Journal of Hydrology (2008) **351**, 261–267

available at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/jhydrol

Three-dimensional mesh resolution control in finite difference groundwater flow models through boxed spatial zooming

Ferenc Székely *

Bocskai utca 13, 1204 Budapest, Hungary

Received 4 March 2007; received in revised form 25 November 2007; accepted 30 November 2007

KEYWORDS
Multi-mesh modeling;
Boxed spatial zooming;
Mesh interface simulator;
Well flow modeling;
MODFLOW compatibility

Summary The iterative composite mesh simulation (CMS) technique operates a coupled system of point centered finite difference groundwater flow models. It allows for high lateral and vertical mesh resolution at the sites of interest, whereas a coarser mesh may be applied in other parts of the formation. The boxed spatial zooming method utilizes a ''box-in-box'' architecture to build the coupled 3D system of embedded (nested) parent and child meshes exhibiting different vertical and lateral extensions and resolutions. A mesh interface simulator is used to equate the heads and balance the fluxes along the common vertical plains and shared lateral layers (interfaces) linking the meshes. This multi mesh simulation method is used by software FLOW and has been successfully tested in multi-aquifer systems against selected unsteady and steady state well flow problems with available analytical solutions. The base compatibility with MODFLOW databases supports wider use of the FLOW simulator.

Introduction

Practical application of finite difference groundwater flow models involves selection of an appropriate mesh resolution in both the lateral and vertical directions according to the problem under consideration. In many cases the actual model investigation can not be confined to a certain site and the actual site model needs to be connected to the available local, regional or even basin scale models. Since the latter employ coarser mesh, special techniques are required to bridge the resolution contrast. The need for high resolution mesh is usually limited to a certain sub-domain extending over a given distance and/or depth. The vertical mesh resolution control may play an important role in the following cases. Shallow groundwater projects (irrigation, drainage, and aquifer cleanup) request high resolution mesh only in the upper sections of the formation, the deep and remote parts of the flow domain may be represented in lower resolution. In contrast, geothermal or deep waste disposal projects may need high lateral and vertical mesh resolution at depth of a few kilometers while the aquifer system away

* Tel.: +36 20 3881730; fax: +36 1 2878000.
E-mail address: fszekely@vnet.hu.

0022-1694/$ - see front matter
doi:10.1016/j.jhydrol.2007.11.038

from the well site and in the overlying part of the formation may be simulated at sufficiently coarser resolution.

The author has developed an iterative, multi-mesh, windowed spatial zooming method (Székely, 1998) allowing for coupling four embedded or nested meshes (models). Homogeneous formation, steady state, three-dimensional (3D) flow is considered, the meshes of different lateral and vertical resolution exhibit telescopic layout and fully penetrate the formation. Vertical planes or interfaces separate the embedded meshes and the point centered finite difference method is used to locate boundary nodes exactly on the interface. Both the first order (heads) and second order (fluxes) boundary conditions are satisfied during the mesh coupling procedure. The entire flow domain is split into several nested sub-models therefore the procedure is referred to as the composite mesh simulator or CMS. The method was extended to analyze unsteady drawdown evolution in heterogeneous formations (Székely, 1999). Based on a literature review, Mehl and Hill (2002) found this method appropriate and efficient for mesh resolution control and adjusted this technique for the 2D block centered MODFLOW (McDonald and Harbaugh, 1988) modeling environment. After correcting the discrepancy caused by the dual flow in overlapping boundary zones, authors applied the method to heterogeneity analysis considering steady state flow. Later the authors expanded this method to 3D transient flow analysis (Mehl and Hill, 2005).

The objective of this study is to extend the windowed spatial zooming method to the system of vertically nested meshes, when the smaller child models partially penetrate the incorporating parent ones. This three-dimensional ''box-in-box'' architecture is called boxed spatial zooming and provides a flexible mesh resolution control facility in both the horizontal and vertical directions.

