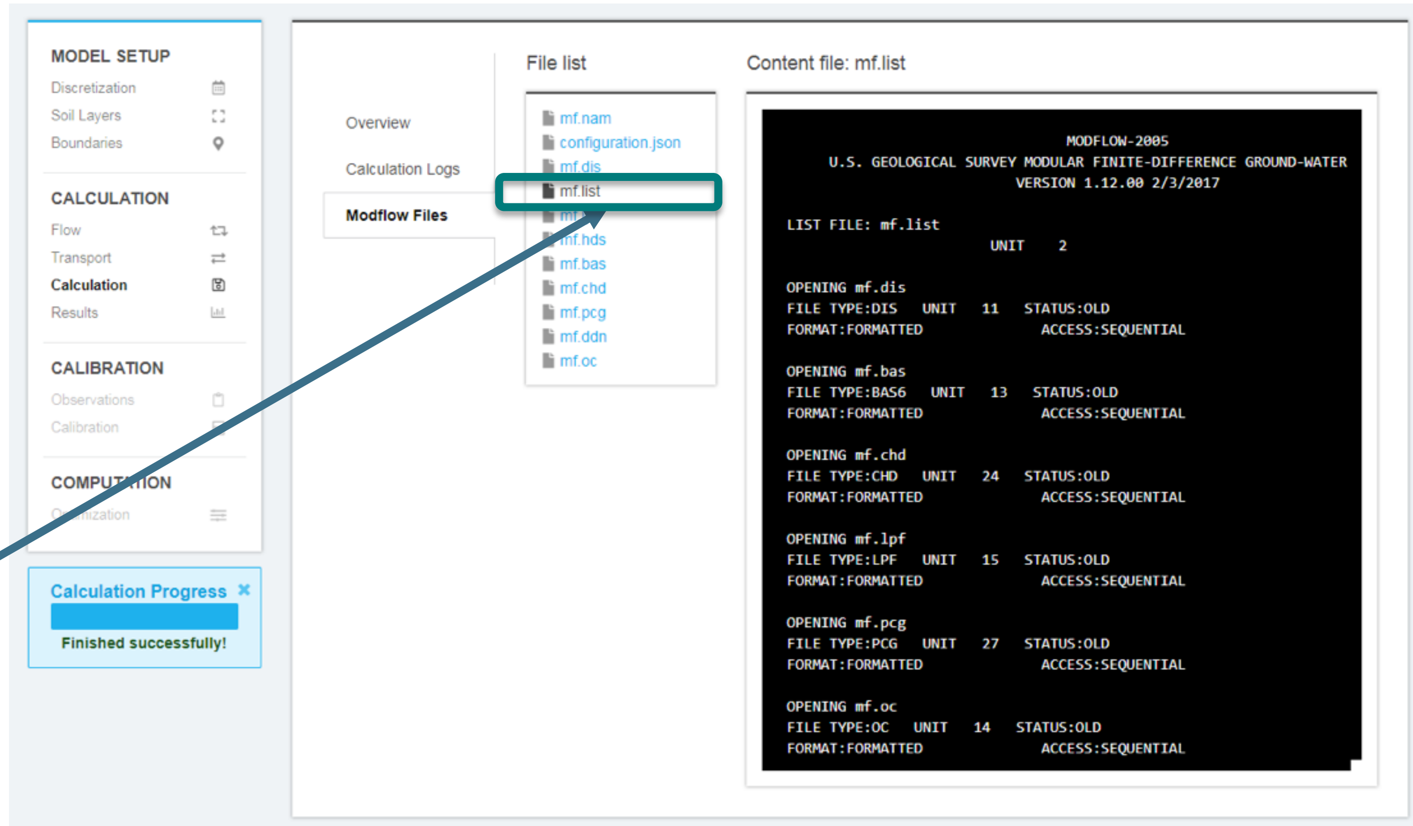
In the Calculation part of the Calculation section, the numerical model can be run. Run the model by clicking on the “Calculate” button.



The progress of the simulation is visible and shows “finished successfully” when MODFLOW terminated. Make sure that in the log on the right side the last sentence is **“Normal termination of simulation”**.

Results – MODFLOW files

Besides the “calculation logs”, other MODFLOW files can be displayed.



