File Menu

This menu allows you to open and save input files and create files that are used for creating initial head conditions for transient models.

Open

This selection opens a dialog that allows you to find and open existing input files (.anaq extension). These files store the data you edit under the Model Input, Plot Input, or Analysis Input menus in XML file format. XML is a common ASCII database file format. You could edit these directly with a text or XML editor, but that is not recommended since it risks corrupting input with improper values or format. When you open a file, the layout of elements in the model is drawn to the plot view.

If you want to be able to open .anaq files by double-clicking on them in Windows Explorer, in addition to opening them from the File/Open menu, the Windows operating system must associate .anaq files with Anaqsim. In case this association was not established during installation you can manually do it with Windows Explorer. To do this, locate a .anaq file in Windows Explorer. Right click on the file and then select Open with, then select Chose default program. In the dialog that pops up, check the box next to Always use the selected program to open this kind of file and then select the Browse button and browse to find the Anaqsim.exe file in the Program Files / Fitts Geosolutions / Anaqsim software directory. Now Windows will associate .anaq files with Anaqsim.exe, and you can open any .anaq file directly from Windows Explorer by double-clicking on it.

Save, Save As

The Save As option brings up a dialog that allows you to save your input to a file with a new name. Using Save saves the input to the same file name. If you have yet to save input and have no filename, it will function like Save As. When you save, you save the input data tables to an XML format database file with the .anaq extension.

Close

This closes the input you are working on and clears all the associated data tables in memory. After selecting Close, you may begin editing a new model.

Save locations for Initial Transient Heads

A transient model needs initial heads so it can compute the head change that occurs during the first time step. These values are needed at the location of each basis point in each spatially-variable area sink, which account for storage fluxes. The initial head values come from a pre-existing model, which could be steady or transient. Initial heads are also retrieved for discharge-specified wells, hydrograph points, and transient line conditions (see Analysis Input Menu for the last two items).

The initial conditions model must have the same number and extent of layers as the transient model, at least in areas with basis points, wells, hydrograph points, or transient line condition lines. When the initial head locations are written, each location is identified by its x,y coordinates and its layer. This is new in release 2020-1; prior releases wrote the x,y coordinates and the internal domain number. The change made in release 2020-1 allows different domain configurations between the initial and the transient models, which can be helpful. Because of this change, you must not mix initial head location or initial head files created prior to release 2020-1 with a release 2020-1 or later model. To avoid incompatibility when you switch to release 2020-1, re-create the initial head location file and the initial heads file as outlined below using release 2020-1.

To create a transient model that has a proper set of initial heads, these steps are necessary:

1. Make sure the transient model you begin to create has been saved to it's own unique file name, different from the file that contains the input for the model that will provide the initial condition heads.
2. Set up the transient model (i.e. uncheck Steady under Model Input/General, establish the time step sequence under Model Input/Time Steps, adjust input for the transient case, set up all spatially-variable area sinks, make the appropriate settings under Analysis Input/Hydrograph Points and Analysis Input/Transient Line Conditions, etc.
3. Select File/Save Locations for Initial Transient Heads, which saves the locations for transient starting heads from the transient model (these are the locations of all basis points associated with spatially-variable area sinks in the transient model and locations of hydrograph points, wells, and transient line conditions. This saves the level and the coordinates of each of these to a binary file with the .ihl extension.
4. Close the transient model.
5. Open the initial conditions model (the one that represents conditions at the start of the transient run). Solve it. Select File/Write Initial Transient Heads, click on the initial heads file you would like to use for this simulation, and click Open. This reads in the locations saved from the .ihl file saved in step 3.
6. A dialog opens asking you to name the binary initial heads file. The default name is the same as the initial conditions input file but with the .hds ending. Anaqsim then writes the initial heads at the locations in the .ihl file to a binary file with the .hds file extension.
7. Close the initial conditions model.
8. Open the transient model. After checking that all model parameters are set correctly for the transient run, select Solve. At this point Anaqsim will ask you to select the .hds file containing the initial heads created in step 6.

When you solve the transient model, the heads are read in and used to determine the head change in the first time step at each basis point.

Write Initial Transient Heads

See the discussion under Save Locations for Initial Transient Heads for an overview of setting up initial heads for transient models.

Save Solution

This allows you to save the model solution after you have solved. Later, you can open the model input file, then load the saved solution, and avoid the "Solve" step. This is particularly handy for large models that have longer solve times, and allows you to save your solution and come back to it later for making plots or doing analysis of the solution. All model objects with their strengths are saved in a binary format, to a file that has the same name as the input file, but with the ".solu" filename extension instead of ".anaq".

Load Saved Solution

This allows you to load in a previously saved model solution. To make use of this, first open the model input file for the model, then load the saved solution, which avoids the need for the "Solve" step. This is particularly handy for large models that have longer solve times, and allows you to save your solution and come back to it later for making plots or doing analysis of the solution. All model objects and their strengths are read in from a binary file that has the same name as the input file, but with the ".solu" filename extension instead of ".anaq".

Export Input Data to Excel File

This causes the entire input database to be written to one excel file in the same directory as the input file (*.anaq), written to a file with the same name but the excel suffix (*.xlsx). The Excel file has multiple sheets, one for each data table. Each sheet contains the same headers as the data tables plus all rows of input. This is a handy way to document model inputs all in one readable file.

Exit

This exits Anaqsim. A dialog asks if you want to save the current input before exiting. The same is achieved by clicking on the red "x" at the upper right corner of the Anaqsim window.

Revision #3
Created 1 July 2023 13:09:03 by Mary Jeddere-Fisher
Updated 12 January 2024 15:16:59 by Mary Jeddere-Fisher