

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



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.

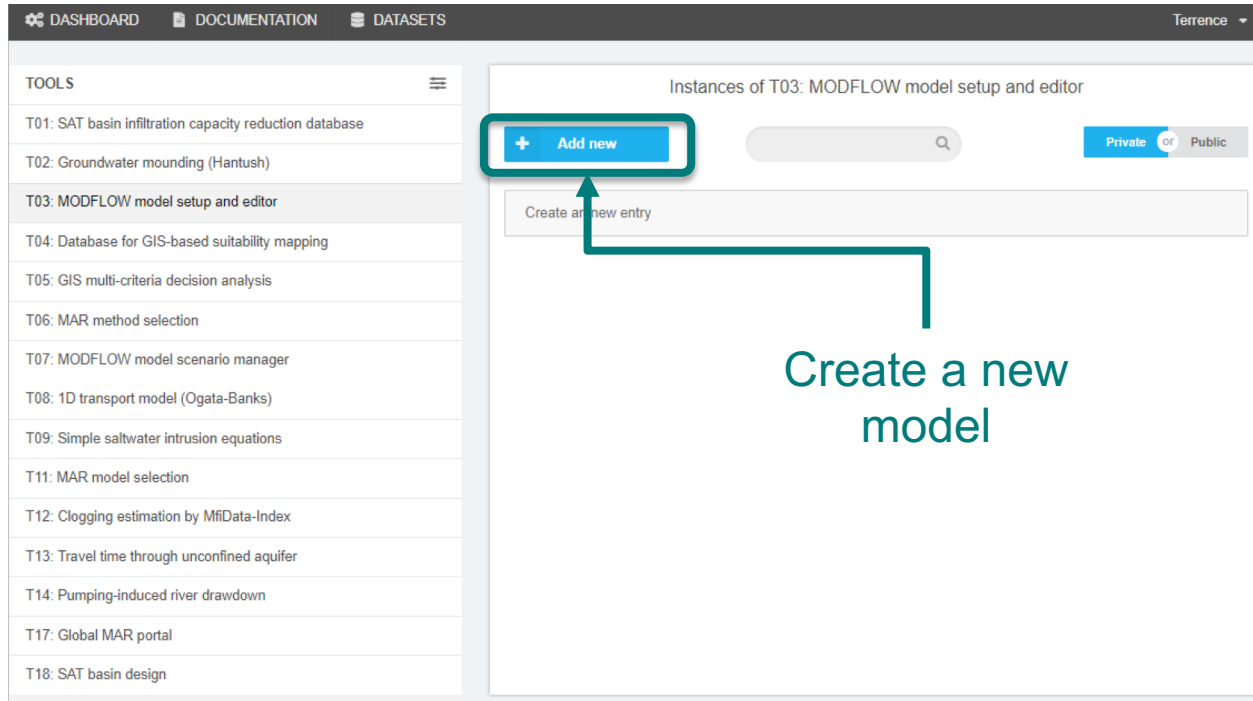
There are two constant boundaries on the left and right side of the model domain. In addition, the aquifer is assumed to be homogenous and isotropic.

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



The screenshot displays the INOWAS platform dashboard. On the left, a sidebar lists various tools (T01 to T18), with 'T03: MODFLOW model setup and editor' selected. The main content area is titled 'Instances of T03: MODFLOW model setup and editor'. A blue button with a plus sign and the text 'Add new' is highlighted with a red box. A red arrow points from this button to the text 'Create a new model' located below the main content area. The dashboard also features a search bar, a 'Private or Public' toggle, and a 'Create a new entry' input field.

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

DASHBOARD DOCUMENTATION Jana Glass

Name: Tutorial 1

Public:

Description: This tutorial helps to set up a steady-state groundwater flow model on the INOWAS platform.

Rows: 10

Columns: 10

Length unit: meters

Start Date: 01.01.2015

End Date: 02.01.2015

Time unit: days

Create model

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).

