WORKSHOP

ON

MODELLING OF HYDROLOGIC SYSTEMS

4-8 September, 2000

MODFLOW

by

B. Chakravorty

Organised by

Hard Rock Regional Centre
National Institute of Hydrology
Belgaum-590 001 (Karnataka)

MODFLOW

Biswajit Chakravorty
Scientist 'C'
Ganga Plain North Regional Centre
National Institute of Hydrology
Patna- 801 505 (Bihar)

1. Introduction to MODFLOW

MODFLOW is a MODular 3-dimensional finite difference groundwater FLOW model developed by McDonald and Harbough of the USGS, USA in 1988. It simulates steady and non-steady flow in three dimensions for an irregularly shaped flow system in which aquifer layer can be confined, unconfined, or a combination of confined and unconfined. Flow from external sources, such as flow to wells, areal recharge, evapotranspiration, flow to drains, and flow through river, can be simulated

MODFLOW uses a modular structure wherein similar program functions are grouped together. The modular structure consists of a main program and a large number of independent subroutines called "modules". The modules, in turn, have been grouped into "packages". Each package deals with a specific aspect of the hydrological system to be simulated. For example, the option Well package simulates the effect of wells, the River package simulate the effect of river etc.

1.1 Mathematical Background

The three dimensional unsteady movement of groundwater of constant density through porous earth material in a heterogenous anisotropic medium can be described by the following partial differential equation:

$$\frac{\partial}{\partial x}\left(K_{xx}\frac{\partial h}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_{yy}\frac{\partial h}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_{zz}\frac{\partial h}{\partial z}\right)-W=S_s\frac{\partial h}{\partial t} \qquad (1)$$

Where,

K_{xx}, K_{yy}, K_{zz}	:	hydraulic conductivity along major axes [LT^{-1}],
h	:	potentiometric head [L],
W	:	volumetric flux per unit volume and represents sources and/or sinks of water [T^{-1}],
S_s	:	specific storage of the porous material [L^{-1}] and,
t	:	time [T].

In general, S_s, K_{xx}, K_{yy} and K_{zz} are function of space, for example; $S_s = S_s(x,y,z)$, $K_{xx} = K_{xx}(x,y,z)$, etc. whereas W and h are functions of space and time i.e $W = W(x,y,z,t)$ and $h = h(x,y,z,t)$. Equation (1) together with specification of flow conditions at the boundaries of an aquifer system and specification of initial head conditions, constitutes a mathematical model of ground water flow.

Except for very simple systems, analytical solutions of equation (1) are rarely possible. So, various numerical methods are employed to obtain an approximate solution of the above equation. One such approach is the *finite-difference* method. The continuous system described by equation (1) is replaced by a finite set of discrete elements in space and time, and the set of finite difference equations are solved numerically which yields *values of head* at specific points and times. These values constitute an approximation to the time-varying head distribution that would be given by an analytical solution of the flow equation.

Possible inflow/outflow terms (W)

- Recharge from rainfall.
- Artificial recharge through wells.
- Pumping through wells.
- Evapotranspiration loss.
- Recharge from river/canal cells.
- Outflow into a river/canal cell.
- Inflow/Outflow across a boundary cell.
- Outflow through drains.
- Spring flow.
- etc.

Input to the Model

- Boundary data.
- Geometry of aquifer.
- Permeability(ies).
- Effective porosity.
- Specific storage.
- Recharge data.
- Evapotranspiration data.
- Surface water – ground water exchange.
- Pumping from/to aquifer.
- Artificial recharge.
- etc.

Output from the Model

- Water levels.
- Drawdowns.
- Water balance of the whole model domain and required sub-regions.
- Distribution of aquifer parameters.
- Distribution of evapotranspiration.
- Distribution of recharge.
- Outflow across model boundaries.
- etc.

