Documentation of a Conduit Flow Process (CFP) for MODFLOW-2005

By W. Barclay Shoemaker, Eve L. Kuniansky, Steffen Birk, Sebastian Bauer, and Eric D. Swain

A product of the Ground-Water Resources Program

Techniques and Methods, Book 6, Chapter A24

U.S. Department of the Interior
U.S. Geological Survey

U.S. Department of the Interior
DIRK KEMPTHORNE, Secretary

U.S. Geological Survey
Mark D. Myers, Director

U.S. Geological Survey, Reston, Virginia: 2007

For product and ordering information:
World Wide Web: http://www.usgs.gov/pubprod
Telephone: 1-888-ASK-USGS

For more information on the USGS—the Federal source for science about the Earth, its natural and living resources, natural hazards, and the environment:
World Wide Web: http://www.usgs.gov
Telephone: 1-888-ASK-USGS

Suggested citation:
Shoemaker, W.B., Kuniansky, E.L., Birk, S., Bauer, S., and Swain, E.D., 2007, Documentation of a Conduit Flow Process (CFP) for MODFLOW-2005: U.S. Geological Survey Techniques and Methods, Book 6, Chapter A24, 50 p.

Preface

This report is intended as a reference manual to explain: (1) selected theoretical principles that govern laminar and turbulent ground-water flow, and (2) how these principles were integrated into MODFLOW-2005 to create the Conduit Flow Process (CFP). Users are advised to read each chapter in this report. Users interested in the motivation and theoretical principles represented in the CFP should read chapters 1 and 2. Chapter 3 documents how these theoretical principles were converted into subroutines and finite-difference approximations for integration into MODFLOW-2005. Chapters 4, 5, and 6 are most relevant for users interested in applying the CFP. Specifically, chapter 4 documents the input instructions required for CFP simulations, chapter 5 provides guidance on assignment of values for parameters required for CFP simulations, and chapter 6 presents an example problem that demonstrates all of the CFP functionality.

The U.S. Geological Survey (USGS) has tested the accuracy of the CFP by designing and running many test problems. Despite our best efforts, errors may still exist. If users identify or suspect errors, please contact the USGS. Distribution of the CFP does not constitute any warranty by the USGS. Furthermore, no responsibility is assumed by the USGS for use or misuse of the CFP computer program. The CFP computer program can be obtained at the Internet address *http://water.usgs.gov/software/ground_water.html*. Updates may be made to both the CFP report and computer program, and users can download updates at the Internet address.

Contents

Figures

Tables

Conversion Factors, Datum, and Acronyms

Multiply	By	To obtain
	Length	
millimeter (mm)	0.03937	inch (in.)
meter (m)	3.281	foot (ft)
meter (m)	1.094	yard (yd)
	Area	
square meter (m^2)	10.76	square foot (ft^2)
square kilometer (km^2)	0.3861	square mile (mi^2)
	Flow rate	
meter per second (m/s)	3.281	foot per second (ft/s)
meter per day (m/d)	3.281	foot per day (ft/d)
cubic meter per day (m^3/d)	35.31	cubic foot per day (ft^3/d)
	Pressure	
pascal per second (Pa)	1,000	centipoise (cP)
	Density	
kilogram per cubic meter (kg/m^3)	0.06243	pound per cubic foot (lb/ft^3)
	Transmissivity	
meter squared per second (m^2/s)	10.76	foot squared per second (ft^2/s)

Temperature in degrees Celsius (°C) may be converted to degrees Fahrenheit (°F) as follows:

$$°F=(1.8\times°C)+32$$

ACRONYMS USED IN REPORT

BCF	Block Centered Flow Package
CAVE	Conduit Aquifer Void Evolution
CFP	Conduit Flow Process
CFPM1	Conduit Flow Process Mode 1
CFPM2	Conduit Flow Process Mode 2
CFPM3	Conduit Flow Process Mode 3
COC	Conduit Output Control File
CRCH	Conduit Recharge Package
HUF	Hydrogeologic Unit Flow Package
LPF	Layer Property Flow Package
MODFLOW	Modular Finite-Difference Ground-Water Flow Model
RCH	Recharge Package
USGS	U.S. Geological Survey

Documentation of a Conduit Flow Process (CFP) for MODFLOW-2005

By W. Barclay Shoemaker, Eve L. Kuniansky, Steffen Birk, Sebastian Bauer, and Eric D. Swain

Abstract

This report documents the Conduit Flow Process (CFP) for the modular finite-difference ground-water flow model, MODFLOW-2005. The CFP has the ability to simulate turbulent ground-water flow conditions by: (1) coupling the traditional ground-water flow equation with formulations for a discrete network of cylindrical pipes (Mode 1), (2) inserting a high-conductivity flow layer that can switch between laminar and turbulent flow (Mode 2), or (3) simultaneously coupling a discrete pipe network while inserting a high-conductivity flow layer that can switch between laminar and turbulent flow (Mode 3). Conduit flow pipes (Mode 1) may represent dissolution or biological burrowing features in carbonate aquifers, voids in fractured rock, and (or) lava tubes in basaltic aquifers and can be fully or partially saturated under laminar or turbulent flow conditions. Preferential flow layers (Mode 2) may represent: (1) a porous media where turbulent flow is suspected to occur under the observed hydraulic gradients; (2) a single secondary porosity subsurface feature, such as a well-defined laterally extensive underground cave; or (3) a horizontal preferential flow layer consisting of many interconnected voids. In this second case, the input data are effective parameters, such as a very high hydraulic conductivity, representing multiple features.

Data preparation is more complex for CFP Mode 1 (CFPM1) than for CFP Mode 2 (CFPM2). Specifically for CFPM1, conduit pipe locations, lengths, diameters, tortuosity, internal roughness, critical Reynolds numbers (N_{Re}), and exchange conductances are required. CFPM1, however, solves the pipe network equations in a matrix that is independent of the porous media equation matrix, which may mitigate numerical instability associated with solution of dual flow components within the same matrix. CFPM2 requires less hydraulic information and knowledge about the specific location and hydraulic properties of conduits, and turbulent flow is approximated by modifying horizontal conductances assembled by the Block-Centered Flow (BCF), Layer-Property Flow (LPF), or Hydrogeologic-Unit Flow Packages (HUF) of MODFLOW-2005.

For both conduit flow pipes (CFPM1) and preferential flow layers (CFPM2), critical Reynolds numbers are used to determine if flow is laminar or turbulent. Due to conservation of momentum, flow in a laminar state tends to remain laminar and flow in a turbulent state tends to remain turbulent. This delayed transition between laminar and turbulent flow is introduced in the CFP, which provides an additional benefit of facilitating convergence of the computer algorithm during iterations of transient simulations. Specifically, the user can specify a higher critical Reynolds number to determine when laminar flow within a pipe converts to turbulent flow, and a lower critical Reynolds number for determining when a pipe with turbulent flow switches to laminar flow. With CFPM1, the Hagen-Poiseuille equation is used for laminar flow conditions and the Darcy-Weisbach equation is applied to turbulent flow conditions. With CFPM2, turbulent flow is approximated by reducing the laminar hydraulic conductivity by a nonlinear function of the Reynolds number, once the critical head difference is exceeded. This adjustment approximates the reductions in mean velocity under turbulent ground-water flow conditions.

Chapter 1. Introduction

Rapid laminar or turbulent ground-water flow can occur in dual porosity aquifers through relatively large interconnected voids. Dual porosity aquifers are those that consist of both a primary porosity, which is due to the soil or rock matrix, and a secondary porosity, which may be due to secondary solution and (or) regional fracturing (Freeze and Cherry, 1979). Within the relatively large voids, frictional and surface-tension forces between the fluid and rock are less significant because the conduit diameters are so large that wall roughness only restricts flow moving close to the conduit wall. In fluid mechanics, the term “hydraulic radius” is defined as the channel cross-sectional area divided by the perimeter of the channel cross-sectional area that is wet (Chin, 2000). As the radius of a pore increases, the hydraulic radius also increases and greater flow will occur under the same energy forces due to less resistance to flow from the pore walls.

Laminar flow in dual porosity aquifers is a flow regime in which water moves along parallel streamlines or layers, and shear stresses within the water are overcome by the viscous forces of the water. Turbulent flow is characterized by streamlines that are no longer parallel but flow in random complex patterns (eddies) with large variations in flow velocity about the mean flow velocity. Ground-water flow in porous media tends to remain at low velocity and laminar under most natural hydraulic gradients. When flow becomes turbulent in both porous media and conduits, some of the energy that drives the flow is lost to movement of water in eddies, and specific discharge no longer increases as rapidly as the head gradient increases.

Dual porosity aquifers are often associated with carbonate aquifer systems. Karst carbonate aquifers underlie a land area covering about 10 to 20 percent of the earth's surface, supplying about 25 percent of the world's population with drinking water (Ford and Williams, 1989). Dual porosity aquifers are vulnerable to contamination due to generally short travel times. Management and protection of dual porosity aquifers require an adequate hydrogeologic characterization of the complex geometry of a subsurface network of fissures and voids responsible for concentrated ground-water flow and rapid transport of pollutants toward zones of discharge. A mathematical formulation with a physically based representation of dual porosity systems is necessary for developing reliable water-management strategies in karst settings.

The Conduit Flow Process (CFP) was developed in response to a need for a computer program that accounts for the dual porosity nature of many aquifers. There also was a desire to provide compatibility with recent advancements to the U.S. Geological Survey (USGS) modular ground-water model (MODFLOW). Many research computer programs are available for simulating dual porosity aquifers, but have not been fully documented for wider use (for example, Clemens and others, 1996; Kiraly, 1998; Ewen and others, 2000; Adams and Parkin, 2002; Bauer, 2002; Birk, 2002). Additionally, the structure of MODFLOW has changed with MODFLOW-2000 (Harbaugh and others, 2000) and MODFLOW-2005 (Harbaugh, 2005), making the ground-water flow computer code even more modular and allowing easier addition of Processes to the code.

The CFP was designed to be flexible enough for use in locations with limited or abundant field data. In some geologic environments, such as Mammoth Cave, Kentucky, detailed information is available (or could be derived) on the location, diameter, tortuosity, and roughness of the subsurface caverns. CFP Mode 1 (CFPM1) was designed with these locations in mind. In other locations, such as the Biscayne aquifer of southern Florida, void connections and distributions are so complicated within preferential flow layers that a complete characterization is not possible. CFP Mode 2 (CFPM2) was designed with these locations in mind; specifically, laminar and turbulent flow through complicated void connections is represented with a limited number of "effective" or "bulk" layer parameters.

The CFP is an outgrowth of long-term research into speleogenesis in karst systems, including the Conduit Aquifer Void Evolution (CAVE) code developed by Clemens (1998) and Hückinghaus (1998). Subroutines of a version of the CAVE code described in Bauer (2002) and Birk (2002) were extracted and modified to work with MODFLOW-2005 to partially create the CFP. The CFP expanded upon the CAVE work by: (1) integrating new equations that handle partially saturated pipe flow; (2) working with traditional MODFLOW units other than meters and seconds; (3) including user-defined, rather than constant values, for critical Reynolds numbers (N_{Re}) that transition ground-water flow from laminar to turbulent; and (4) creating preferential flow layers (CFPM2) that also transition between laminar and turbulent flow conditions, while requiring less input data than conduit flow pipes. A final advancement is the creation of CFP Mode 3 (CFPM3), which simultaneously simulates conduit flow pipes (CFPM1) and preferential flow layers (CFPM2).

1.1. Purpose and Scope

This report documents the CFP, which was designed as a Process for MODFLOW-2005. The CFP simulates dual porosity aquifers that can be mathematically approximated by coupling the traditional ground-water flow equation with a discrete network of cylindrical pipes (CFPM1), and (or) inserting a preferential flow layer that uses a turbulent hydraulic conductivity to simulate turbulent horizontal flow (CFPM2) conditions. The pipes can represent dissolution features or fractures and can be fully saturated or partially saturated under laminar or turbulent conditions. The preferential flow layers may represent: (1) a porous media in which turbulent flow may occur under observed hydraulic gradients, or (2) horizontal preferential flow zones within an aquifer for which the explicit geometry of the secondary porosity is not well defined. The CFP simulates steady-state and transient hydraulics of the dual porosity system. Transport and chemical reactions are not within the scope of this version of the CFP.

1.2. Previous Studies

The evolution of karst aquifers and cavern formation and the conceptual models of karst systems are beyond the scope of this report, but are well covered by White (1969; 1993; and 2003), Wolfe and others (1997), Klimchouk and others (2000), and Ford (2003). Ground-water age dating at 12 large karst springs in Florida was conducted by Katz (2004), who found ground-water mean transit times of 5 to 40 years that reflect the slower movement of solutes through small openings in the aquifer matrix throughout the spring contributing area. Estimation of land-surface areas contributing recharge to four large karst springs in north-central Florida is described by Shoemaker and others (2004), who found composite areas contributing recharge that extend from about 310 to 1,890 km^2.

Single porosity models based on the ground-water flow equation, called equivalent porous media models or single continuum models, have been successfully applied to dual porosity systems to simulate water balances. Single porosity models are limited, however, in their ability to simulate transport processes (Thrailkill, 1986; Kuniansky and Holigan, 1994; Teutsch and Sauter, 1998; Kuniansky and others, 2001; Svenson, 2001; Sepúlveda, 2002; Scanlon and others, 2003). The ground-water flow equation solved by single porosity models is the continuity equation for flow, with Darcy's law (Darcy, 1856) incorporated for porous media with assumptions that water is incompressible and of constant density and viscosity (Raudkivi and Callander, 1976, p. 43; Bouwer, 1978, p. 202; Bear, 1979, p. 93). This equation is valid for ground-water flow problems when the velocity of ground water is slow and laminar, and pores are small (less than about 10 mm). In fluid mechanics, this equation is sometimes referred to as a potential flow equation, in which flow is defined by fluid pressure and potential energy, and the kinetic energy that arises from the ground-water velocity is assumed to be negligible.