The screenshot shows the INOWAS dashboard interface. At the top, there are navigation links for 'DASHBOARD' and 'DOCUMENTATION', and a user profile 'Jana Glass'. The main area is divided into two sections. On the left, there is a form for model configuration. The 'Name' field contains 'Tutorial 1' and there is a 'Public' toggle switch. The 'Description' field contains the text: 'This tutorial helps to set up a steady-state groundwater flow model on the INOWAS platform.' Below this, there are input fields for 'Rows' (10) and 'Columns' (10), a 'Length unit' dropdown set to 'meters', 'Start Date' (01.01.2015), 'End Date' (02.01.2015), and 'Time unit' (days). A blue 'Create model' button is at the bottom left. On the right, there is a map of Europe. A small square button with a house icon is highlighted with a red box in the top right corner of the map area.

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.

The screenshot displays the INOWAS platform interface. On the left, there is a form for defining model properties:

- Name:** Tutorial 1
- Public:** A toggle switch is currently turned off.
- Description:** This tutorial helps to set up a steady-state groundwater flow model on the INOWAS platform.
- Rows:** 10
- Columns:** 10
- Length unit:** meters
- Start Date:** 01.01.2015
- End Date:** 02.01.2015
- Time unit:** days

A blue button labeled "Create model" is located at the bottom of the form. On the right, a map of Europe shows a blue bounding box drawn around a region in Central Europe. A tooltip above the box says "Click first point to close this shape." A green box labeled "Finish" is positioned over the top-right corner of the bounding box, with a green arrow pointing to the text "Finish editing bounding box" on the right side of the image.

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.

MODEL SETUP

- Discretization
- Soil Layers
- Boundaries

CALCULATION

- Flow
- Transport
- Calculation
- Results

CALIBRATION

- Observations
- Calibration

COMPUTATION

- Optimization

Spatial Discretization

Time Discretization

Rows	Columns	Cell height	Cell width	Length unit
10	10	28149.7	30204	meters

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.

The screenshot shows a web dashboard with a dark header containing 'DASHBOARD', 'DOCUMENTATION', 'DATASETS', and a user profile 'Jana Glass'. On the left, a 'TOOLS' sidebar lists various tools, with 'T03: Numerical groundwater modelling and optimization' selected. The main content area is titled 'Instances of T03: Numerical groundwater modelling and optimization'. It features a 'Create new' button and an 'Import' button (highlighted with a red box and an arrow). Below these buttons is a table with the following data:

No.	Name	Tool	Date created	Created by
1	New numerical groundwater model	T03	03/14/2019 14:21	robert
2	Rio Primero Base Model 5	T03	04/03/2019 15:44	inowas
3	Flopy-tutorial-1	T03	04/01/2019 12:54	terrence
4	Flopy-tutorial-2	T03	04/01/2019 13:05	terrence
5	Rio Primero Base Model 5	T03	04/02/2019 11:46	inowas

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.

Import Model

File Requirements

- The file has to be a json-file.
- The file will be validated against [this](#) json-schema.
- Examples can be found [here](#).

Select File

Metadata

- 📁 Tutorial 1 Model Area
- 📄 The file includes the model area of tutorial 1
- 📏 Cols: 10, Rows: 10
- 👁 private
- ↔ 1000 m
- ↑ 1000 m

Map

Map showing a coastal area with streets and a highlighted region (Brasilia Teimosa) in a blue dashed box. The map includes labels for streets like Avenida Herculanu Bandeira, Avenida Antonio de Góes, and Rua da Serra. A red location pin is visible on the map.

Leaflet | © OpenStreetMap contributors

Cancel **Import**

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.

Tools > T03. Numerical groundwater modelling and optimization → Tutorial 1 Model Area

MODEL SETUP

- Discretization
- Soil Layers
- Boundaries

CALCULATION

- Flow
- Transport
- Calculation
- Results

CALIBRATION

- Observations
- Calibration

COMPUTATION

- Optimization

Spatial Discretization

Rows	Columns	Cell height	Cell width	Length unit
10	10	100	100	meters

Save

Visualize and edit the spatial discretization

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

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.

The screenshot displays the 'Time Discretization' configuration interface. On the left, a sidebar lists navigation options: MODEL SETUP (Discretization, Soil Layers, Boundaries), CALCULATION (Flow, Transport, Calculation, Results), CALIBRATION (Observations, Calibration), and COMPUTATION (Optimization). The main panel shows the 'Time Discretization' settings. It includes a 'Start Date' field set to '01.01.2015' and an 'End Date' field also set to '01.01.2015'. The 'Time unit' is set to 'days'. A table below these fields shows the discretization parameters:

Start Date	nstp	tsmult	steady
01.01.2015	1	1	<input checked="" type="checkbox"/>

Below the table, there are buttons for '+ 1 Day', '+ 1 Month', and '+ 1 Year'. A blue box at the bottom indicates 'Total time: 1 days'. A 'Save' button is located in the top right corner of the main panel.