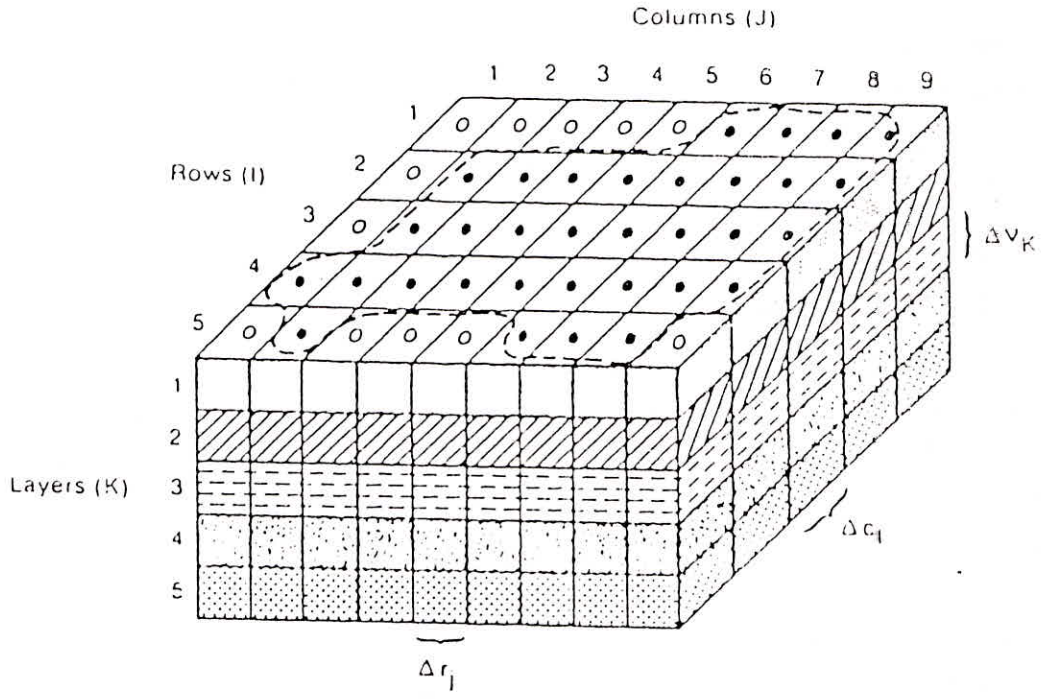
1.2 Model Discretization

MODFLOW discretizes the model domain with a mesh of blocks called `cells' in which medium properties are assumed to be uniform. The plan view rectangular discretization results from a grid of mutually perpendicular lines may be variably spaced. The varying thickness of vertical layers of aquifer systems are transformed into a set of parallel `layers'. The location of each cell is described in terms of rows, columns, and layers. Figure 1 shows the discretization of aquifer system with 5 rows, 9 columns and 5 layers. The width of cells in the row direction at a given column (j) is designated Δr_j, the width of cells in the column direction at a given row (i) is designated Δc_i, and the thickness of cells in a given layer (k) is designated Δv_k. Within each cell there is a point called a 'node' at which groundwater head is to be calculated.

The model distinguishes a cell into: i) variable-head cell (the head varies with time), ii) constant head cell (the head is constant), or iii) no flow or inactive cell (no flow takes place within the cell). Figure 2 illustrates the distribution of boundary code entries typical model layer. This kind of arrays are to be defined for each layer.

The period of simulation is divided into a series of 'stress period' within which stress parameters are constant. Each stress period, in turn, is divided into a series of time steps as shown in Figure 3. The user specifies the length of the stress period, the number of time steps at each stress period, and the time step multiplier. Using these terms, the program calculates the length of each time step in the stress period.

With above discretization in space and time, equation (1) will lead to a system of simultaneous linear algebraic equation. MODFLOW utilizes iterative methods to obtain the solution of the system's equation.



Explanation

- Aquifer Boundary
- Active Cell
- 0 Inactive Cell
- Δr_J Dimension of Cell Along the Row Direction. Subscript (J) Indicates the Number of the Column
- Δc_I Dimension of Cell Along the Column Direction. Subscript (I) Indicates the Number of the Row
- Δv_K Dimension of the Cell Along the Vertical Direction. Subscript (K) Indicates the Number of the Layer

Figure 1: A discretized hypothetical aquifer system.

1.3. Overall Structure

MODFLOW consists of a main program and a large number of highly independent subroutines called 'modules' and 'packages'. The functions performed for a typical simulation are shown with the help of a Flow Chart in Figure 4. Thus, within a simulation, there are three nested loops: a stress-period loop, within which there is a time-step loop, which in turn contains an iteration loop.

1.4 Summery of MODFLOW Packages

Various modules are provided in MODFLOW to deal with various field situations. These are Basic package, Block centered flow package, River package, Recharge package, Well package, Drain package, Evapotranspiration package, General-head boundary package and the simulation technique packages viz., Strongly implicit package (SIP), Slice-successive overrelaxation package (SSOR) etc. Conceptualization and implementation of these modules are briefly discussed below:

1.4.1. Basic package (BAS)

The Basic package handles a number of administrative tasks for the model It reads data for:

i. the discretization of the model domain with a mesh of blocks called cells, the location of which are described in terms of rows, columns and layers as described in Figure 1.

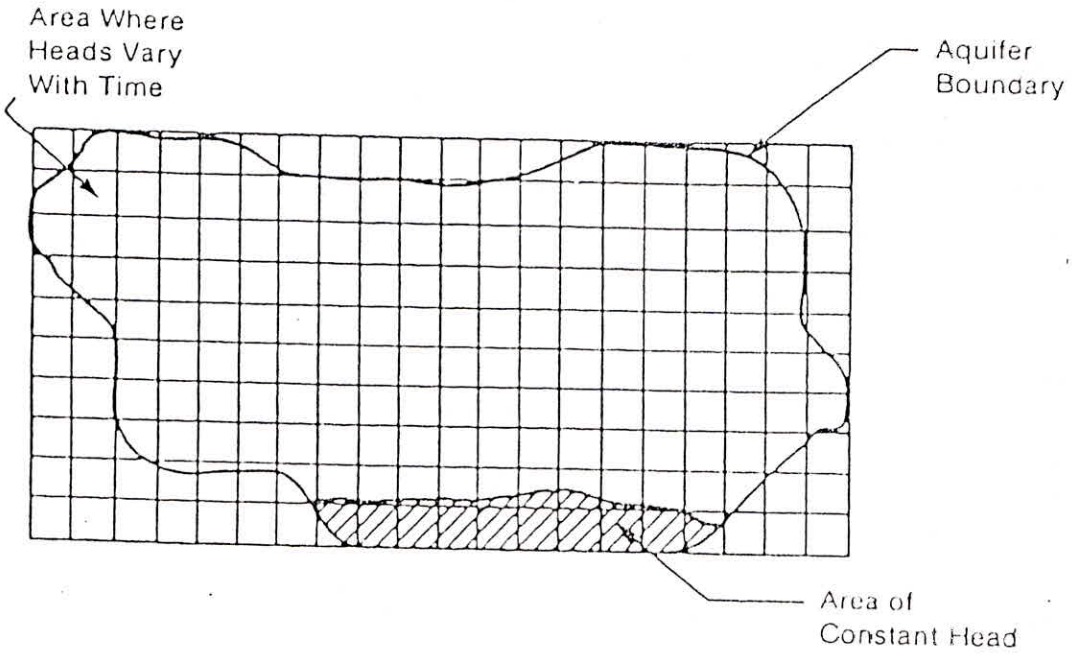
ii. the selection of major options and the designation of their input unit numbers. For a given model, the user has to select the major options by specifying a unit number of input file. The program links the unit numbers with the corresponding input files which are to be prepared by the user.

iii. specifying initial and boundary conditions after distinguishing a cell into variable-head, constant-head or inactive cell. Figure 2 illustrates distribution of bondary code entries for a typical model layer. For specifying the initial condition, the 'starting heads' for the first time step are specified by the user

iv. the discretization of simulation time into sterss period, time step and the time step multiplier as illustrated in Figure 3. The user specifies the length of stress period, the number of time steps at each stress period, and the time step multiplier. Using these terms, the program calculates the length of each time step in the stress period.