Several types of deterministic numerical models, also called distributed parameter models, have been applied to dual porosity systems (fig. 1). The simplest approach is the equivalent porous media approach (sometimes called the single continuum approach, heterogeneous continuum approach, distributed parameter approach, or smeared conduit approach) (fig. 1A). Another approach is the linking of two flow regimes with an exchange term, called the dual continuum approach (fig. 1B). With this approach there are two flow regimes for each aquifer. One regime represents the rock matrix, which has a small hydraulic conductivity and a large porosity (diffuse flow regime); the other regime has a large hydraulic conductivity and small porosity to represent the higher velocity flow (conduit or pipe flow). With the dual continuum approach, both continuums are linked by a flux exchange term, and heads in each continuum are iteratively solved until there is head convergence. Three additional approaches are a discrete fracture network (fig. 1C), a variation on fracture network simulations in which multiple fracture networks are coupled (fig. 1D), and the coupling of a single continuum model with a discrete pipe network (fig. 1E).

Each approach has advantages and disadvantages. A single continuum approach is the simplest to apply. Generally, this approach can be used for regional flow problems if the investigation scale is much greater than the scale of the heterogeneities, and for water-resources investigations in which the models can be calibrated to flow and head information and only water budgets are desired. The discrete fracture approach can be used if transient solute-transport responses are desired for a system dominated by conduits. Knowledge of the fracture network geometry, however, is required if this approach is used. If the geometry is not available, sometimes a dual continuum model can be applied (Teutsch and Sauter, 1998). The advantage of the dual continuum approach is that the detailed geometry of the conduits is not required (Teutsch, 1993; Sauter, 1993; Lang, 1995). The dual continuum model, however, is not always capable of simulating transport processes on a small scale (Mohrlok, 1996). The disadvantages of discrete multiple fracture networks are the requirement of detailed knowledge of a fracture network at multiple scales and the application of computationally intensive codes that have long computer simulation time and memory requirements (Lang, 1995). The coupling of a single continuum model with a discrete conduit network model allows the integration of detailed information about the conduits in areas where the geometry of the conduits may be known, providing a more physically representative model in these areas.

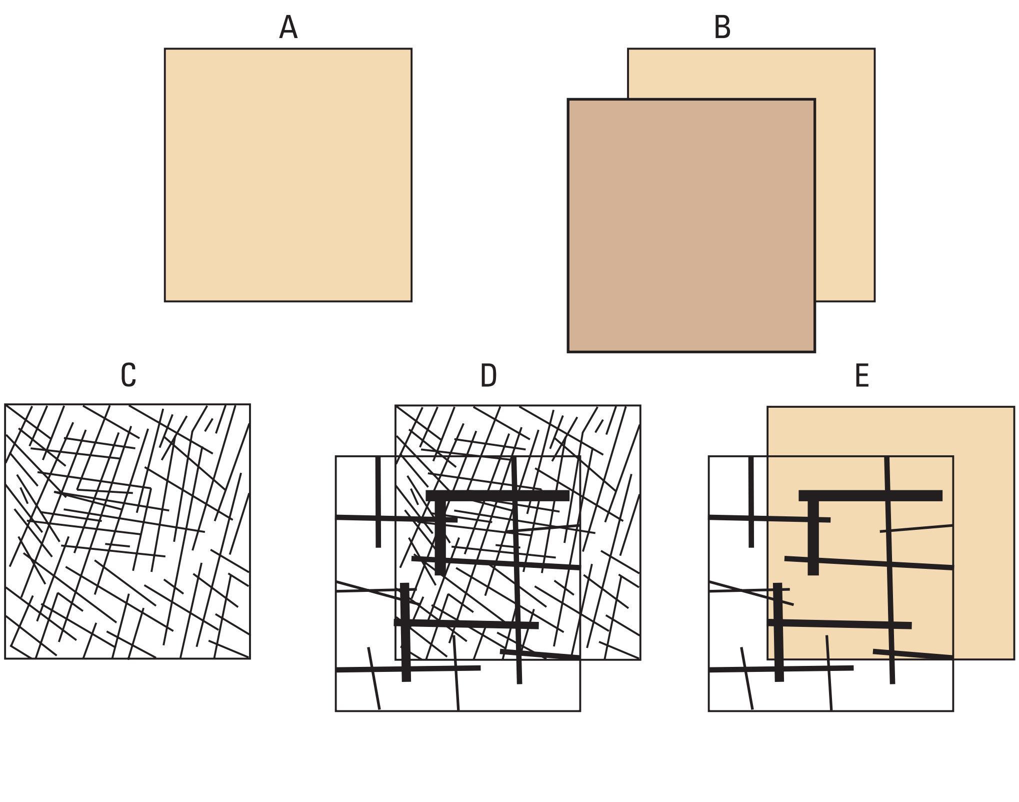


Figure 1. Approaches to karst modeling *(A)* single continuum, *(B)* double continuum, *(C)* discrete fractures, *(D)* discrete multiple fracture networks, and *(E)* discrete conduit coupled to single continuum (Modified from Bauer, 2002; fig. 2.4).

Simulations of advective transport are frequently performed with equivalent porous media models of a dual porosity system using estimates of effective porosity. Effective porosity is defined as the percentage of interconnected void space per unit volume of aquifer. By using field tracer test data or geochemical isotope age data in dual porosity systems, one can use the equivalent porous media approach to perform transport simulations by finding an effective porosity that will yield the average time of travel to match the tracer test or geochemical breakthrough data. Because pore velocity is inversely proportional to effective porosity, a very small (less than 5 percent) effective porosity value is often required to match field measured times of travel when, in actuality, the percentage by volume of connected pore space could be more than 40 percent (Knochenmus and Robinson, 1996; Kuniansky and others, 2001; Renken and others, 2005). In these cases, the transport parameters used with the equivalent porous media model are not authentic, and these models could be limited in their application to transport processes. This lack of authenticity is due, in part, to equivalent porous media models that assume fluid flow is uniformly distributed over the thickness of many carbonate aquifers, when in fact, a very small percentage of the total aquifer thickness explains most of the ground-water flow.

1.3. Acknowledgments

Georg Teutsch, Martin Sauter, Rudolf Liedl, and graduate students from the University of Tubingen, Germany, are acknowledged for developing the CAVE computer program. Subroutines from CAVE were extracted, modified, and integrated into MODFLOW-2005 to partly create the CFP computer program. Within the USGS, Arlen Harbaugh, Thomas Reilly, and Paul Barlow provided technical support, including suggestions for assuring that CFP preserved MODFLOW modularly. The CFP code development and documentation was funded by the USGS Ground-Water Resources Program. Additionally, the basis and need for developing CFPM2 were a result of the work on the Biscayne aquifer of southern Florida by Kevin J. Cunningham, Steve Krupa, Michael A. Wacker, Robert A. Renken, Michael Zygnerski, and Joanne Dixon, USGS. Keith J. Halford, USGS, provided very useful discussions and feedback on the mathematical implementation of turbulent flow in a preferential flow layer (CFPM2). R. Steven Regan, USGS, is acknowledged for reviewing and revising FORTRAN programming. Andrew Long, Alan Shapiro, Christy Crandall, Thomas Reilly, Paul Barlow, and Robert Renken, USGS, provided useful colleague review comments, which enhanced the organization and readability of the final documentation. Melissa Hill, Southwest Florida Water Management District, and Thomas Reimann, Institute for Groundwater Management, Germany, are gratefully acknowledged for beta testing the CFP computer program.

Chapter 2. Conduit Flow Process (CFP) Methodology

Theoretical concepts and limitations that are the basis for development of the CFP are briefly described. These concepts include Reynolds numbers (R_e), the limitations of Darcy's law for porous media ground-water flow, and methods for coupling a discrete pipe network to MODFLOW-2005 (CFPM1). Also presented is the mathematical basis for computing horizontal turbulent flow in preferential flow layers (CFPM2). Finally, the concept of upper and lower critical Reynolds numbers (N_{Re}) for transient laminar and turbulent ground-water flow conditions is described as implemented for transient CFP simulations.

2.1. Reynolds Numbers and Limitations of Darcy's Law for Porous Media

Darcy's law was empirically derived. The laboratory studies by Darcy (1856) for one-dimensional laminar flow through porous media along a known cross-sectional area, A [L^2], indicated a constant of proportionality between specific discharge (also called specific velocity, Darcy velocity, or mean velocity), q [LT^{-1}], and the hydraulic gradient, $\frac{dh}{dl}$ [dimensionless], for water flow through small pores (less than 10 mm). This constant of proportionality is called hydraulic conductivity, K [LT^{-1}]:

$$Q = -KA\frac{dh}{dl} \Rightarrow K = -\frac{Q}{A}\frac{\Delta l}{\Delta h} = -q\frac{\Delta l}{\Delta h}, \tag{1}$$

where

Δh [L] is the measured head difference,
Δl [L] is the length over which the head difference is measured,

and

Q [L^3T^{-1}] is volumetric discharge.

Darcy's law is valid only for laminar flow conditions (fig. 2). Reynolds numbers, R_e, are dimensionless numbers used in fluid mechanics to indicate whether flow is laminar or turbulent. The R_e represents the ratio of fluid inertial to viscous forces. The critical Reynolds number, N_{Re}, is the value of R_e above which flow becomes turbulent and below which flow is laminar. The R_e is computed and defined from the equation:

$$R_e = \frac{Vd_{pore}\rho}{\mu} = \frac{Vd_{pore}}{\nu}, \tag{2}$$

where

V is the mean flow velocity [LT^{-1}] across a cross-sectional area (also called Darcy velocity) and is equal to the volumetric flow, Q [L^3T^{-1}], divided by the cross-sectional area perpendicular to the direction of flow, A [L^2];

d_{pore} is the mean pore size diameter [L];

ρ is the density of the water [ML^{-3}];

μ is the absolute or dynamic viscosity of water [$ML^{-1}T^{-1}$];

and

ν is the kinematic viscosity of water [L^2T^{-1}].

The Darcy velocity (also called mean velocity or specific velocity), not the pore velocity (which is equal to the Darcy velocity divided by effective porosity), is used for R_e calculations. The pore velocity is needed for transport simulations, whereas the Darcy velocity is used in textbooks and laboratory experiments to define the N_{Re} at which flow switches from laminar to turbulent. Porous media flow tends to remain at low velocity and laminar under many natural gradients because of the small pore diameters and the effect of surface tensional forces between the water and rock. For various porous media, the N_{Re} at which flow switches from laminar to turbulent ranges from 1 to 60, and is dependent on the smoothness of the grains, tortuosity of the connected pore spaces, average pore diameter, temperature, and other properties of the media. When flow becomes turbulent, some of the flow energy is lost by the movement of water in eddies, and specific discharge does not increase as rapidly as head gradients increase. Thus, in turbulent conditions, flow is no longer a linear function of head gradients (fig. 2), as in equation 1, thereby limiting the applicability of Darcy's law. There also may be limitations to the applicability of Darcy's law for very low permeability media, but this has not been thoroughly tested (Ingebritsen and Sanford, 1998).

It is not possible to know exact field distributions of N_{Re} for a specific aquifer or porous media. The N_{Re} can be approximated through laboratory experiments. Many researchers have measured specific discharge under different gradients and fit a line through these values to back calculate the N_{Re} at which specific discharge becomes a nonlinear function of the gradient (hydraulic conductivity would no longer be a constant). The larger the pore diameter and smoother the grain surfaces, such as might occur in a well sorted gravel point bar deposit, the higher the N_{Re}. Additionally, as hydraulic conductivity increases (generally because of larger pore spaces), the N_{Re} increases. For porous media, representative lengths have been used for d_{pore} to calculate R_e, such as a representative grain size (frequently d_{50}, the diameter of the sieve size at which 50 percent of the grains pass in a standardized sieve analysis performed in a geotechnical analysis). Many textbooks

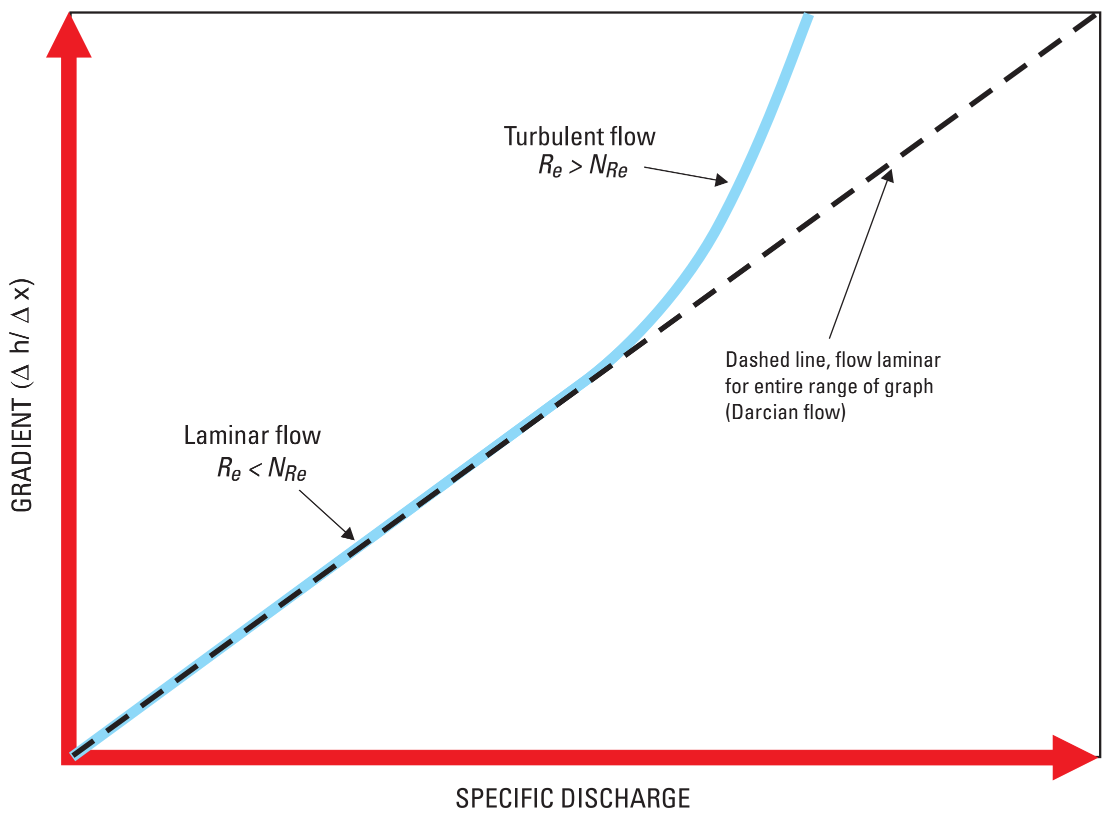


Figure 2. Specific discharge relative to hydraulic gradient (Modified from Bear, 1979; Freeze and Cherry, 1979). R_e is the Reynolds number, and N_{Re} is the critical Reynolds number.

show the range of N_{Re} for porous media from 1 to 10 (Bear, 1979; Freeze and Cherry, 1979). Schneebeli (1955) used glass spheres and found flow became turbulent at a range of R_e from 5 to 60.

