Purposes: (1) To illustrate how changing the solver parameters can affect the results. (2) To illustrate how changing the step size can affect the results.

The model will have 100 rows and columns each 100 m wide and a single layer 10 m thick. The top of the model has an elevation of 0. There is one transient stress period with a length of 100000 s (= 27.8 hours). Use a single time step for the initial model. The aquifer properties will use the default values for ModelMuse.

Kx = 0.0001 m/s

Specific Storage 1e-1 m-1

Change the output file type for the heads to binary.

Activate the well package and place a single well in the center of the model with a pumping rate of -0.01 m3/s = (36 m3/hour).

First set of model variations: Set HCLOSE and RCLOSE among the following. For each model run list the cumulative percent discrepancy and make a contour plot of the results.

|  |  |  |
| --- | --- | --- |
| HCLOSE | RCLOSE | Cumulative Percent Discrepancy |
| 1 | 1 |  |
| 0.1 | 0.1 |  |
| 0.01 | 0.01 |  |
| 0.001 | 0.001 |  |
| 0.0001 | 0.0001 |  |

What differences do you observe among the contour plots?

Second set of model variations: Delete the model results and then run the model again with the values of HCLOSE and RCLOSE that you think give the best results. Import those model results and make a contour plot of them.

Vary the number of time steps in the stress period among the following values. For each model run list the cumulative percent discrepancy and make a contour plot of the results.

|  |  |
| --- | --- |
| Number of steps | Cumulative Percent Discrepancy |
| 1 |  |
| 10 |  |
| 100 |  |
| 1000 |  |

What differences do you observe among the contour plots?

Does a poor percentage discrepancy indicate that there is a problem with the model results?

Does a good percentage discrepancy indicate that there is not a problem with the model results?