1.4.2. Block-centered flow package (BCF)

The Block-centered flow package (BCF) computes the conductance components of the finite-difference equation which determine flow between adjacent cells. It also computes the terms that

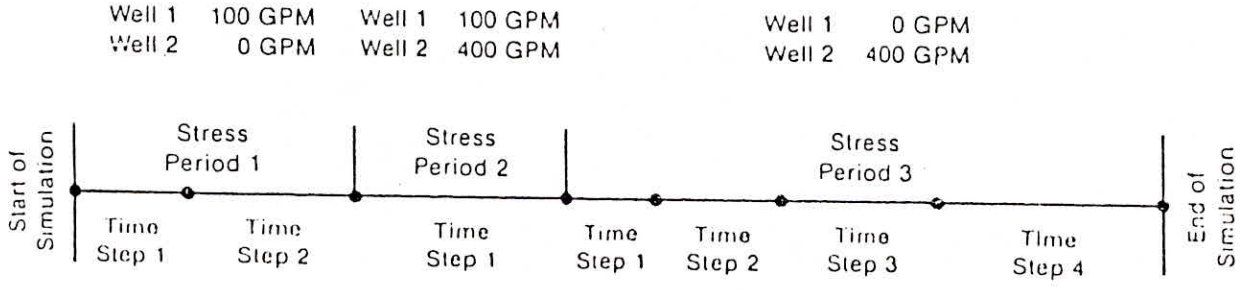


0	1	1	1	1	1	0	0	0	0	0	0	0	1	1	1	1	1	0	0
1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
0	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
0	0	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
0	0	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
0	0	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
0	0	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
0	0	0	0	0	0	1	1	1	1	1	1	1	1	1	1	1	1	0	0
0	0	0	0	0	0	0	-1	-1	-1	-1	-1	-1	-1	-1	-1	-1	0	0	0

IBOUND Codes

< 0 Constant Head
= 0 No Flow
> 0 Variable Head

Figure 2: Example of the boundary array (IBOUND) for a single layer.



$$Delt(1) = \frac{PERLEN * (1-TSMULT)}{1 - TSMULT^{**}NSTP}$$

$$Delt(m+1) = TSMULT * Delt(m)$$

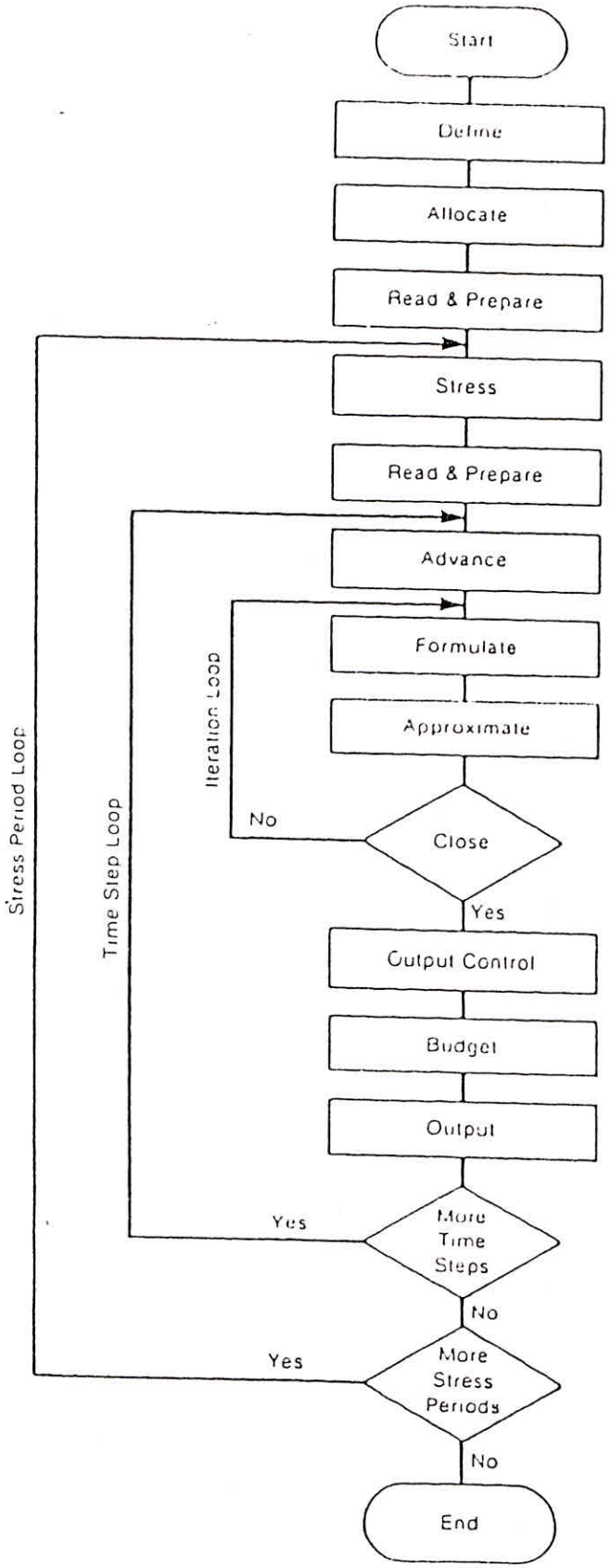
Specified by User

PERLEN.....Length of Stress Period
TSMULT.....Time Step Multiplier
NSTP.....Number of Time Steps in Stress Period

Calculated by Program

Delt(m).....Length of Time Step m

Figure 3: Division of simulation time into stress periods and time steps.