2.2. Coupling a Discrete Pipe Network with a Laminar Flow Model (CFPM1)

The main objective of CFPM1 is to couple a laminar flow model (MODFLOW-2005) with a discrete pipe network to approximate the dual porosity nature of many flow systems. The laminar flow model code (MODFLOW) is well documented in Harbaugh and McDonald (1996), McDonald and Harbaugh (1988), Harbaugh and others (2000), and Harbaugh (2005). Laminar and turbulent flow within fully and partially saturated pipes and open channels is discussed in more detail in Rouse (1950), Daily and Harleman (1966), Henderson (1966), Vennard and Street (1975), and Chin (2000). This section briefly describes theoretical equations for: (1) the laminar flow model, (2) pipe flow, (3) correcting flow in partially filled pipes, (4) computing ground-water exchange between MODFLOW and the discrete pipe network, and (5) correcting ground-water exchange between MODFLOW and partially filled pipes.

2.2.1. Laminar Flow Model

The laminar flow model, MODFLOW-2005 (Harbaugh, 2005), solves the ground-water flow equation using a block-centered finite-difference approximation. The partial differential equation for three-dimensional ground-water flow is:

$$\frac{\partial}{\partial x}\left(K_{xx}\frac{\partial h}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_{yy}\frac{\partial h}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_{zz}\frac{\partial h}{\partial z}\right)\pm W = S_s\left(\frac{\partial h}{\partial t}\right), \quad (3)$$

where

K_{xx}, K_{yy}, and K_{zz} are values for hydraulic conductivity [LT^{-1}] along the x, y, and z axes, respectively;
h [L] is the potentiometric head;
W [T^{-1}] is a volumetric flux per unit volume;
S_s [L^{-1}] is specific storage;

and

t [T] is time (McDonald and Harbaugh, 1988).

This equation assumes that ground water is incompressible with constant density and viscosity, and that flow is laminar (Darcian). Hydraulic conductivity (K_{xx}, K_{yy}, and K_{zz}) may vary in space depending on the heterogeneity of the porous media in the geologic formations forming the aquifers or confining units. Finally, MODFLOW-2005 has numerous packages to provide linear and nonlinear variations of specified head, specified flux, and head-dependent flux boundary conditions. The details regarding boundary conditions and solution of the ground-water flow equation are well documented in many MODFLOW manuals, and therefore, are not included in this report.

2.2.2. Pipe Network Geometry

The pipe network consists of many pipes, each having two ends called nodes (fig. 3). A node of a pipe is the location where up to six individual pipes can connect. Each node is located at the center of a finite-difference cell, and there can be only one node within a finite-difference cell. Finite-difference cells can have equal or different row and column lengths. Model layers composed of finite-difference cells also can have constant or variable thicknesses. Pipes can connect diagonally between two finite-difference cells within and between adjacent model layers. Note that the resolution of the model grid should be fine enough for the conduit features to span multiple grid cells. A pipe network, for example, cannot be designed within a single finite-difference cell.

2.2.3. Equations for Pipe Flow

When a pipe is fully saturated, flow within the pipe can be assumed to be one dimensional along the axis of the pipe. Experiments of the late 1850s on flow of water in straight cylindrical pipes indicated that the head loss along the pipe varied “directly with velocity head and pipe length, and inversely with pipe diameter” (Vennard and Street, 1975, p. 380). Darcy, Weisbach, and other researchers in hydraulics proposed the following general equation, which is applicable to both turbulent and laminar flow:

$$\Delta h = h_L = f\frac{\Delta l}{d}\frac{V^2}{2g}, \quad (4)$$

where

Δh or h_L is the head loss [L] measured along the pipe of length Δl [L],
f is the friction factor [unitless],
d is pipe diameter [L],
V is the mean velocity [LT^{-1}],

and

g is the gravitational acceleration constant [LT^{-2}].

The friction factor in equation 4 is a function of R_e of the flow and the relative roughness of the pipe. The relative roughness of a pipe is usually defined using the mean height of the conduit wall micro-topography. Equation 4 is known as the Darcy-Weisbach equation. Because the mean pipe flow velocity, V, is equal to the volumetric flow, Q [L^3T^{-1}], divided by the

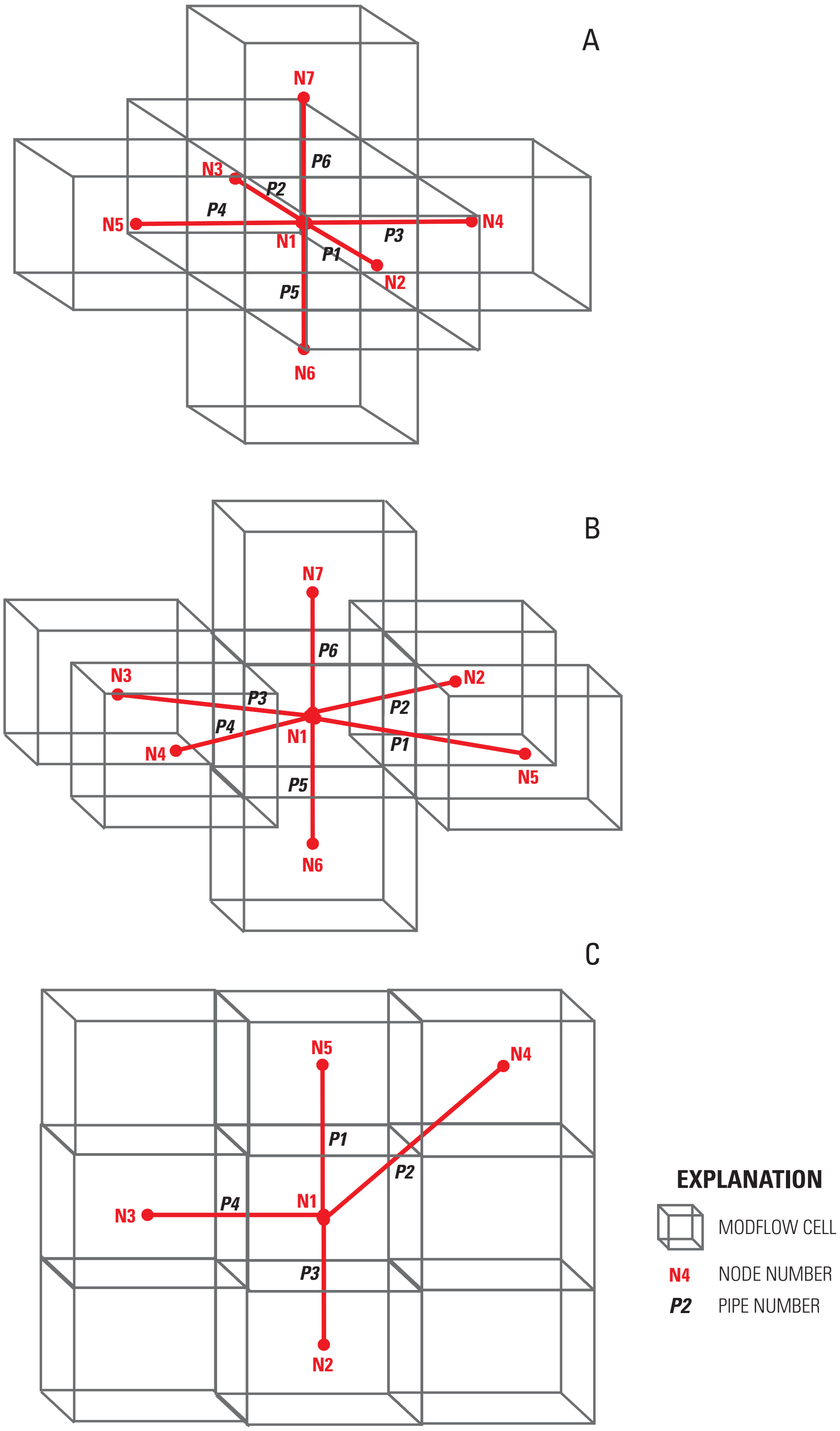


Figure 3. Three possible configurations of conduit flow pipes within MODFLOW cells.

cross-sectional area perpendicular to flow, A [L^2], the Darcy-Weisbach equation can be reformulated to solve for volumetric flow rates, Q [L^{3}T^{-1}], rather than head losses Δh, as follows:

$$Q = A\sqrt{\frac{\Delta h d 2g}{f \Delta l}}, \tag{5}$$

Laminar flow is characterized by a parabolic velocity distribution along a cross section of the circular pipe. Laminar pipe flow can be approximated with the Hagen-Poiseuille equation (Vennard and Street, 1975, p. 390–391). The Hagen-Poiseuille equation was written in terms of the head gradient along the pipe, the viscosity of the fluid, and the pipe diameter squared by Craft and others (1962) and Halford (2000) as:

$$Q = -\frac{\pi d^4 \rho g \Delta h}{128 \mu \Delta l \tau} \quad \text{or} \quad Q = -A\frac{\rho g d^2 \Delta h}{32 \mu \Delta l \tau}, \tag{6}$$

where τ is the tortuosity [unitless] of the pipe, which is defined as the distance actually traveled by a particle flowing through the center of the pipe divided by the straight line distance between the nodes of the pipe. Note that for CFPM1 pipe flow calculations, ρ and μ are computed as a function of the ground-water temperature.

For turbulent flow in a pipe, the friction factor in the Darcy-Weisbach equation was defined empirically through experiments involving smooth and rough pipes. The functional relation between f and R_e used for turbulent flow in CFPM1 is given by the empirical Colebrook-White formula (Colebrook and White, 1937; Dreybrodt, 1988):

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{k_c}{3.71d} + \frac{2.51}{R_e\sqrt{f}}\right), \tag{7}$$

where k_c [L] represents the mean roughness height of the conduit wall micro-topography. The Colebrook-White equation was derived such that it approaches a smooth pipe equation at a low R_e and a rough pipe equation at a high R_e (Morris, 1963). According to Horlacher and Ludecke (1992), the turbulent flow rate through a pipe is given by combining the Darcy-Weisbach equation (equation 5) with the Colebrook-White equation as:

$$Q = -\sqrt{\frac{|\Delta h| g d^5 \pi^2}{2\Delta l \tau}} \log\left(\frac{2.51\nu}{\sqrt{\frac{2|\Delta h| g d^3}{\Delta l \tau}}} + \frac{k_c}{3.71d}\right)\frac{\Delta h}{|\Delta h|}. \tag{8}$$

2.2.4. Corrections for Flow in Partially Filled Pipes

A special case occurs when the conduit pipes are partially filled; for example, when the water elevation in the pipe is less than the top elevation of the pipe. A flow correction was derived based on work by Munson and others (1994), White (1994), and Chin (2000, p. 89), who described flow in noncircular conduits using the hydraulic radius, rather than a conduit diameter, in the formulations for laminar or turbulent flow. According to Munson and others (1994) and White (1994), using the hydraulic radius as a basis for computing frictional head losses in noncircular conduits is usually accurate to within 15 percent for turbulent flow and to within 40 percent for laminar flow. The approach taken here deviates slightly from this prior work by computing an effective diameter, d_e, of a fully saturated pipe with the same hydraulic radius as the partially filled pipe. For partially filled cases, the d_e of a pipe is used in the Hagen-Poiseuille and Darcy-Weisbach equations instead of d to compute laminar or turbulent flow rates.