The boxed spatial zooming technique

Fig. 1 shows the vertical section of a CMS modeling system comprising three meshes. The master mesh M1 extends over the whole simulation domain. The intermediate mesh M2 occupies the central and lower part of M1, whereas the target mesh M3 is located in the middle zone of M2. The pairs M1—M2 and M2—M3 meshes exhibit parent—child relationship. The domains occupied by child models are represented as impermeable sub-domains in parent meshes. The blocks of parent models occupied by the child models are made impermeable by the software during simulation. The vertical interfaces are set along the separating vertical planes, the lateral interfaces follow the model layers shared by the parent and child meshes. In this particular case mesh M2 has one external or outer lateral interface with M1 in the shared layer marked with dashed lines. Mesh M3 has two outer lateral interfaces at the top and the bottom of model layers shared with mesh M2 (marked with crosses). The outer interfaces of child models are considered as internal to the parent models. Nodes at all outer interfaces are assumed to be fix head boundary nodes, whereas nodes along internal interfaces are considered as variable head nodes. The main objective of the CMS simulation is to provide head distribution along the fix head boundary of the target mesh.

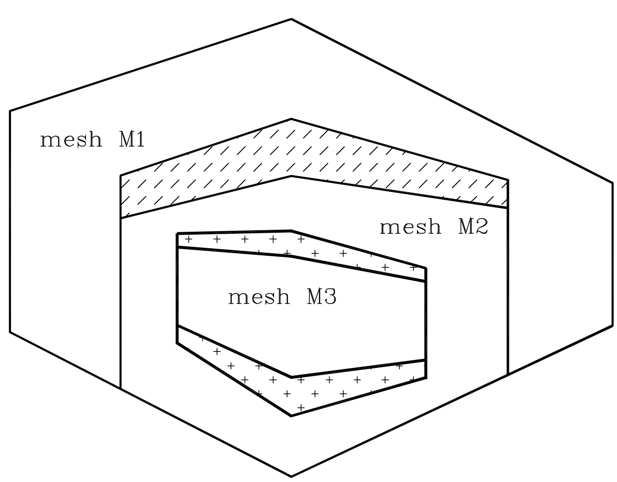

Figure 1 Vertical section through a mesh system M1—M2—M3 with vertical and lateral interfaces.

The iterative mesh interface simulator (MIS) has been primarily developed to simulate the response to groundwater extraction or recharge applied in the target mesh (Székely, 1998). The method can also be successfully utilized also in cases when external stress is additionally applied in parent meshes. MIS iteratively activates (runs) meshes M1, M2 and M3 (or additionally M4) in a predefined sequence at each time step. Considering a certain external flux (pumping, injection, infiltration, evapotranspiration, leakage from/to streams or lakes) in the target mesh M3 a bidirectional scan over the mesh system is performed.

In case of a drawdown simulation, an upstream scan is applied first which starts by running model M3 at assumed (zero) boundary conditions at all vertical and lateral interfaces. This simulation yields an approximate (biased) flux function in the outer boundary nodes to be passed to the coarser parent mesh M2. A simulation is performed in mesh M2 and the approximate boundary fluxes are transferred to the master mesh M1. The upstream scan completes with simulation over M1 considering the actual boundary conditions according to the recharge—discharge conditions of the master model. This run generates an approximate drawdown distribution at the M1—M2 interface which is used at the start in downstream scan.

The latter begins with the simulation of the drawdown effect in M2 with previously estimated boundary fluxes at M2—M3 interface and at improved internal head boundaries just imported from mesh M1. This simulation provides a corrected first order boundary condition for mesh M3. Iteration concludes with simulation in mesh M3 using the improved boundary heads and the predefined external stresses. The inter-mesh iterations (scans) terminate at a predefined maximum number. The MIS technique provides balance of fluxes across and equality of heads along all the interfaces, which is the main advantage of the CMS simulation. In their 2D and 3D studies Mehl and Hill (2002, 2005) apply a preparatory simulation using the coarse parent mesh to approximate head distribution over the entire domain. This defines a reasonable initial head distribution at the interface of parent (master) and child (target) meshes.

The groundwater flow simulation program FLOW by the author with the option of CMS modeling is used in this study. The finite difference equation written for a particular node