DEFINE — Read data specifying number of rows, columns, layers, stress periods, and major program options.

ALLOCATE — Allocate space in the computer to store data

READ AND PREPARE — Read data which is constant throughout the simulation. Prepare the data by performing whatever calculations can be made at this stage.

STRESS — Determine the length of a stress period and calculate terms to divide stress periods into time steps

READ AND PREPARE — Read data which changes from one stress period to the next. Prepare the data by performing whatever calculations can be made at this stage

ADVANCE — Calculate length of time step and set heads at beginning of a new time step equal to heads calculated for the end of the previous time step.

FORMULATE — Calculate the coefficients of the finite difference equations for each cell

APROXIMATE — Make one cut at approximating a solution to the system of finite difference equations.

OUTPUT CONTROL — Determine whether results should be written or saved on disk for this time step. Send signals to the BUDGET and OUTPUT procedures to indicate exactly what information should be put out

BUDGET — Calculate terms for the overall volumetric budget and calculate and save cell-by-cell flow terms for each component of flow.

OUTPUT — Print and save heads, drawdown and overall volumetric budgets in accordance with signals from OUTPUT CONTROL procedure

Figure 4: Overall program structure.

determine the rate of movement of water to and from storage.

1.4.3. River Package (RIV)

Rivers and streams contribute water *to the groundwater system or drain water from it* depending on the head gradient between the river and the groundwater. The purpose of the river package (RIV) is to simulate the effect of flow between surface water features and groundwater systems. To do this, a river or a stream is divided into reaches (Figure 5) so that each reach is completely contained in a single cell. Stream aquifer seepage is simulated between each reach and the model cell that contains that reach. It is assumed that there is no change in the stream stage because of the stream-aquifer interaction.

If a stream-aquifer system shown in Figure 6a can be conceptualised such that the stream-aquifer interconnection is represented as a simple conductance through which one dimensional flow occurs (Figure 6b), flow between the stream and groundwater in each cell is given by:

$$QRIV = CRIV * (HRIV - h_{i,j,k}) \quad \text{for } h_{i,j,k} > RBOT$$

or,

$$QRIV = CRIV * (HRIV - RBOT) \quad \text{for } h \leq RBOT \qquad (2)$$

where,

- QRIV is the flow between the stream and the aquifer,
- CRIV = K*L*W/M, is the hydraulic conductance of stream-aquifer interconnection. K is the conductivity of the streambed, L is the length of the stream reach, W is the width of the stream reach, and M is the thickness of the streambed.
- HRIV is the stream level.
- $h_{i,j,k}$ is the groundwater head at the node in the cell underlying the stream reach.
- RBOT is the elevation of the bottom of the streambed layer.

If hydrologic condition indicate that seepage from a stream will increase as the local water table declines, but will reach the limiting condition (Figure 7) when water table reaches the elevation h_l, RBOT should be taken equal to h_l, and CRIV should be calculated as $K*L*W/(HRIV-h_l)$. Because the vertical hydraulic gradient beneath the stream is approximately one, seepage from stream into cell i,j,k is given approximately by the product K*L*W.

1.4.4. Recharge Package (RCH)

The recharge package (RCH) is designed to simulate areally distributed recharge to the