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

The screenshot displays the software interface for defining soil layers. On the left, a sidebar titled 'MODEL SETUP' has 'Soil Layers' highlighted with a red box. The main area shows a 'Properties' panel for the '0: Top layer'. The 'Layer name' is 'Top layer', 'Layer type' is 'confined', 'Layer description' is '-', 'Layer average calculation' is 'harmonic mean', and 'Rewetting capability' is 'No'. A red box highlights the 'Layer name', 'Layer type', and 'Layer description' fields.

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:

The screenshot shows a software interface for setting soil layer properties. The interface is divided into several sections:

- MODEL SETUP:** Includes options for Discretization, Soil Layers, and Boundaries.
- CALCULATION:** Includes options for Flow, Transport, Calculation, and Results.
- CALIBRATION:** Includes options for Observations and Calibration.
- COMPUTATION:** Includes an option for Optimization.

The main area displays a map with a green highlighted area. Below the map, there is a table for 'Top elevation, top [m a.s.l.]' with the following data:

Zone	Priority	Top elevation [m a.s.l.]
Default	0	50

Three red boxes highlight the following elements:

- + Add Zone:** A button to add a new parameter zone.
- Top elevation [m a.s.l.]:** An input field for setting a single value.
- Import raster file:** A button to import a raster file.

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.

The screenshot shows the 'MODEL SETUP' sidebar with 'Soil Layers' selected. The main panel displays the 'Properties' for the 'top' layer, with a 'Save' button highlighted in a red box. Below the map, the 'Smoothing' section is visible, and the 'Top elevation [m a.s.l.]' input field is set to 50, also highlighted in a red box.

Zone	Priority	Top elevation [m a.s.l.]
Default	0	50

Set single value to 50

Soil layer properties (4)

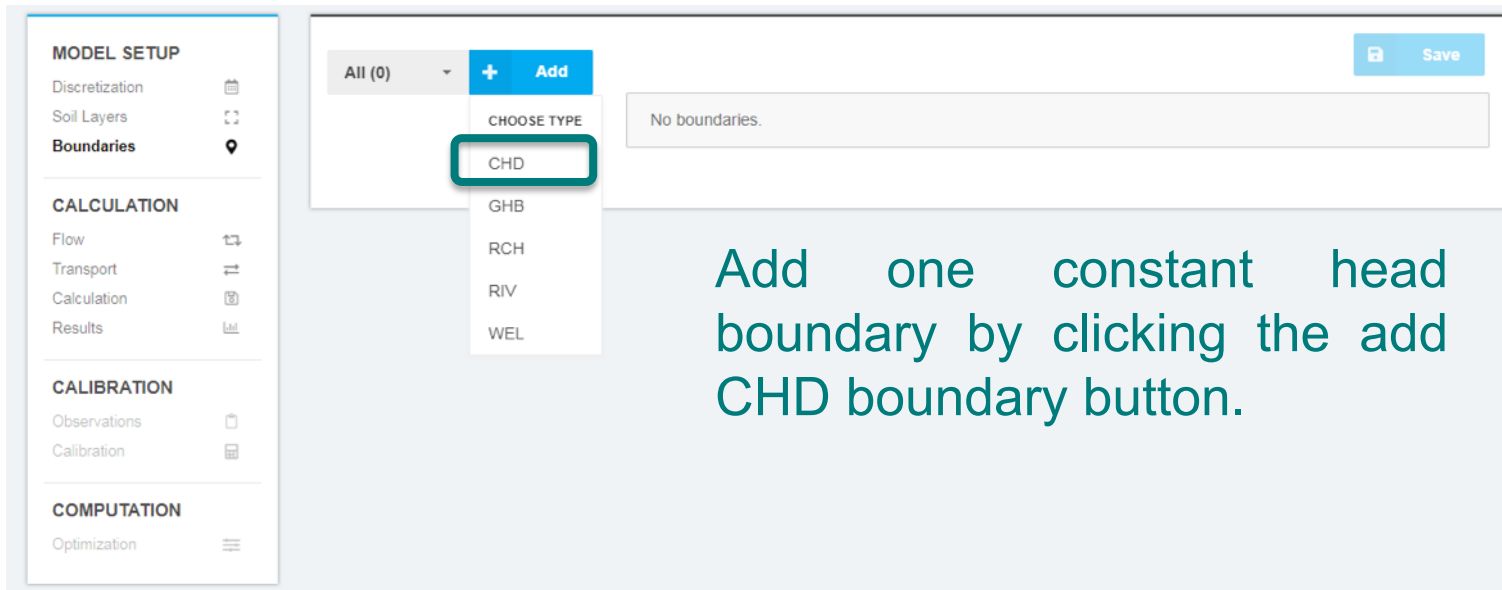
Now also define the following soil parameters as previously described for the top layer elevation.