The d_e of a pipe can be formulated as a function of θ (fig. 4), where θ is a function of data that are readily available; specifically, the hydraulic depth, D[L], and pipe diameter, d[L]. θ is approximated at pipe nodes and is bounded by two straight lines drawn to the center of the pipe from the intersections of the pipe water level with the pipe wall (fig. 4). The D in the pipe is defined as the depth of the flow section; specifically, D is the difference in elevation between the water level in the pipe and bottom of the pipe (fig. 4). From figure 4 it can be seen that:

$$\sin\left(\frac{\theta}{2}\right) = \frac{y}{r}, \tag{9}$$

and:

$$\theta = 2\sin^{-1}\left(\frac{y}{r}\right). \tag{10}$$

Using $x = r - D$ (fig. 4) and $r = d/2$ yields:

$$y = \sqrt{r^2 - x^2} = \sqrt{r^2 - (r-D)^2} = \sqrt{D(d-D)}, \tag{11}$$

and:

$$\theta = 2\sin^{-1}\left(\frac{2\sqrt{D(d-D)}}{d}\right). \tag{12}$$

If the elevation of water in the pipe is greater than the elevation of the center of the pipe, then:

$$\theta_2 = 2\pi - \theta. \tag{13}$$

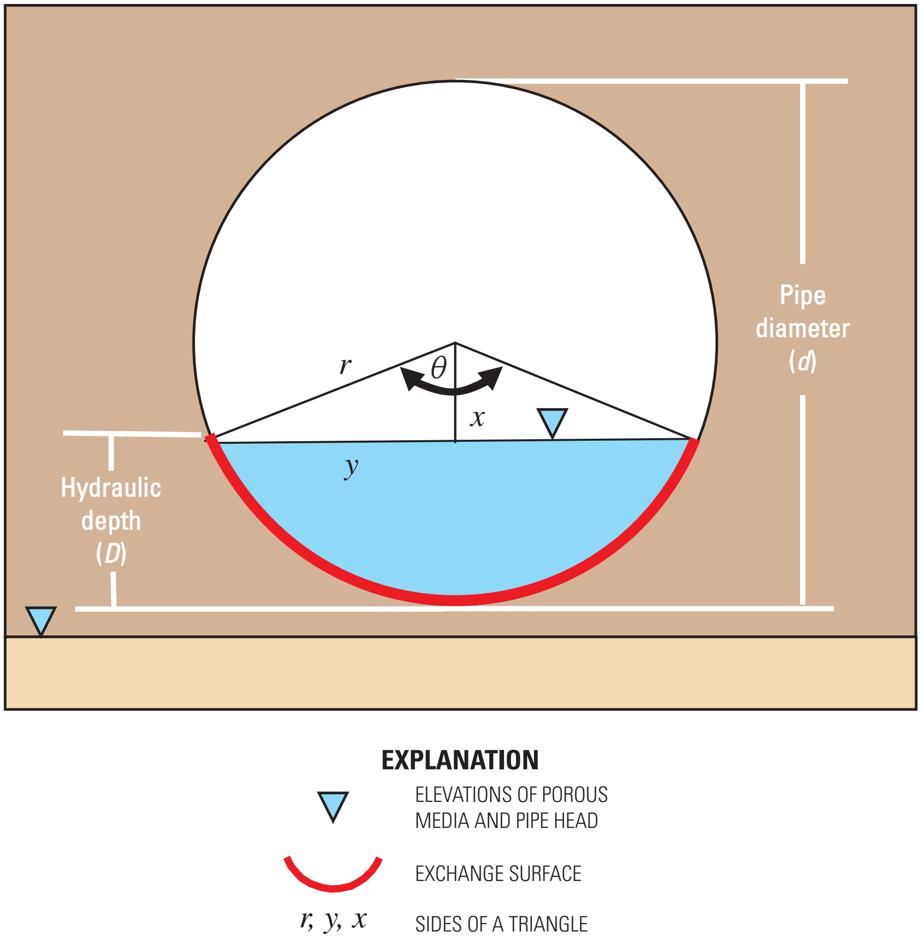


Figure 4. Partially filled pipe and hydraulic properties and variable definition. The symbol θ is approximated at pipe nodes and is bounded by two straight lines drawn to the center of the pipe from the intersections of the pipe water level with the pipe wall.

The d_e can now be formulated as a function of θ or θ_2 using the definition of hydraulic diameter (French, 1985):

$$d_e = 4A / P, \tag{14}$$

where P [L] is the wetted perimeter. For a fully saturated pipe of radius r, d_e is given as:

$$d_e = \frac{4\pi r^2}{2\pi r} = 2r = d. \tag{15}$$

For a partially filled pipe, assuming that the pipe is horizontal, A and P are given by (French, 1985):

$$A = \frac{1}{8}\left(\theta - \sin(\theta)\right) d^{\,2}, \tag{16}$$

and:

$$P = \frac{1}{2}\theta d, \tag{17}$$

which yields:

$$d_e = d\left(1 - \frac{\sin(\theta)}{\theta}\right). \tag{18}$$

If the elevation of water in the pipe is greater than or equal to the elevation of the center of the pipe, then θ_2 from equation 13 is used in equations 16, 17, and 18.

Changes in storage also must be considered in partially filled pipes as the water level in the pipe rises or falls. Storage changes in a pipe are given by the difference of the water filled volume at times t_0 and t_1. Specifically, storage changes, Q_S [L^3T^{-1}], are given by:

$$Q_S = \frac{V_{t1} - V_{t0}}{t_1 - t_0}, \tag{19}$$

where the volume of water, V [L^3], in the pipe is given by:

$$V_t = A \Delta l \tau \tag{20}$$

and:

$$V_t = \frac{1}{8}\left(\theta_t - \sin(\theta_t)\right) d^2 \Delta l \tau \tag{21}$$

A limitation of this approach for simulating flow in partially filled pipes is the assumption that the slope of the bottom elevation of the pipe is zero. In partially filled cases, the slope of the bottom elevation of the pipe also drives flow in addition to the head loss along the pipe. Errors in the flow calculations, therefore, are likely in applications where this assumption is violated; for example, in partially filled caverns or caves with sloping bottom elevations. In contrast, errors in the flow calculations should decrease as the slope of the bottom elevation decreases.

2.2.5. Exchange of Water between the Pipe Network and Laminar Flow Model

The linear equation used by the CFP to compute the exchange of ground water between MODFLOW and pipes is given by:

$$Q_{ex} = \alpha_{j,i,k}(h_n - h_{j,i,k}), \tag{22}$$

where

Q_{ex} [L^3T^{-1}] is the volumetric exchange flow rate,

$\alpha_{j,i,k}$ is the pipe conductance at MODFLOW cell j,i,k [L^2T^{-1}],

h_n is the head [L] at pipe node n located at the center of the MODFLOW cell,

and

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

The pipe conductance is a resistance term that limits ground-water exchange between the pipes and porous media (similar to the conductance terms used in the other head-dependent flow packages of MODFLOW). The CFP assumes ground-water exchange occurs along the entire pipe, and is not turbulent. Negative Q_{ex} indicates that exchange flow is from the porous media into the conduit pipe(s); conversely, positive Q_{ex} indicates that flow is from the conduit pipe(s) into the porous media.

2.2.6. Corrections to Exchange for Partially Filled Pipes

When pipes are either partially filled or dry or when porous media heads lie below the top elevation of the conduit pipe (fig. 5), pipe conductances are automatically limited by CFP. In these instances, the partial surface area of pipe walls, A_p [L^2], that is expected to participate in ground-water exchange is internally computed and used to modify the pipe conductances ($\alpha_{j,i,k}$). For example, consider a case where a conduit pipe is partially filled, with the elevation of the porous media head ($h_{j,i,k}$) less than the elevation of the bottom of the pipe (fig. 5). In this case, flow from the pipe to the underlying aquifer occurs only within the inner submerged area of this pipe wall, A_p, which is computed with the following equation:

$$A_p = P \Delta l \tau, \tag{23}$$

where the perimeter, P, is computed with the following equation (French, 1985, p. 12):

$$P = \frac{1}{2}\theta d \tag{24}$$

Again, if the elevation of water in the pipe equals or exceeds the elevation of the center of the pipe, then θ_2 is used in equation 24.

The hydraulic depth, D, is needed to compute θ. In most partially filled cases, D is taken as the difference in elevation between the water level in the pipe and bottom of the pipe (fig. 4). An exception is the case where the porous media head is less than the top elevation of the pipe, but greater than the water level in the pipe (fig. 5E). In this exception, the D is estimated as the difference in elevation between the porous media head and the bottom of the pipe. This exception may better represent the total partial area actively exchanging ground water. Nine scenarios (fig. 5) are programmed in CFPM1 in which modifications to the pipe conductance could be needed because of partially saturated pipes and (or) porous media heads. Modifications to the pipe conductance also were programmed for a dry pipe with the porous media head greater than the bottom elevation of the pipe (figs. 5C and F), even though this scenario seems unlikely to occur in real ground-water flow systems.

2.3. Computations of Turbulent Flow in Preferential Flow Layers (CFPM2)

While many carbonate aquifers do have mapped conduits that resemble pipes, such as the Woodville karst plain of Florida (Davis, 1996), some carbonate aquifers have units of high vuggy porosity (fig. 6B). In the Biscayne aquifer of southern Florida, for example, much of the ground-water flow is constrained into stratiform layers characterized by

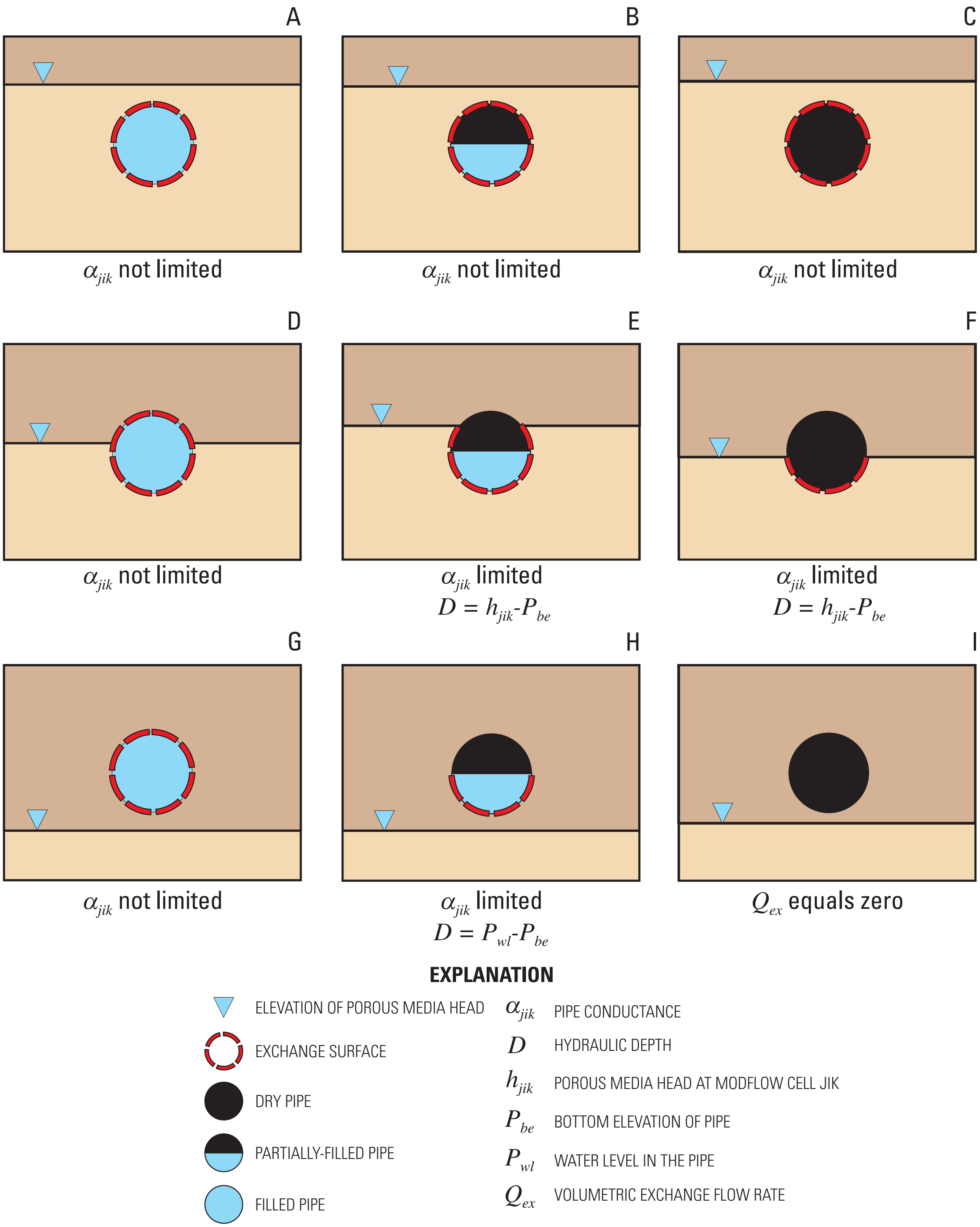


Figure 5. Scenarios where modifications are made to pipe conductances based on pipe and MODFLOW heads.