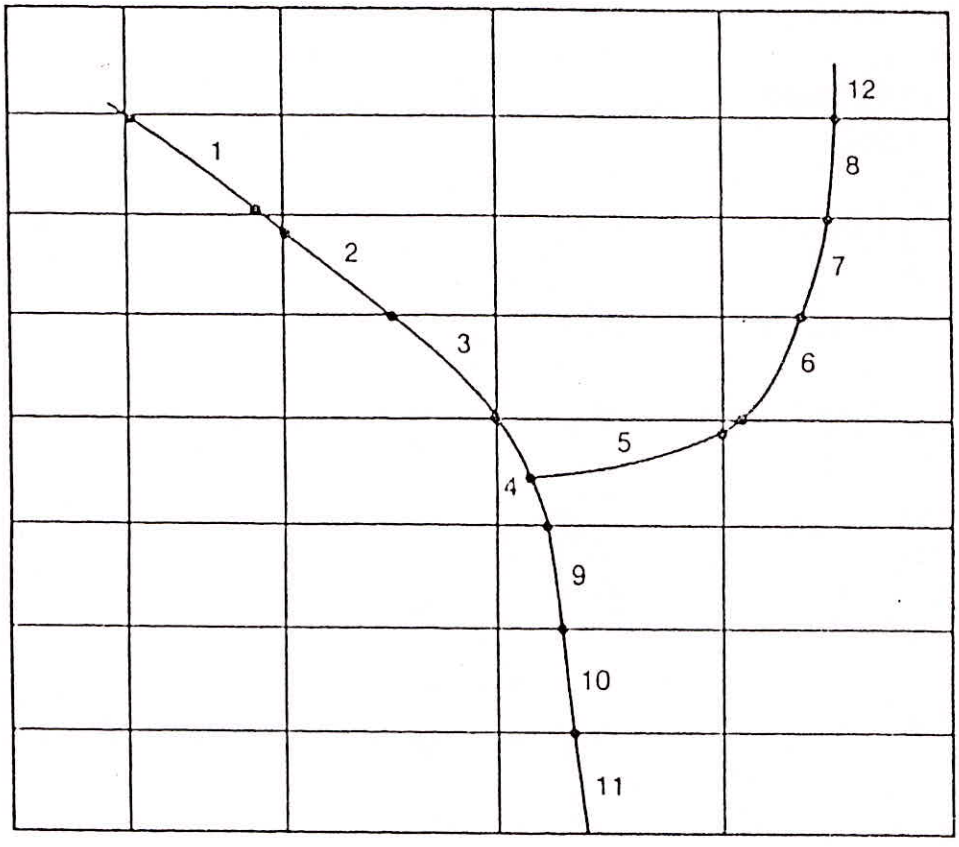
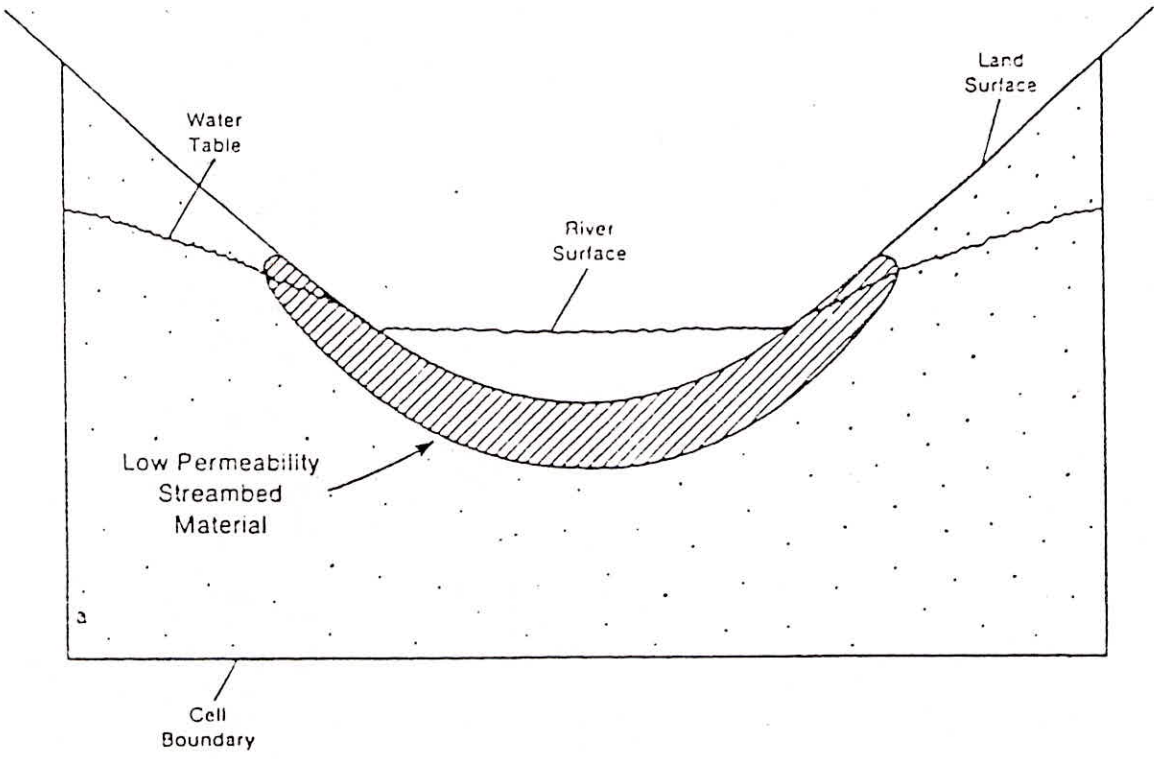


Figure 5: Discretization of a stream into reaches. Some small reaches are ignored.



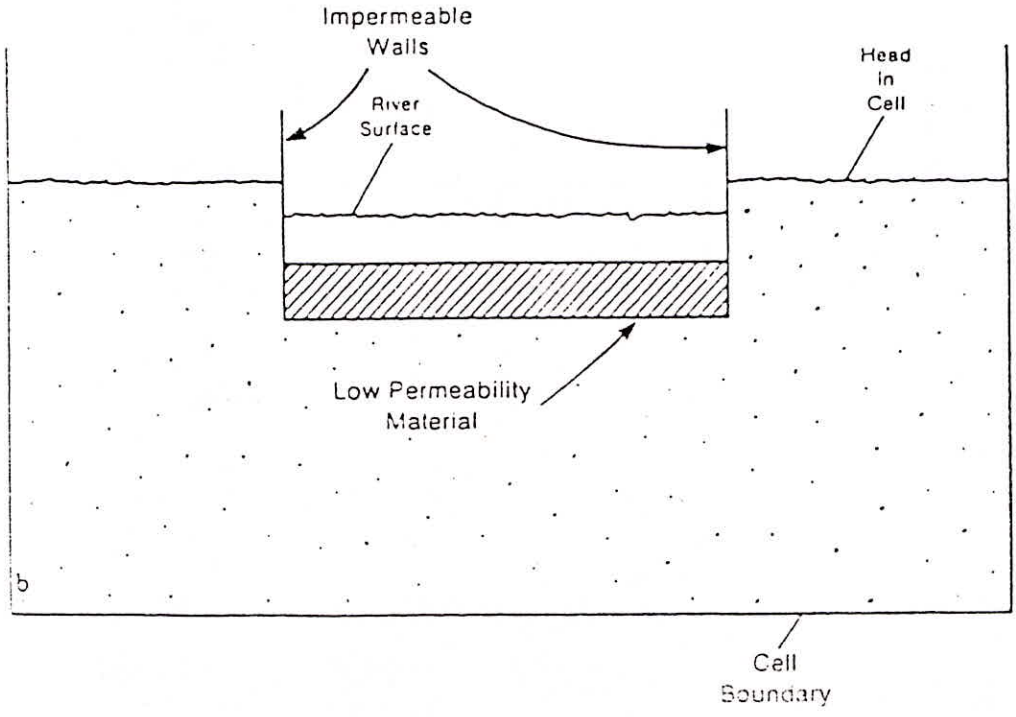


Figure 6 (a) Cross section of an aquifer containing a stream and (b) Conceptual representation of stream-aquifer interconnection in simulation.