- 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.



The screenshot shows the 'Boundaries' section of the software interface. On the left, a sidebar contains a 'MODEL SETUP' menu with 'Boundaries' selected. The main area shows a list of boundaries with a count of 'All (0)'. A blue '+ Add' button is visible, and a dropdown menu is open below it, listing boundary types: CHD, GHB, RCH, RIV, and WEL. The 'CHD' option is highlighted with a green box. A 'Save' button is in the top right. A text box in the center of the main area says 'No boundaries.'. A teal text overlay on the right side of the screenshot reads: 'Add one constant head boundary by clicking the add CHD boundary button.'

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

All (1) + Add

Type: CHD Name: CHD west Selected layers: Top layer x

CHD west ...

Edit boundary on map

Bacia do Pina

Corredor Rio

Brasília-Teimosa

Rua Delton

Rua Medusa

Rua Babilônia

Rua Paris

Rua Espalarte

Rua Anicim

Rua Poraque

Rua da Serra

Rua Garibaldi

Rua Artur Bernardes

Rua Djalberto Pires

Rua Afrânio

Avenida Brasil Formosa

Avenida Hercúlio Carneira

Tridino de Góes

Jardim Beira Rio

Rua São Luís

Rua São Paulo

Rua Artur Lages

Rua José Rios

Rua Cayo Cavalcanti

Rua Manoel

Rio de Janeiro

OP1

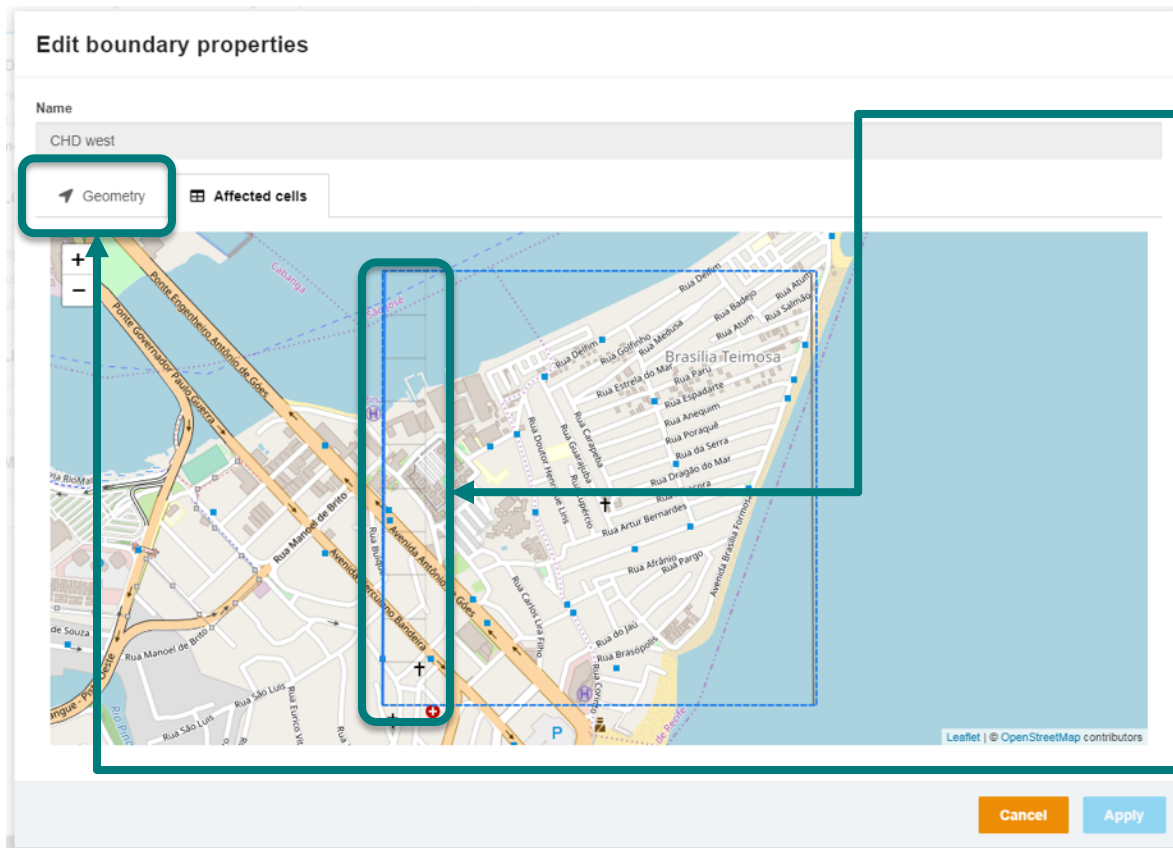
Leaflet | © OpenStreetMap contributors

Time dependent boundary values at observation point

Start Date	SHead (m)	Ehead (m)
01-01-2015	60	60

Boundary conditions (4)

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

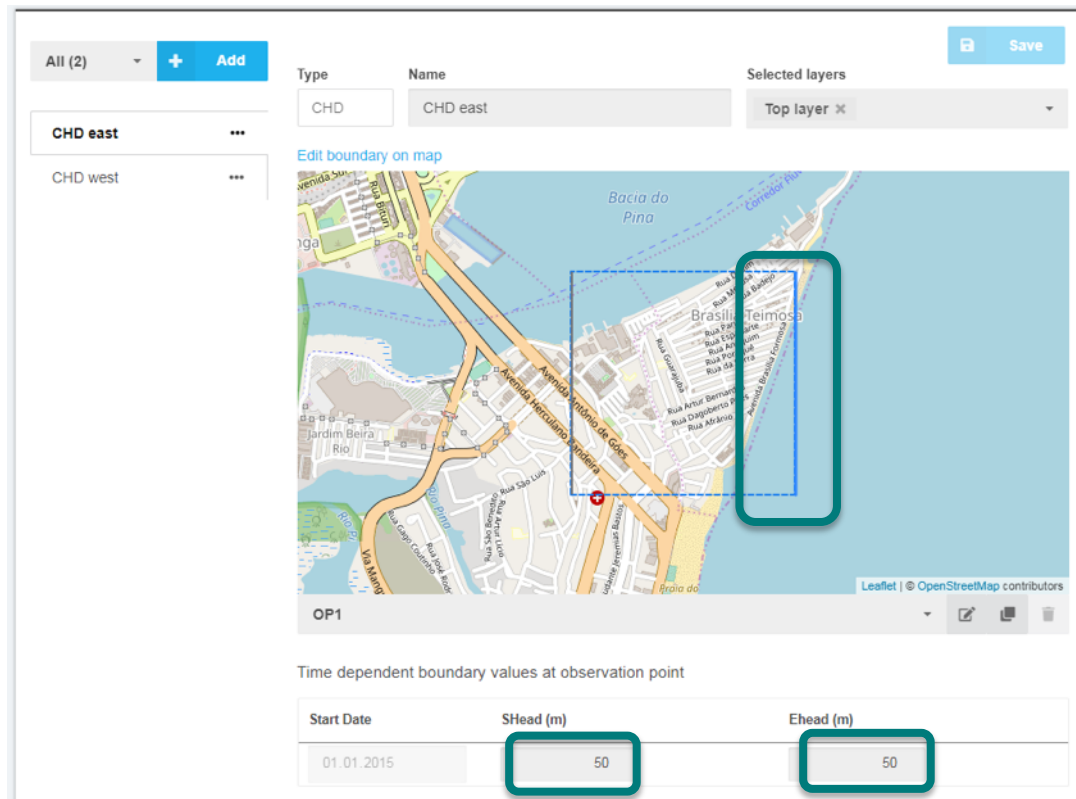


Make sure that at the western side of the boundary, all 10 cells are visible.

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).

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.