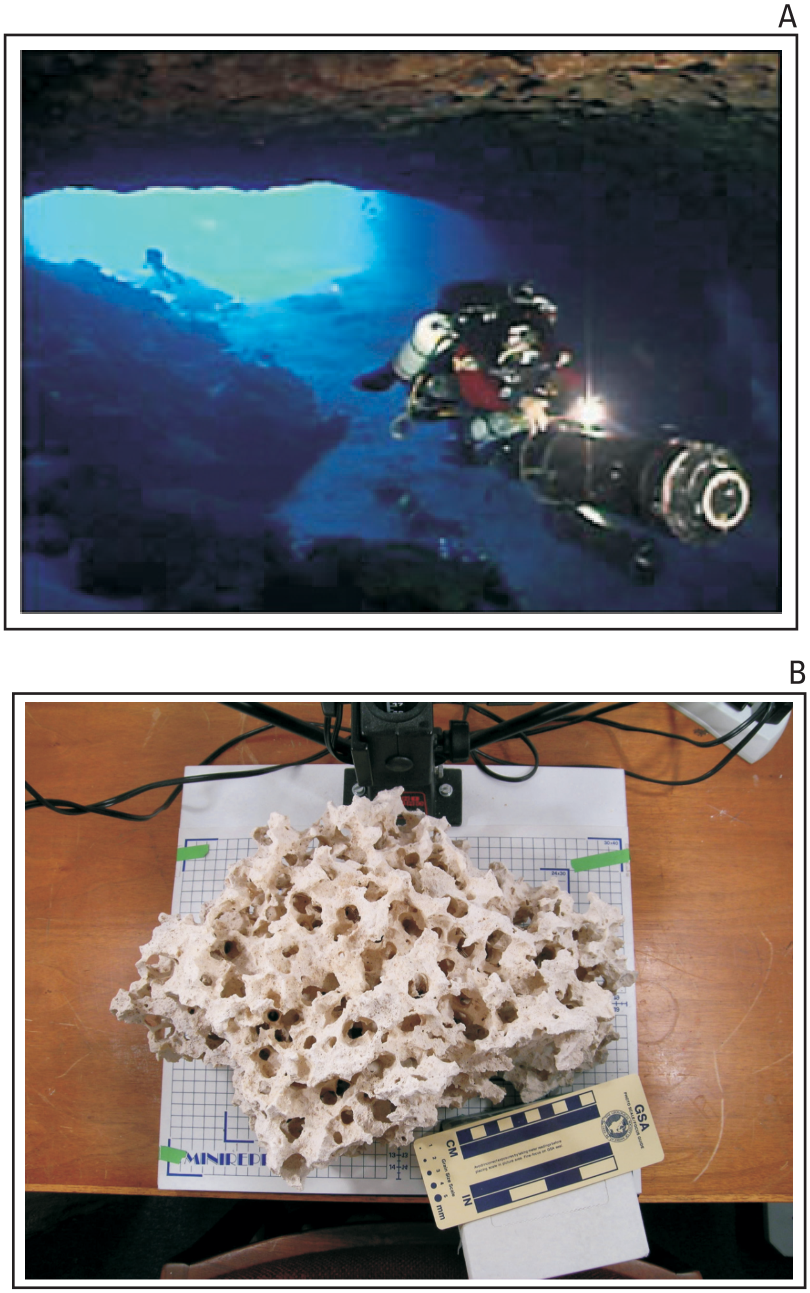


Figure 6. *(A)* Wakulla Springs, and *(B)* sample collected from the Biscayne aquifer in southeastern Florida (From K.J. Cunningham and others, U.S. Geological Survey, written commun., 2006).

touching-vug porosity of paleobiogenetic origin (Cunningham and others, 2006). Although regional internal characterization of these voids is unlikely, the bounding depths to the tops and bottoms of geologic layers that contain these voids are obtainable (Cunningham and others, 2006). The CFPM2 was designed to represent preferential ground-water flow layers with potential turbulent ground-water flow. If CFPM2 is used and flow is turbulent, only flow in the horizontal plane (along rows and columns) differs from standard MODFLOW-2005 computations.

The basis for CFPM2 modifications to horizontal flow is the nonlinearity of the relation between specific discharge and hydraulic gradient when flow is turbulent (fig. 2). Note that CFPM2 does not make use of a pipe network; rather, the existing MODFLOW solvers compute turbulent flow using a turbulent hydraulic conductivity, computed as a power function of the Reynolds number. The approach for calculating a turbulent hydraulic conductivity in the preferential flow layers is similar to the approach for simulating turbulent flow in a well bore developed by Halford (2000). An adjustment factor,

F_{adj} [dimensionless], is used for adjusting the laminar hydraulic conductivity. Halford (2000) sets the adjustment factor equal to the square root of the ratio of the pipe friction factor for laminar flow divided by the pipe friction factor for turbulent flow. CFPM2 deviates from Halford's (2000) approach by making F_{adj} a function of the square root of the ratio of the user specified critical N_{Re} divided by the R_e:

$$K_{turb} = F_{adj} K_{lam} . \tag{25}$$

and:

$$F_{adj} = \sqrt{\frac{N_{\mathrm{Re}}}{R_e}} = \sqrt{\frac{K_{lam} \Delta h_{crit}}{K_{turb} \Delta h}} ,$$

$$\text{when } \Delta h > \Delta h_{crit}\text{; and } F_{adj} = 1\text{, when } \Delta h \le \Delta h_{crit}, \tag{26}$$

where

Δh is the computed head difference [L] between two horizontally adjacent cells,

and

Δh_{crit} is the critical head difference [L].

The Δh_{crit} is computed from the N_{Re} as will be discussed in the next paragraph. The result is turbulent hydraulic conductivities that are increasingly less than the laminar hydraulic conductivity as the head gradient between cells increases, creating a nonlinear relation between specific discharge and hydraulic gradient. As can be seen in the above equation, the turbulent hydraulic conductivity is a function of itself, requiring the use of any of MODFLOW-2005 iterative solvers. The K_{turb} would continue to be adjusted until the calculated change in head between iterations is less than the user specified convergence criteria specified for the solver package.

The Δh_{crit} is dependent upon the user-defined N_{Re}, and the laminar hydraulic conductivity specified, K_{lam}. When Darcy's law is substituted for the mean velocity (defined as discharge divided by the cross-sectional area, Q/A) in the Reynolds number equation:

$$N_{\mathrm{Re}} = \frac{K_{lam} \dfrac{\Delta h_{crit}}{\Delta l} d_{pore}}{v} , \tag{27}$$

By rearranging the terms in equation 27, Δh_{crit} is constant and equal to:

$$\Delta h_{crit} = \frac{N_{\mathrm{Re}} \Delta l v}{K_{lam} d_{pore}} . \tag{28}$$

In summary, when the head difference driving flow is greater than Δh_{crit}, turbulent flow is activated by CFPM2, which reduces laminar hydraulic conductivities into turbulent hydraulic conductivities reproducing the nonlinear relation between specific discharge and hydraulic gradients.

To demonstrate the accuracy of the CFPM2 approach, comparisons were made between turbulent and laminar specific discharges computed using Darcy's law, the Darcy-Weisbach equation, and the CFPM2 approach. Hypothetical head gradients were used for each equation. Under turbulent conditions with CFPM2, laminar hydraulic conductivities were converted into turbulent hydraulic conductivities once the Δh_{crit} was exceeded (fig. 7). The example problem assumed v is equal to 0.000001 m^2/s, d_{pore} is equal to 0.5 m, Δl is equal to 100 m, K_{lam} is equal to 1.157 m/s, and the N_{Re} is equal to 500. Using equation 28, Δh_{crit} is equal to 0.000864 m. Hypothetical head gradients were generated that range from 0.00001 to 0.049 m. The relation between turbulent specific discharge and hydraulic gradient becomes nonlinear when the Δh_{crit} and the N_{Re} are exceeded (fig. 7). In cases where the Δh_{crit} is not exceeded, flow is laminar and the CFPM2 approach computes specific discharge using laminar hydraulic conductivities and Darcy's law, as expected.

There are advantages and limitations to the CFPM2 approach. The limited data requirements for CFPM2 when compared to CFPM1 are a major advantage for CFPM2. Users only need to assign upper and lower N_{Re}, the mean ground-water temperature for computing viscosity, and mean void diameters to compute turbulent flow. The CFPM2 approach, however, is less physically based than the CFPM1 approach. Specifically, complex void distributions are not explicitly represented. Thus, the CFPM2 approach may be limited for simulation of transport if the aquifer is composed of discrete conduit networks rather than preferential flow layers. The impact on ground-water flow of complex void distributions is mimicked by CFPM2 using mean void diameters, upper and lower N_{Re}, and the very large laminar hydraulic conductivities that are typical of larger diameter conduits.

A further limitation of the CFPM2 approach is apparent for two- and three-dimensional problems due to the finite-difference solution scheme implemented in MODFLOW. MODFLOW computes flow through all faces of a MODFLOW cell using the head differences and conductance terms. The resultant velocity vector for each MODFLOW cell is not computed by MODFLOW; however, resultant velocity vectors generally are computed by post-processors using the cell-by-cell flow budget file and plotted on maps as flow vectors. The CFPM2 is designed like MODFLOW in that the check for horizontal turbulent or laminar flow is calculated on each vertical face of every MODFLOW cell. It is possible for one side of a cell to be under turbulent flow conditions and another side to be under laminar flow conditions. An indicator code (turbulence code) is printed in a separate CFPM2 output file that documents whether the flow components on each MODFLOW cell face are laminar or turbulent.

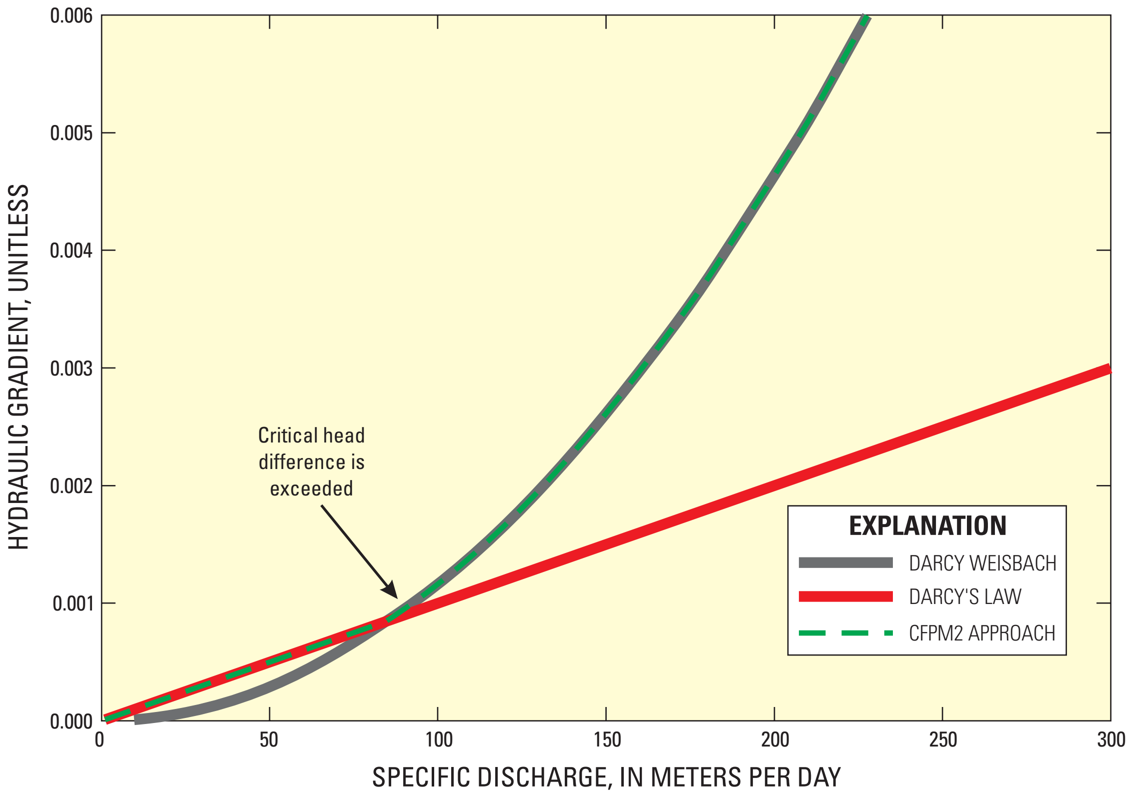


Figure 7. Effects of CFPM2 turbulent hydraulic conductivities on specific discharge relative to Darcy's law and the Darcy-Weisbach equation.

2.4. Upper and Lower Critical Reynolds Numbers

Note that both CFPM1 and CFPM2 require users to assign two critical Reynolds numbers, N_{Re}, that transition flow between laminar and turbulent. Due to conservation of momentum, flow in a laminar state tends to stay laminar and flow in a turbulent state tends to stay turbulent. Therefore, an upper N_{Re} needs to be assigned when a CFPM1 pipe and (or) CFPM2 layer with laminar flow transitions to turbulent flow, and a lower N_{Re} needs to be assigned when a CFPM1 pipe and (or) CFPM2 layer with turbulent flow transitions to laminar flow. During a transient simulation, for example, with lower and upper N_{Re} equal to 2000 and 3000, laminar flow in a hypothetical conduit pipe or layer would transition to turbulent when the R_e equals or exceeds 3000, respectively, and remain turbulent until the R_e is less than or equal to 2000. Conversely, turbulent flow in the hypothetical conduit pipe or layer would transition to laminar when the R_e is less than or equal to 2000, and remain laminar until the R_e is again equal to or exceeds 3000. In the pipe experiments of Reynolds and numerous similar experiments in hydraulics laboratories, this phenomena was repeatedly observed. For porous media, upper and lower N_{Re} have not been observed; therefore, upper and lower N_{Re} for CFP simulations of porous media can be set approximately equal, for example, 11 and 10, respectively). Additionally, it has been observed that between the upper and lower N_{Re}, the discharge in pipes is a function of mean velocity to the power of greater than 1 but less than 2 (Vennard and Street, 1975, chap. 7).

Chapter 3. Description of Conduit Flow Process (CFP) Programming

Conduit flow theory was integrated into the MODFLOW-2005 finite-difference solution scheme. Implementation of this theory required many FORTRAN subroutines to be developed or integrated into the MODFLOW-2005 source code. This section of the report documents not only the finite-difference form of many of the theoretical equations governing conduit flow, but also the CFP subroutines as they appear in the FORTRAN main file for MODFLOW-2005.

3.1. Integration of Simulation Modes

Three CFP modes were implemented into the MODFLOW-2005 source code, specifically, CFPM1, CFPM2, and CFPM3. Several new subroutines were written specifically for CFPM1, including subroutines that: (1) compute flow in partially filled pipes, (2) account for changes in pipe storage when pipes are partially filled, (3) compute ground-water exchange between the conduit pipes and porous media based on the pipe conductance specified for a model cell, and (4) correct ground-water exchange between partially filled conduit pipes and porous media. Pipe nodes also were set to dry when heads in the pipes were at or below the bottoms of the pipes. As a result, CFPM1 rigorously represents the dual porosity nature of many flow systems.