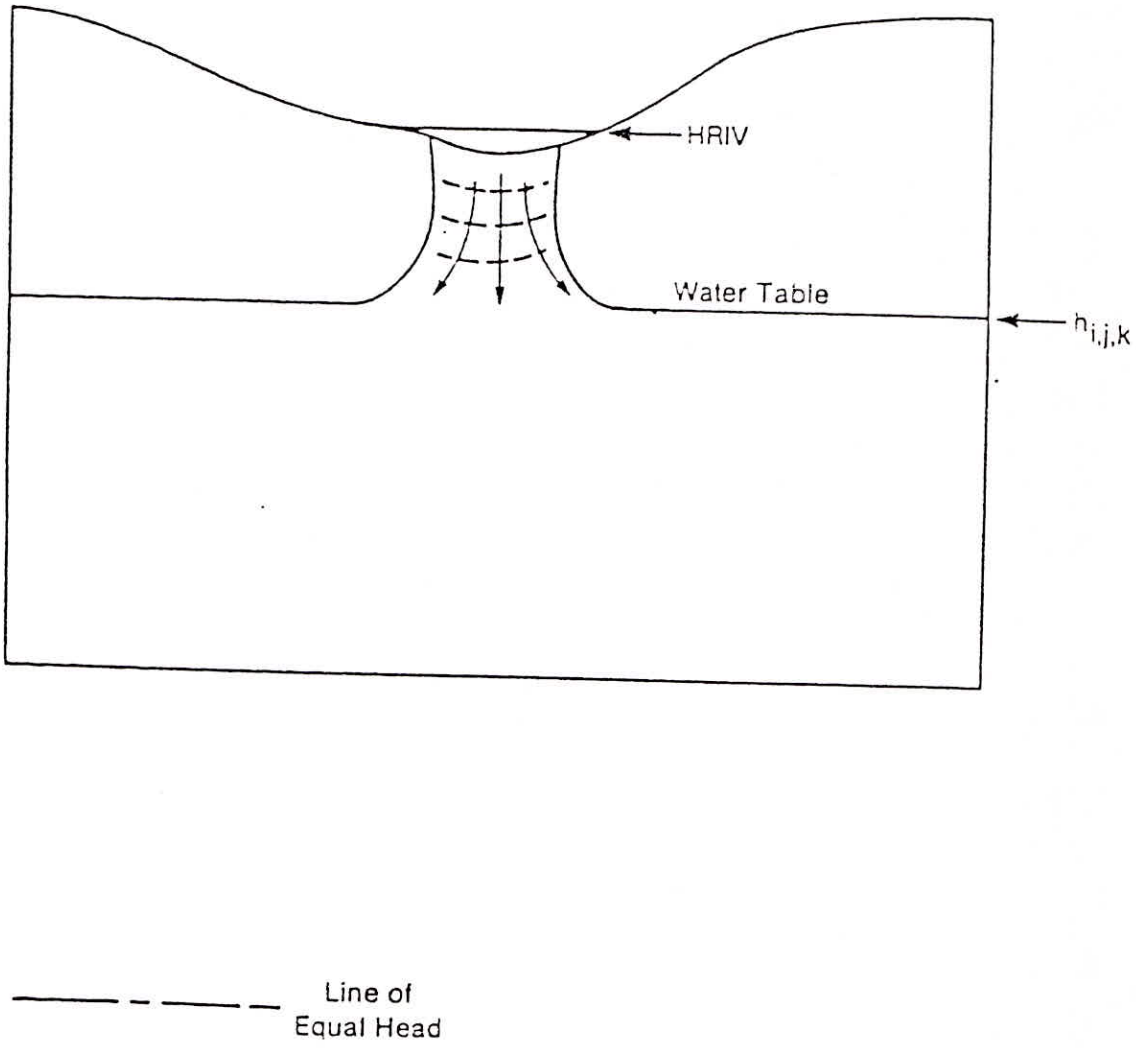


Figure 7: Limiting seepage from a stream at unit hydraulic gradient.

groundwater flow system. Most commonly, areal recharge to the groundwater are from the precipitation infiltration. Since there is no method devised for estimating groundwater recharge. So, the recharge rate is assumed to be equal to some percentage of average annual precipitation. The recharge is often adjusted during calibration. Recharge applied to the model is defined as

$$QR_{i,j} = I_{i,j} * \Delta R_j * \Delta C_i \quad (3)$$

Where,

$QR_{i,j}$ is the recharge to the cell (i,j). $[L^3 T^{-1}]$

$I_{i,j}$ is the recharge flux to the map area, $\Delta R_j * \Delta C_i$, of cell. $[LT^{-1}]$.

In the package, values of recharge flux, $I_{i,j}$, are read into a two dimensional array. The cell within each vertical column to which recharge is applied is specified through the recharge option code. The options include:

i. Application of recharge to model layer 1. There is no recharge added to no-flow cells.

ii. Application of recharge to any cell in the vertical column as specified by layer numbers. No recharge is added to the no-flow cells.

iii. Application of recharge to the uppermost active cell in the vertical column provided there is no constant head cell above it in the column. If there is a constant-head cell in a vertical column of cells and there is no active cell above, then no recharge is applied to this column.

1.4.5. Well Package (WEL)

The Well package (WEL) is designed to simulate the inflow or outflow through recharging or pumping wells. Wells are handled in the Well package by specifying the location of each individual well and its rate, Q. Negative values of Q are used to indicate well discharge, while positive values of Q indicate a recharging well.