The screenshot shows a software interface for setting boundary conditions. On the left, there is a list of boundary types: 'CHD east' and 'CHD west'. The 'CHD east' boundary is selected. Below the map, there is a table for 'Time dependent boundary values at observation point'.

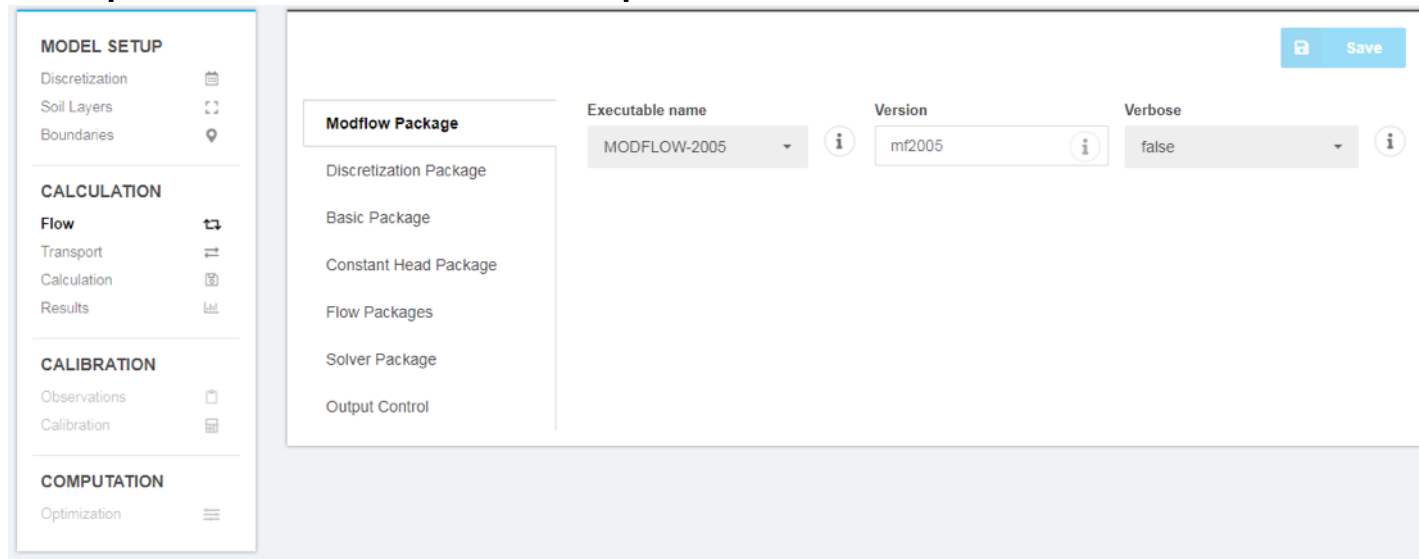
Start Date	SHead (m)	Ehead (m)
01.01.2015	50	50

Don't forget to save the changes!

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.

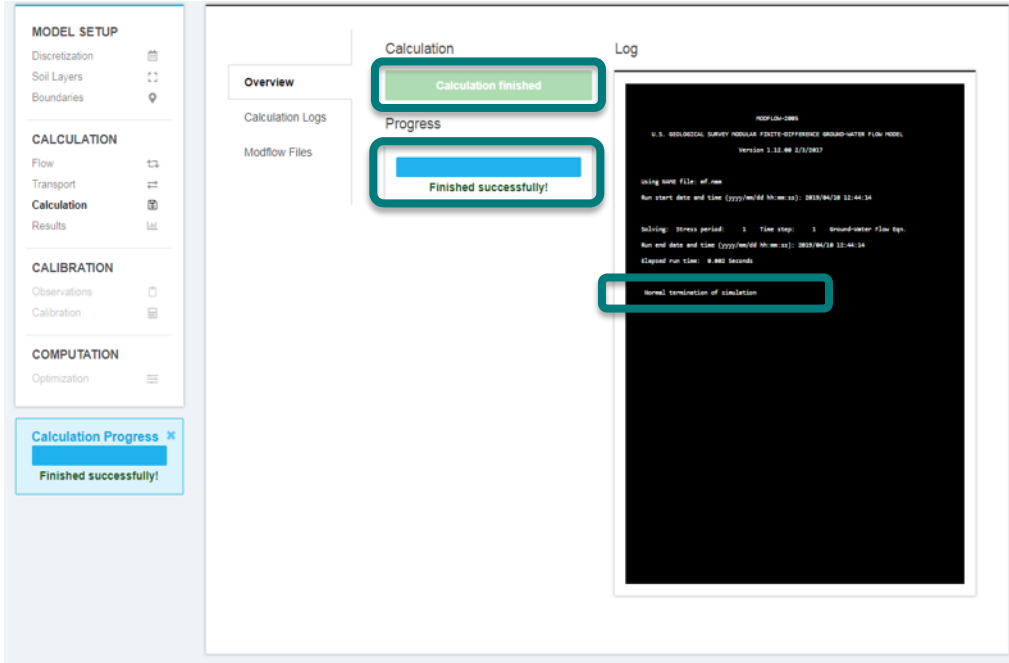
The preconditioned conjugate-gradient Package (**PCG**) as solver and the Layer-Property Flow Package (**LPF**) as flow package are utilized. All default parameters can be kept constant.