CFPM2 does not activate a pipe network. Rather, several new subroutines were developed within MODFLOW-2005 that: (1) determine whether horizontal flow through the vertical faces of model cells is laminar or turbulent; (2) adjust conductances assembled by the Block-Center Flow (BCF), Layer-Property Flow (LPF), and Hydrogeologic-Unit Flow (HUF) Packages if flow is turbulent; (3) adjust flows in the budget subroutines for the BCF, LPF, and HUF Packages if flow is turbulent; and (4) write results related to turbulent flow into text files for interpretation and plotting. Less programming was required for CFPM2, which was a consequence of the goal for this mode to represent turbulent ground-water flow if needed, while requiring less data on the geometry and hydraulic behavior of the secondary porosity.

CFPM3 is a result of the MODFLOW-2005 modular design. Occasionally, new Packages and Processes for MODFLOW work together without the need for much interface programming. This is specifically the case for CFPM1 and CFPM2; during experimentation and testing, it was discovered that these Modes could work together. Simultaneous activation of CFPM1 and CFPM2 is named CFPM3, and the CFP Input File is designed to accommodate the data needed to iteratively solve coupled CFPM1 and CFPM2 flow equations. Although data requirements are intensive for CFPM3, this Mode may be useful for modeling a porous media and conduit flow layers interacting with a network of large secondary interconnected voids that resemble pipes.

3.2. Conduit Flow Process (CFP) Program Flow

The CFP subroutines can be grouped into specific categories; namely allocate and read, read and prepare, distribute direct recharge, exchange, formulate, approximate, output control, water budget, and output (fig. 8). The functionality and details of the key subroutines in these categories are described below. Also discussed is the compatibility of the CFP with other MODFLOW-2005 developments.

3.2.1. Conduit Flow Process (CFP) Allocate and Read Subroutine (CFP1AR)

CFPMODULE defines all the variables needed for the CFP. A CFP allocate and read (CFP1AR) subroutine allocates the arrays and variables needed for CFP simulations. CFP1AR begins by reading the CFP Mode (either 1, 2, or 3). If CFPM1 is active, the maximum number of conduit nodes, pipes, and MODFLOW layers are read. Based on these data, many arrays used to solve pipe flow equations are allocated, such as arrays that hold pipe diameters, internal roughness, tortuosity, and N_{Re}. Following these allocation statements, the length and time units of the model are used to set the value of the gravitational constant for pipe-flow calculations. The mean pipe water temperature, in degrees Celsius (°C), is then read and used with the length and time units of the model to calculate ground-water dynamic and kinematic viscosity. Viscosity is related to the internal fluid friction and shearing forces as parcels of ground water move past one another. Dynamic viscosity is defined as a constant of proportionality between a shear stress and a velocity gradient, and is approximated in CFP based on water temperature by the following empirical equations from Weast (1979). If the water temperature (T_w) is greater than or equal to 0°C and less than 20°C, dynamic viscosity (μ) is calculated as:

$$\mu = \frac{10^{\left(\frac{1301 - 1.30233}{998.33 + 8.1855(T_w - 20) + 0.00585(T_w - 20)^2}\right)}}{1000}. \quad (29)$$

If the water temperature (T_w) is greater than or equal to 20°C and less than 100°C, dynamic viscosity (μ) is calculated as:

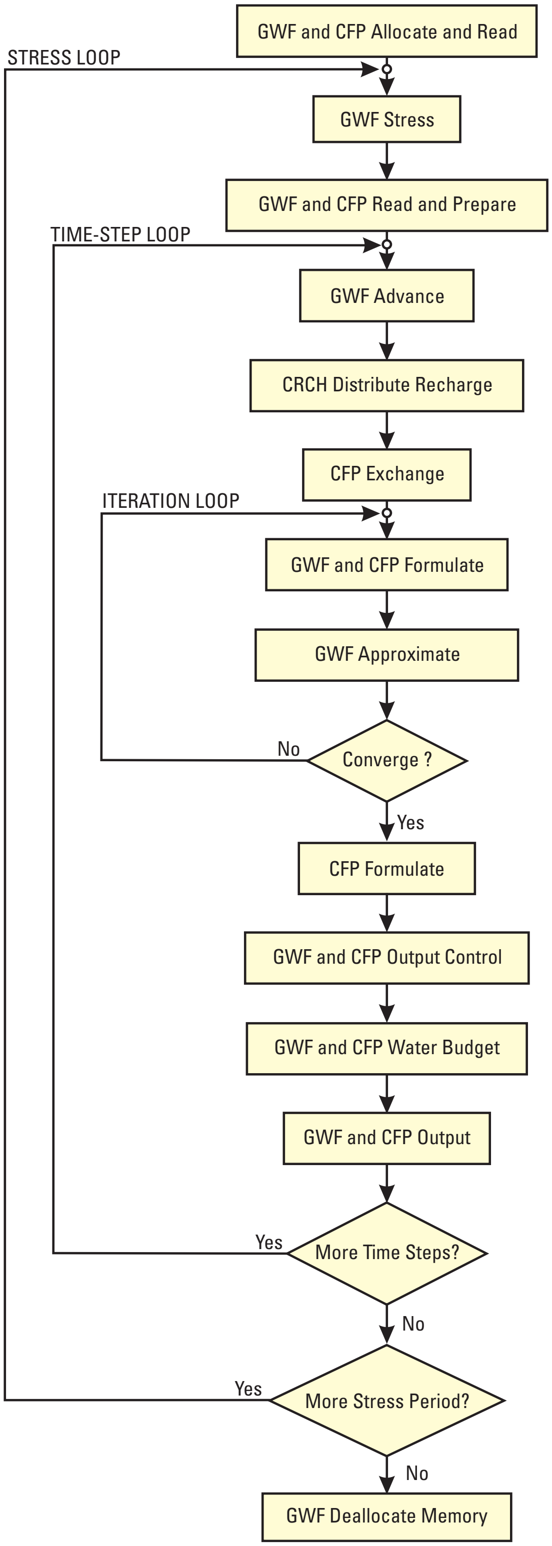
$$\mu = 10^{\left(\frac{1.3272(20 - T_w) - 0.001053(T_w - 20)^2}{(T_w + 105)}\right)} \left(\frac{1.002}{1000}\right). \quad (30)$$

These empirical equations create an exponential decline of dynamic viscosity with increasing water temperature (fig. 9).

Kinematic viscosity, v, is used in conduit layer and pipe flow equations, and simply equals the dynamic viscosity divided by the fluid density. Fluid density is computed in CFP using the following empirical relation between T_w and fluid density, ρ, from Weast (1979):

$$\rho = \frac{\begin{pmatrix} 999.83952 + 16.945176(T_w) - 7.9870401 - 3(T_w^2) \\ -46.170461 - 6(T_w^3) + 105.56302 - 9(T_w^4) - 280.54253 - 12(T_w^5) \end{pmatrix}}{\left(1 + 16.879850 - 3(T_w)\right)}. \quad (31)$$

This empirical equation also creates a nonlinear and exponential decrease in water density with increasing water temperature (fig. 10).



EXPLANATION

GWF	Ground-water flow process
CFP	Conduit flow process
CRCH	Conduit recharge package

Note: CFP formulate is called again after the iteration loop to equilibrate pipe heads and flows with porous media heads and flows

Figure 8. Integration of the Conduit Flow Process (CFP) into MODFLOW-2005.

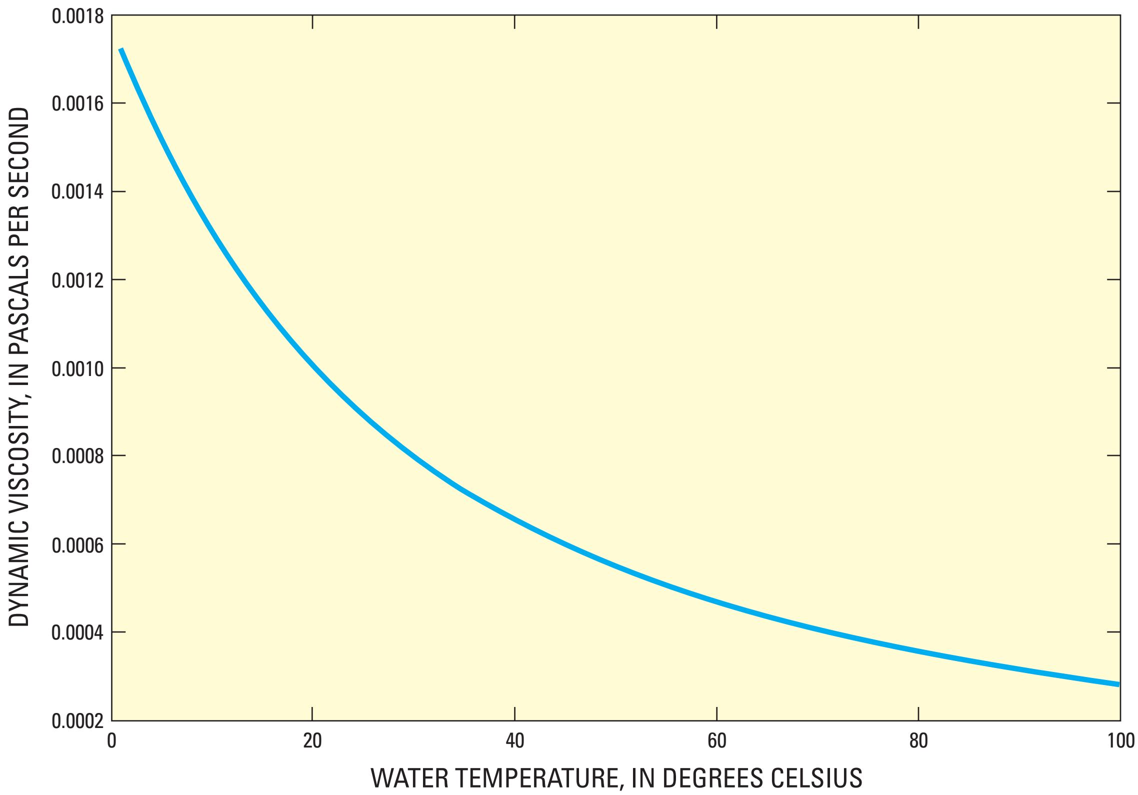


Figure 9. Relation used in the Conduit Flow Process to calculate dynamic viscosity using ground-water temperature.

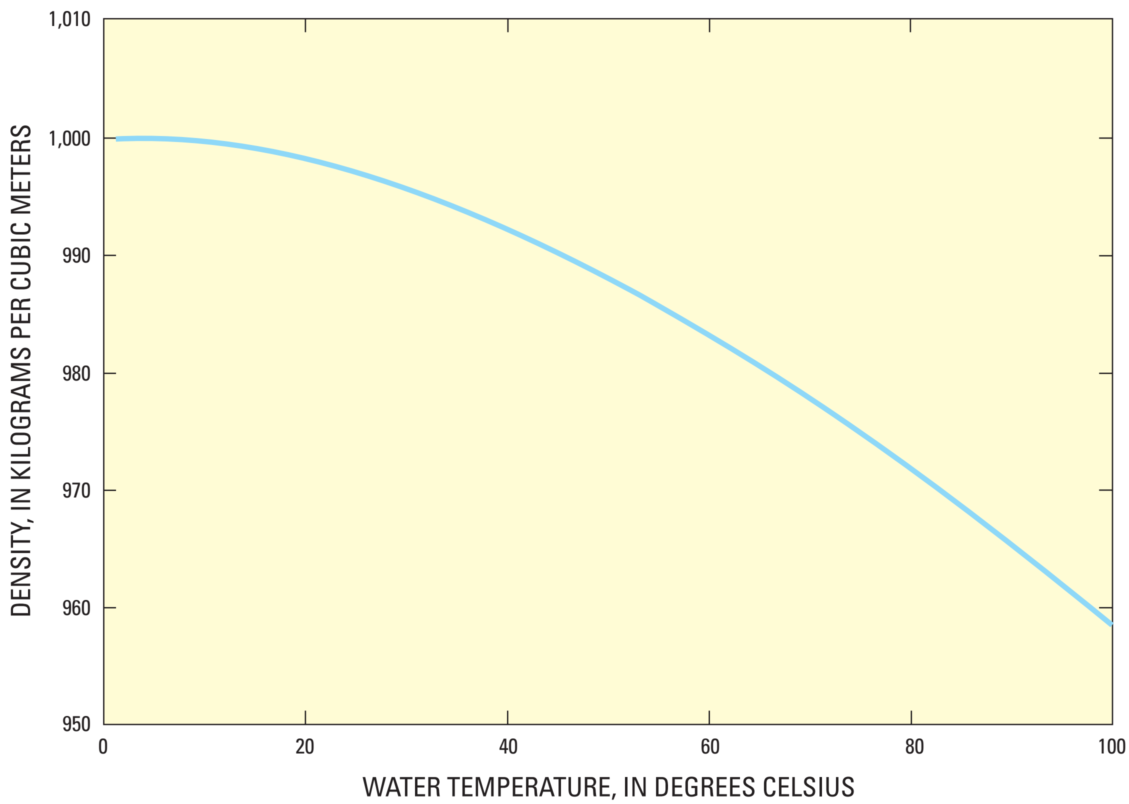


Figure 10. Relation used in the Conduit Flow Process to calculate fluid density using ground-water temperature.