1.4.6. Drain Package (DRN)

The Drain Package is designed to simulate the effect of both open and closed drains. The Drain Package works in much the same way as the River Package, except the leakage from the drain to the aquifer is not allowed. Figure 8 shows a cross section through a cell containing a drain. The drain is assumed to run only partially full, so that head within the drain is approximately equal to the median drain elevation, $d_{i,j,k}$, and to prevail only locally. The head at the model node, $h_{i,j,k}$, is actually an average value for the cell and is assumed to prevail at some distance from the drain. The head losses between the $h_{i,j,k}$ and $d_{i,j,k}$ are assumed to be caused by the three flow processes: the convergent flow towards the drain, flow through material of different conductivity immediately around the drain, and flow through the wall of the drain. If the total head loss, $h_{i,j,k} - d_{i,j,k}$, may be assumed proportional to the

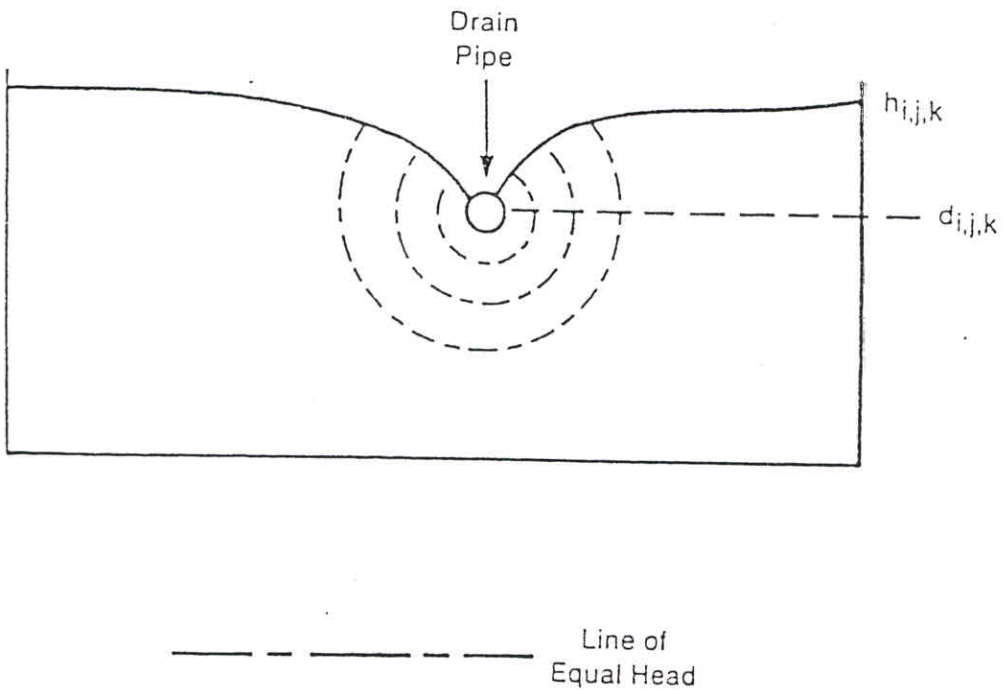


Figure 8: Cross section through cell i,j,k illustrating head loss in convergent flow into drain.

discharge, QD, the drain function can be described by the equation pair:

$$QD_{i,j,k} = 0 \qquad \text{for } h_{i,j,k} \leq d_{i,j,k} \qquad (4)$$

$$QD_{i,j,k} = CD_{i,j,k} * (h_{i,j,k} - d_{i,j,k}) \qquad \text{for } h_{i,j,k} > d_{i,j,k} \qquad (5)$$

The coefficient, $CD_{i,j,k}$, is a lumped (or equivalent) conductance describing all the head losses mentioned above. Attempts could be made to calculate values of CD by developing approximate equations for conductance for the three flow processes, and then calculate the equivalent series of conductance. In practice, it is more common to calculate CD from measured value of QD and h-d using Equation 5. If h-d is not accurately known, CD is usually adjusted during the model calibration in order to match measured values of QD to model calculated values.

1.4.7. Evapotranspiration Package (ET)

The evapotranspiration package (ET) simulates the effects of plant transpiration and the direct evaporation in removing water from the saturated groundwater. The simulation is based on the following equations:

$$Q_{ETi,j} = R_{ETMi,j} * AREA_{i,j} \qquad \text{if } h_{i,j,k} > h_{si,j} \qquad (6)$$

$$Q_{ET} = 0 \qquad \text{if } h_{i,j,k} < (h_{si,j} - d_{i,j}) \qquad (7)$$

$$Q_{ET} = R_{ETM} * AREA_{i,j}(h_{i,j,k} - (h_{si,j} - d_{i,j}))/d_{i,j} \qquad \text{if } (h_{si,j}-d_{i,j}) \leq h_{i,j,k} \leq h_{si,j} \qquad (8)$$

Where,

$Q_{ETi,j}$ is the evapotranspiration in volume of water per unit time[$L^3 T^{-1}$].

$R_{ETMi,j}$ is the maximum rate of evapotranspiration in volume of water per unit area per unit time [$L^3 L^{-2} T^{-1}$].

$AREA_{i,j}$ is the surface area of the cell [L^2].

$h_{i,j,k}$ is the head, or the water table elevation in the cell (i,j) [L]

$h_{si,j}$ is the water table elevation at which the evapotranspiration loss reaches the maximum value. It is taken as the average land surface elevation :

$d_{i,j}$ is the extinction depth. When the distance between $h_{si,j}$ and $h_{i,j,k}$ exceeds $d_{i,j}$, evapotranspiration ceases [L].