The screenshot shows a software interface for configuring MODFLOW parameters. On the left is a sidebar menu with sections: MODEL SETUP (Discretization, Soil Layers, Boundaries), CALCULATION (Flow, Transport, Calculation, Results), CALIBRATION (Observations, Calibration), and COMPUTATION (Optimization). The 'Flow' option is selected. The main panel shows a list of packages on the left: Modflow Package, Discretization Package, Basic Package, Constant Head Package, Flow Packages, Solver Package, and Output Control. The 'Modflow Package' is selected, and its parameters are shown on the right: 'Executable name' is a dropdown menu set to 'MODFLOW-2005', 'Version' is a text input field containing 'mf2005', and 'Verbose' is a dropdown menu set to 'false'. A 'Save' button is located in the top right corner of the main panel.

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 screenshot displays the software interface during a model calculation. On the left, a sidebar contains navigation options: MODEL SETUP (Discretization, Soil Layers, Boundaries), CALCULATION (Flow, Transport, Calculation, Results), CALIBRATION (Observations, Calibration), and COMPUTATION (Optimization). A 'Calculation Progress' window shows a blue progress bar and the text 'Finished successfully!'. The main area is divided into 'Calculation' and 'Log' sections. The 'Calculation' section shows a green 'Calculation finished' button and a blue 'Finished successfully!' button. The 'Log' section displays a terminal window with the following text:

```
MODFLOW-2005
U.S. GEOLOGICAL SURVEY MODULAR FLOWTO-OVERFLOW GROUND-WATER FLOW MODEL
VERSION 2.12.00 (2/3/2007)

Using name file: mf.nam
Run start date and time (yyyy/mm/dd hh:mm:ss): 2020/04/28 12:44:34

Initling... Stress period: 1 Time step: 1 Groundwater flow bps.
Run end date and time (yyyy/mm/dd hh:mm:ss): 2020/04/28 12:44:34
Elapsed run time: 8.462 seconds

Normal termination of simulation
```

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.

The screenshot displays a software interface with a sidebar on the left containing sections for MODEL SETUP, CALCULATION, CALIBRATION, and COMPUTATION. A 'Calculation Progress' indicator shows 'Finished successfully!'. The main area is divided into 'Overview', 'Calculation Logs', and 'Modflow Files'. The 'Modflow Files' section shows a 'File list' with files: mf.nam, configuration.json, mf.dis, mf.list, mf.hds, mf.bas, mf.chd, mf.pcg, mf.ddn, and mf.oc. The 'mf.list' file is highlighted with a red box. An arrow points from the text 'Other MODFLOW files (e.g. mf.list) can be checked here' to this box. To the right, the 'Content file: mf.list' is displayed on a black background with white text, showing the following content:

```
MODFLOW-2005
U.S. GEOLOGICAL SURVEY MODULAR FINITE-DIFFERENCE GROUND-WATER
VERSION 1.12.00 2/3/2017

LIST FILE: mf.list
UNIT 2

OPENING mf.dis
FILE TYPE:DIS UNIT 11 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL

OPENING mf.bas
FILE TYPE:BAS6 UNIT 13 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL

OPENING mf.chd
FILE TYPE:CHD UNIT 24 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL

OPENING mf.lpf
FILE TYPE:LPF UNIT 15 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL

OPENING mf.pcg
FILE TYPE:PCG UNIT 27 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL

OPENING mf.oc
FILE TYPE:OC UNIT 14 STATUS:OLD
FORMAT:FORMATTED ACCESS:SEQUENTIAL
```