Following calculations of ground-water density and viscosity, the CFP1AR subroutine reads the geometry of the pipe network. A pipe node is conceptualized as the end of a pipe segment, or intersection of up to six pipes in the horizontal or vertical directions (fig. 3A). Each node is located at the center of a finite-difference cell, and there can be only one node within a finite-difference cell. Finite-difference cells can have equal or different row and column lengths. Model layers composed of finite-difference cells also can have constant or variable thicknesses. Diagonally adjacent porous media cells can be linked by conduit flow nodes and pipes (fig. 3C). Also, it is important to place nodes in all cells where conduit pipes occur. Without nodes, porous media heads will not be affected by the presence of conduit pipes. Note the resolution of the model grid should be fine enough so the conduit features span multiple grid cells. A pipe network, for example, cannot be designed within a single finite difference cell.

Each node is assigned a node number and each two-node segment is assigned a pipe number. The developer of a site-specific application of CFPM1 should design a drawing of the conduit network within each MODFLOW layer prior to generating this dataset. Each node location is assigned a MODFLOW row number (i), column number (j), and layer number (k). For each node (in), the node numbers that are connected are listed and then the pipe segment numbers that are connected to node in are listed. The elevation of each node is also assigned. For each pipe, the diameter, tortuosity, and roughness are read and assigned. Tortuosity affects the pipe length, which is adjusted by multiplication of the tortuosity factor.

Individual conduit pipes have constant length, diameter, tortuosity, and internal roughness (fig. 11); however, these properties can vary from pipe to pipe. Thus, in cases with abundant field data on the void architecture and hydraulic behavior, complex two- or three-dimensional networks of

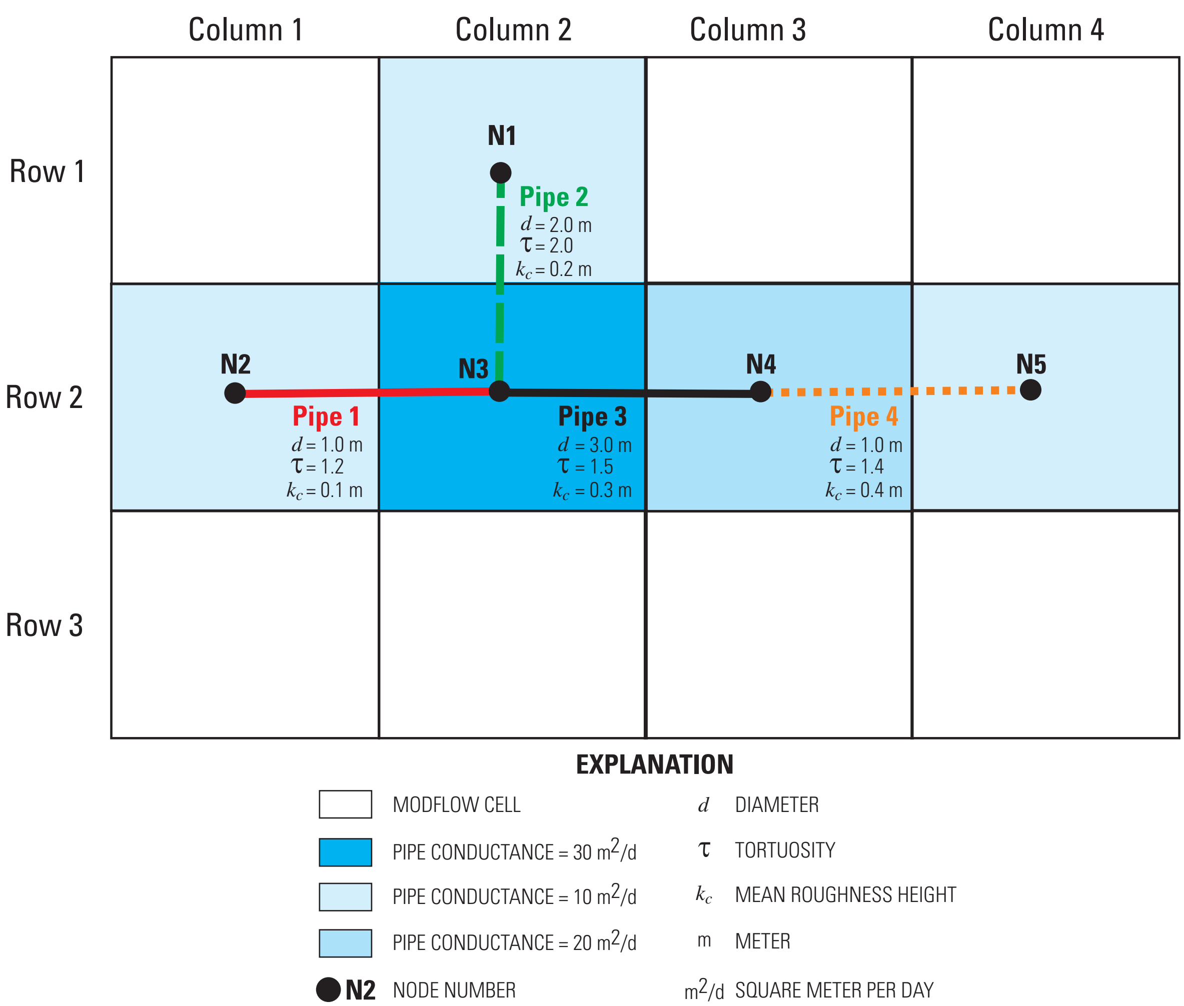


Figure 11. Possible variations in properties of conduit flow pipes in MODFLOW cells.

conduit flow pipes and nodes can be designed to represent interconnected or dead-end voids in the subsurface. Flow calculations assume pipe nodes are located in the center of MODFLOW cells. An exception is in the vertical direction, for which there are two options. First, pipe nodes can be assigned elevations above a datum, and therefore, are not restricted to center elevations of MODFLOW cells. Second, pipe nodes can be assigned a distance above or below the center of the MODFLOW cell (fig. 12). With this second option, if the distance is set to zero, pipe nodes are assumed to exist at the vertical center of the MODFLOW cell.

After reading the elevation of pipe nodes, the CFP1AR subroutine reads a flag (SA_EXCHANGE) that determines whether the pipe conductance is: (1) assigned by the user in the CFPM1 input file, or (2) computed internally based on a user-defined conduit wall permeability and the surface area of the pipes connected to the nodes. The finite-difference equation used to compute the pipe conductance is discussed in more detail later.

Three parameters controlling iterations of conduit pipe flow equations are subsequently read by CFP1AR. These include a convergence criterion (EPSILON) for the Newton-Raphson iterations for pipe flow equations, an integer number for the maximum number of Newton-Raphson iterations (NITER), and a parameter of relaxation (RELAX) that determines the step length of the Newton-Raphson iterations. For EPSILON, use a very small number such as 0.000001. Changing RELAX to a value slightly less than 1.0 may facilitate convergence of the pipe flow equations. If convergence cannot be achieved within NITER iterations, the CFP will stop and a warning will be printed in the MODFLOW listing file. An integer print flag (P_NR) for the results of the Newton-Raphson iterations is subsequently read by the CFP. If P_NR equals 1, the maximum node head change for Newton-Raphson iterations is printed in the MODFLOW output file along with the MODFLOW iteration number. If P_NR equals 0, results from the Newton-Raphson iterations are not printed. Newton-Raphson iterations are formulated to compute node heads that sum the flows to the nodes to zero, and are discussed in more detail later.

Pipe hydraulic data subsequently are read by the CFP1AR subroutine. These include the pipe number, diameter, tortuosity, roughness, and critical Reynolds numbers (N_{Re}). These hydraulic properties can vary from pipe to pipe (fig. 11). Following hydraulic properties, node heads (N_HEAD) are subsequently read by the CFP1AR subroutine, one for each node. Node heads are the piezometric heads for constant-head nodes (positive values) or a flag (-1), indicating that heads will be calculated during simulation. Finally, pipe conductances or conduit wall permeability terms are read by the CFP1AR subroutine, one for each MODFLOW cell that contains a node.

If CFPM2 or CFPM3 are active, the CFP1AR subroutine reads data that are needed to activate conduit flow (preferential flow) layers in the iterative solutions. First, the total number of CFPM2 layers is read, followed by the conduit flow layer numbers. For example, a model could have 10 total layers, with layer numbers 2, 5, and 8 assigned as conduit flow layers.

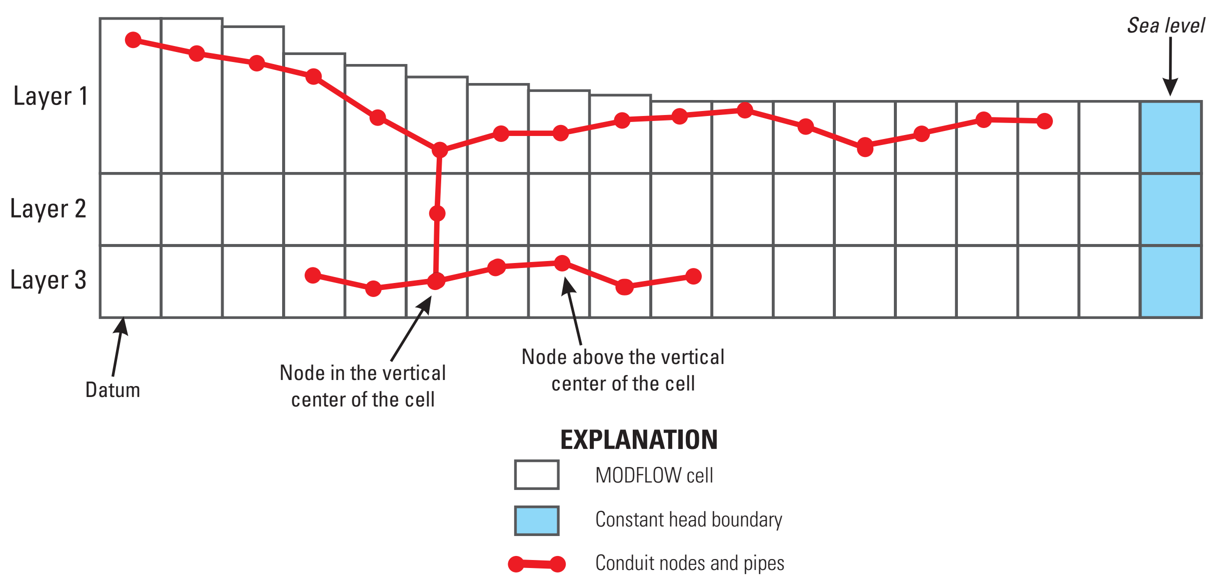


Figure 12. Possible variations in elevation of conduit nodes in MODFLOW cells.

Mean ground-water temperature in the preferential flow layers is subsequently read and used to compute fluid density and viscosity. Following water temperature, mean void diameters and upper and lower critical Reynolds (N_{Re}) numbers are read for each preferential flow layer. For example, a model with three conduit flow layers would have three mean void diameters and N_{Re}; specifically, one for each conduit layer. These mean void diameters and N_{Re} are used to switch flow equations from laminar to turbulent as discussed in chapter 2. Finally, the CFP1AR subroutine also defines and allocates all the arrays and variables needed for the Conduit Recharge (CRCH) Package and Conduit Output Control (COC) File.

3.2.2. Newton-Raphson Iterations

Computations of pipe flows, storage changes, and matrix exchanges depend on heads at pipe nodes; however, node heads are unknown. Newton-Raphson iterations (figs. 13 and 14) are used to solve for node heads. Newton-Raphson iterations solve nonlinear equations by finding the root of the equations (Mehl, 2006). Specifically, the derivative (slope of a tangent line) of a nonlinear equation is calculated with an initial guess of the dependent variable (node head). The value of the dependent variable (node head) where the tangent line crosses zero on an axis (Mehl, 2006, fig. 3) is used as the updated dependent variable (node head). This procedure repeats itself until convergence is achieved (the nonlinear equation approximately equals zero) or until the maximum number of iterations is reached.

Newton-Raphson iterations solve for node heads that satisfy Kirchhoff's law (Clemens, 1998; Horalacher and Lüdecke, 1992). Kirchhoff's law states that the summation of the volumetric flows to node *in*, including pipe flows (Q_{ip}), individual matrix exchange (Q_{ex}), direct recharge (Q_R), and changes in storage (Q_s) should equal zero, specifically:

$$\sum_{i=1}^{np} Q_{ip} - Q_{ex} + Q_R \pm Q_s = 0 \quad . \qquad (32)$$

Note that changes in pipe storage are only computed when the water level in the pipe drops below the top of the pipe. An assumption is made that the node head applies along the length of the pipe in the encompassing MODFLOW cell, so that volumetric storage changes are computed.