Evapotranspiration is applied to only one cell in each vertical column beneath the map area. The user designates the cell using one of the two options:

i. evapotranspiration is always drawn from the uppermost layer of the model. No evapotranspiration occurs in no-flow or constant-head cells.

ii. The cell, from which the evapotranspiration is to be taken, is specified by the user in a two-dimensional array.

1.4.8. General Head Boundary Package (GHB)

The General Head Boundary (GHB) Package simulates flow into or out of a cell i,j,k from an external source. The flow into a cell is assumed proportional to the difference between the head in the cell and the head assigned to the external source, i.e.

$$Q_{bi,j,k} = C_{bi,j,k} * (h_{bi,j,k} - h_{i,j,k}) \quad (9)$$

Where,

$Q_{b,i,j,k}$ is the flow into cell i,j, k from the source.[$L^3 T^{-1}$]

$C_{b,i,j,k}$ is the conductance between the external source and the cell i,j,k.[$L^2 T^{-1}$]

$h_{bi,j,k}$ is the head assigned to the external source [L]

$h_{i,j,k}$ is the head in the cell [L].

The relationship between cell i,j,k and the external source is shown schematically in Figure 9. Since flow into or out of the cell continues to increase without limit as the head difference between the cell and the source increases (Equation 8), one should be careful in using the GHB package.

1.4.9. Solution technique packages

i. The Strongly implicit procedure (SIP) is a method of solving a large system of simultaneous linear equations by iteration. The SIP package is designed to solve the system of finite-difference equations formed by previous packages.

ii. The Slice-successive overrelaxation (SSOR) technique is implemented by dividing the finite difference grid into vertical 'slices' and grouping the node equations into discrete sets, each corresponding to a slice. In every iteration, these sets of equations are processed in turn, resulting in a new set of estimated head values for each slice.

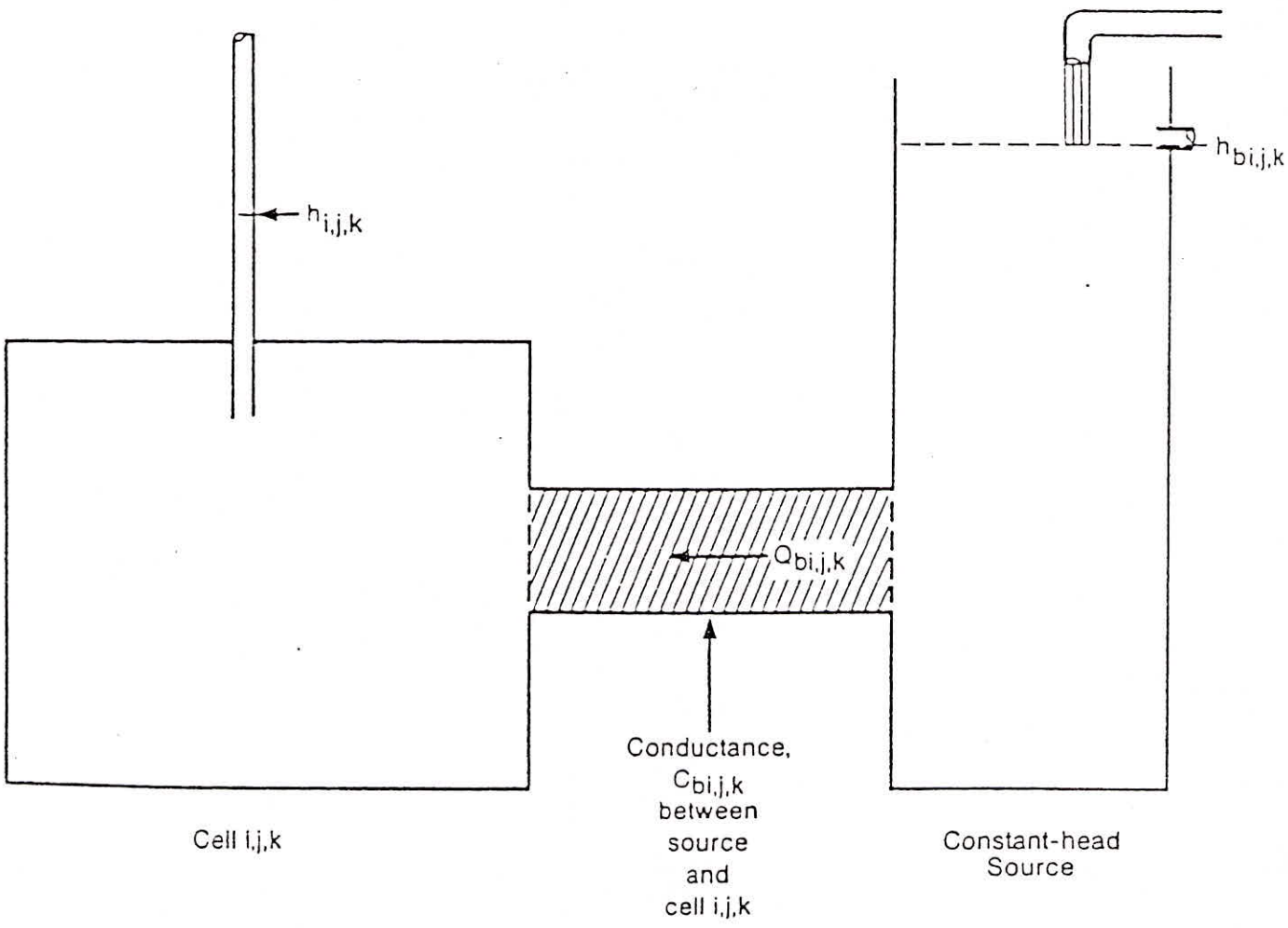


Figure 9: Schematic diagram illustrating principle of general-head boundary package.