Other MODFLOW files (e.g. mf.list) can be checked here

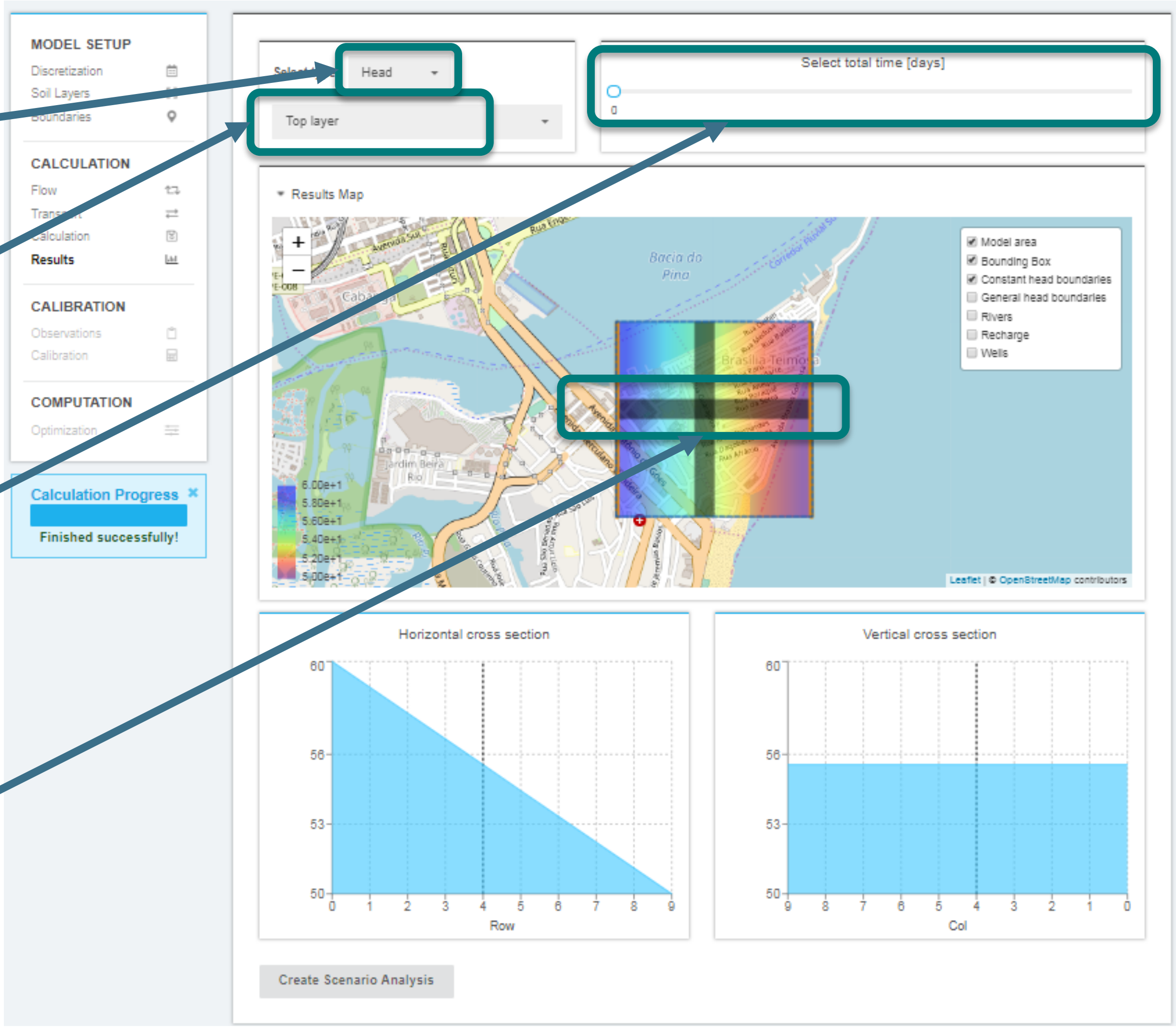
Results –head and drawdown visualisation

Head or drawdown can be displayed.

Choose different layers (In this case, the model has only one layer).

If a transient simulation is run, also different time steps can be visualized.

Click different rows to evaluate the heads at different cross-sections



Contact

Thank you for going through Tutorial 1. If you have any comments or questions, please contact us!

TECHNISCHE UNIVERSITÄT DRESDEN

INOWAS
MAR Junior Research Group

Further Tutorials about the INOWAS platform:

Tutorial 2- transient groundwater flow modeling and scenario analysis

Tutorial 3- set up of solute transport model