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

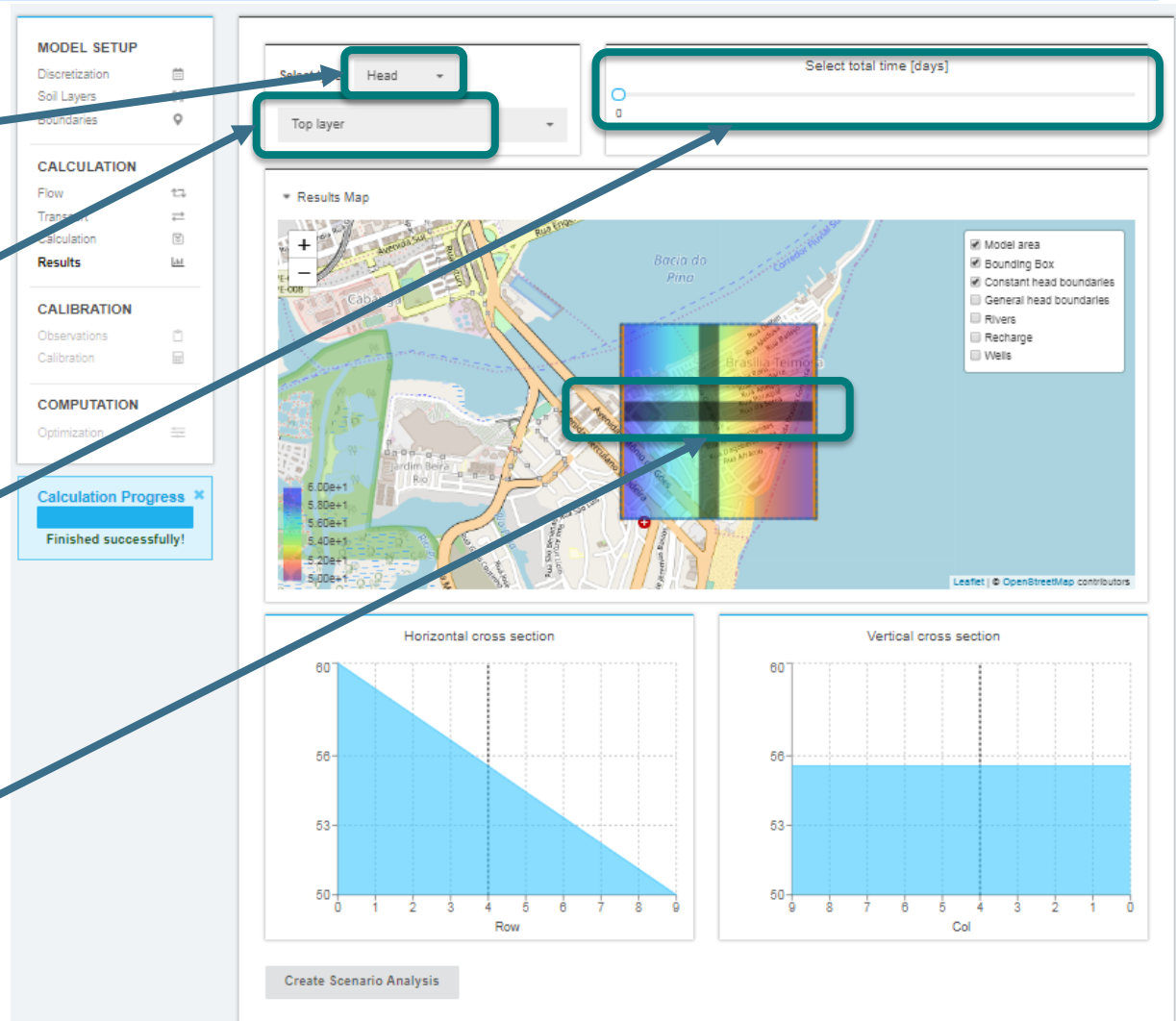
Results –head and drawdown visualisation

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!



Further Tutorials about the INOWAS platform:

Tutorial 2- transient groundwater flow modeling and scenario analysis

Tutorial 3- set up of solute transport model