During Newton-Raphson iterations, volumetric flow driven by the head loss along the pipe (Q_{ip}) is computed using either the Hagen-Poiseuille equation (White, 1988) or Darcy-Weisbach equations (Horlacher and Ludecke, 1992; Hucking-haus, 1998), depending upon a comparison of Reynolds numbers (R_e). If flow is laminar; that is, if the internally computed R_e is less than the user-defined critical Reynolds number (N_{Re}), the Hagen-Poiseuille equation is applied as:

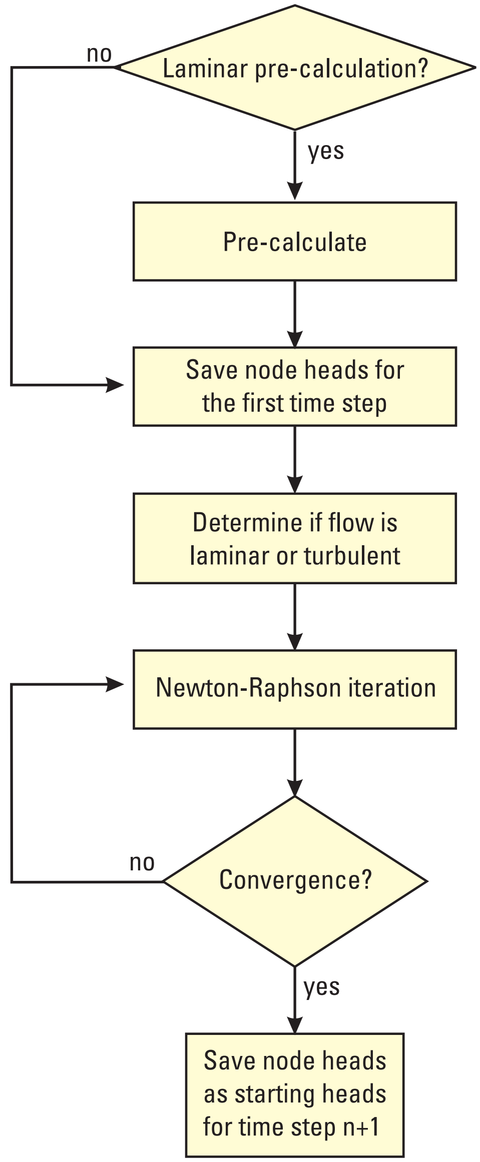


Figure 13. Conduit pipe formulate (CFPM2FM) subroutine and approximate procedures.

$$Q_{ip} = -\frac{\pi d_{ip}{}^{4} g\left(h_{in} - h_{neighbor}\right)}{128\,\nu \Delta l_{ip}\,\tau_{ip}} \quad , \qquad (33)$$

where

$h_{neighbor}$ is the node head at the other end of the pipe segment,

d_{ip} is the pipe diameter,

l_{ip} is the pipe length,

and

τ_{ip} is the pipe tortuosity.

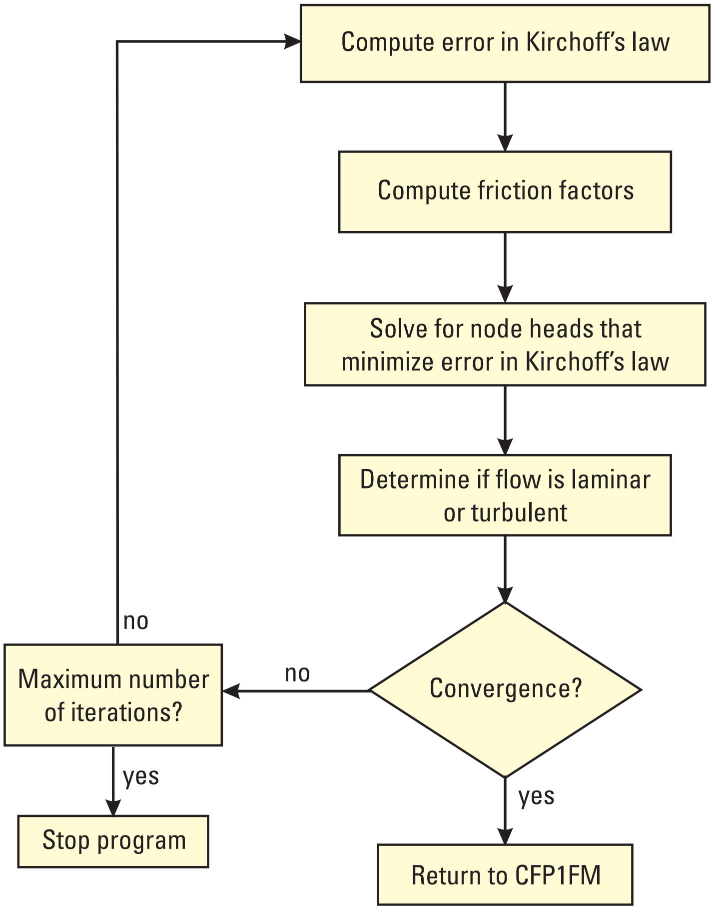


Figure 14. Newton-Raphson iteration steps.

If flow is turbulent, that is, if the internally computed R_e is greater than the user-defined N_{Re}, the Darcy-Weisbach equation is applied as:

$$Q_{ip} = -\sqrt{\frac{\left|\left(h_{in} - h_{neighbor}\right)\right| g d_{ip}^{\,5} \pi^2}{2\Delta l_{ip} \tau_{ip}}} \log\left(\frac{2.51\nu}{4\sqrt{\frac{2\left|\left(h_{in} - h_{neighbor}\right)\right| g d_{ip}^{\,3}}{\Delta l \tau_{ip}}}} + \frac{k_c}{3.71 d_{ip_{ip}}} + \right) \frac{\left(h_{in} - h_{neighbor}\right)}{\left|\left(h_{in} - h_{neighbor}\right)\right|} , \quad (34)$$

where A_{ip} is the cross-sectional area of the pipe *ip*. The equation used to internally compute the R_e for pipe *ip* is:

$$R_e = \frac{V_{ip} d_{ip}}{\nu_{ip}} , \quad (35)$$

where V_{ip} is the pipe mean flow velocity [LT^{-1}], and is computed as the pipe volumetric flow divided by the flow cross-sectional area based on the pipe diameter d_{ip} [L].

A special case occurs in which Newton-Raphson iterations are not used; specifically, at the first iteration of the first time step and first stress period (fig. 13). In this case, node heads are set equal to the elevation of the top of the node, and the sum of the flows driven by head losses along the pipe is pre-calculated using the Hagen-Poiseuille equation. When the model has advanced past this initial iteration, node heads from the previous time step are used to start Newton-Raphson iterations for the current time step.

3.2.3. Exchange Subroutines

A subroutine was modified and integrated into MODFLOW-2005 to compute exchanges of ground water between conduit pipes and porous media. The total volumetric exchange (Q_{tex}) [L^3T^{-1}] between all the conduit pipes and porous media is computed by summing the individual volumetric exchanges (Q_{ex}) [L^3T^{-1}] as follows:

$$Q_{tex} = \sum_{in=1}^{mxnode} \sum_{ip=1}^{np} Q_{ex} , \qquad (36)$$

where

in	is an integer subscript for each node,
mxnode	is the maximum number of nodes in the model,
ip	is an integer subscript for each pipe that connects to node *in*,

and

np	is the number of pipes connected to node *in*.

Total exchange flow, Q_{tex}, is reported in CFP and MODFLOW water budgets.

Individual volumetric exchanges, Q_{ex}, are computed using the Darcy formulation:

$$Q_{ex} = \alpha_{j,i,k}(h_{in} - h_{j,i,k}) , \qquad (37)$$

where

$\alpha_{j,i,k}$	is the pipe conductance [L^2T^{-1}],
h_{in}	is the head [L] at node *in* computed by the CFP,

and

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

This equation assumes that ground-water exchange is not turbulent. Negative Q_{ex} indicates exchange flow is from the porous media into the conduit pipe(s). Conversely, positive Q_{ex} indicates flow is from the conduit pipe(s) into the porous media.

Two options are available for assembling $\alpha_{j,i,k}$. When using the first option (SA_EXCHANGE = 0), $\alpha_{j,i,k}$ is entered by the user for each node in the CFP Input File and no internal computation of the pipe conductance is done by the computer code. A secondary option (SA_EXCHANGE = 1) is available in CFP that internally computes $\alpha_{j,i,k}$ using pipe geometry data assigned in the CFP Input File for the pipes that connect to the node. When SA_EXCHANGE = 1, conduit wall permeability terms ($K_{j,i,k}$) are assigned in the CFP Input File for each node, and $\alpha_{j,i,k}$ is internally computed by the computer program as:

$$\alpha_{j,i,k} = \frac{\sum_{ip=1}^{np} K_{j,i,k}\pi d_{ip}\frac{1}{2}(\Delta l_{ip}\tau_{ip})}{r_{ip}} , \qquad (38)$$

where

$K_{j,i,k}$	is the conduit wall permeability term [LT^{-1}] in MODFLOW cell *j,i,k*,
π	is the mathematical constant pi [unitless],
d_{ip}	is the diameter [L] of pipe *ip* connected to node *in*,
Δl_{ip}	is the straight-line length between nodes [L] of pipe *ip* connected to node *in*,
τ_{ip}	is the tortuosity [unitless] of pipe *ip* connected to node *in*,

and

r_{ip}	is the radius of the pipe [L].

Due to incomplete knowledge of the conduit geometry and the rock matrix permeability at the conduit walls, and regardless of the option selected to assemble $\alpha_{j,i,k}$, it is likely that this parameter will be modified during model calibration.

Ground-water exchange between conduit layers (CFPM2) and porous media layers is computed with existing MODFLOW-2005 subroutines. No modifications were made to subroutines that compute vertical flow. For example, vertical conductances provide the resistance to exchange between conduit and porous media layers, vertical flow corrections are activated under dewatered or "perched" conditions (Harbaugh, 2005), and conduit layer or porous media cells can convert from wet to dry (Harbaugh, 2005, p. 5–7 to 5–11). When using conduit layers (CFPM2 and CFPM3), both conduit layers and porous media layers should be assigned as fully convertible. For example, assign all layers (both conduit and porous media layers) as LAYCON = 3 if the BCF package is active, or layer type flag greater than zero if the LPF or HUF packages are active. Conductances for MODFLOW layers that are not convertible are not updated every iteration; therefore, modifications for turbulence will not be performed every iteration as needed in nonconvertible layers.

3.2.4. Conduit Pipe Recharge Subroutine

The CFP1RCH1RP subroutine routes recharge, Q_R, directly into a pipe node without infiltrating through the porous media. Specifically, a fraction of the diffuse recharge (assigned in the traditional MODFLOW RCH Package) can be routed directly into pipe nodes. The routed direct recharge is subtracted from the diffuse recharge (assigned in the traditional MODFLOW RCH Package) for traditional MODFLOW water-budget calculations. This functionality may be useful for scenarios in which rainfall runs directly into karst features, such as sinkholes or swallets. Swallets commonly occur in the transition zone between upland regions and karst plains in Florida. Many of these features receive untreated stormwater and other surface waters directly from urban areas (Means and

Scott, 2005). The direct conduit recharge may be less than or equal to the diffuse areal recharge assigned in the standard MODFLOW-2005 Recharge (RCH) Package. For example, if the direct conduit recharge in a karst environment is 10 percent of the diffuse areal recharge, a multiplier of 0.1 can be assigned in the new CRCH Package. If the direct conduit recharge in a karst environment is 100 percent of the diffuse areal recharge, a multiplier of 1.0 can be assigned in the new CRCH Package. The CFP1RCH1RP subroutine reads node numbers and diffuse recharge fractions (P_CRCH) that route water directly into pipe nodes.

3.2.5. Subroutine for Solution of Conduit Flow Process Mode 2 (CFPM2) Heads

In CFPM2 layers, a nonlinear form of the finite-difference equation in MODFLOW results from the adjustment of conductance once the critical head difference along a row or column is exceeded, as previously described. The solution for CFPM2 heads uses existing MODFLOW iterative solvers with different conductances for laminar and turbulent flow conditions as described in chapter 2. The laminar conductance remains a constant specified in the input as long as the flow is laminar; however, the turbulent conductance is a nonlinear power function of the Reynolds number. Subroutine CFPM2FM was created and integrated into MODFLOW-2005 to modify conductances assembled by the BCF, LPF, and HUF packages every iteration when flow through a face of a MODFLOW cell is determined to be turbulent (when the calculated head difference exceeds the critical head difference).

MODFLOW is designed to compute horizontal flow through vertical faces of a model cell. Exceptions are the last row and column. No flow occurs through the front faces of model cells in the last row. Likewise, no flow occurs through the right faces of model cells in the last column. Note the resultant flow vector for each MODFLOW cell is not computed by MODFLOW; however, resultant vectors generally are computed by post-processors using the cell-by-cell flow budget file, and plotted on maps as velocity vectors. The CFPM2 accounts for turbulence in the flow computations through the faces of MODFLOW cells, but does not compute the resultant flow vector nor determine whether the resultant vector is laminar or turbulent.

Recall from equation 28 that the Δh_{crit} along the side of a cell in the row or column direction remains constant based on the K_{lam} and lower or upper N_{Re} specified and is calculated as follows:

$$\Delta h_{crit_{j+1/2,i,k}} = \frac{N_{re_k} v_k \Delta l_{j+1/2,i,k}}{K_{lam\, j,i,k} d_{void_k}}, \tag{39}$$

$$\Delta h_{crit_{j-1/2,i,k}} = \frac{N_{re_k} v_k \Delta l_{j-1/2,i,k}}{K_{lam\, j,i,k} d_{void_k}}, \tag{40}$$

$$\Delta h_{crit_{j,i+1/2,k}} = \frac{N_{re_k} v_k \Delta l_{j,i+1/2,k}}{K_{lam\, j,i,k} d_{void_k}}, \tag{41}$$

and:

$$\Delta h_{crit_{j,i-1/2,k}} = \frac{N_{re_k} v_k \Delta l_{j,i-1/2,k}}{K_{lam\, j,i,k} d_{void_k}}. \tag{42}$$

At the onset of turbulent flow, the turbulent inter-nodal conductance along rows ($CR^{kiter}_{turb_{j+1/2,i,k}}$ and $CR^{kiter}_{turb_{j-1/2,i,k}}$) and columns ($CC^{kiter}_{turb_{j,i+1/2,k}}$ and $CC^{kiter}_{turb_{j,i-1/2,k}}$) are computed as:

$$CR^{kiter}_{turb_{j+1/2,i,k}} = CR_{j+1/2,i,k} \sqrt{\frac{CR_{j+1/2,i,k} \left|\Delta h_{crit_{j+1/2,i,k}}\right|}{CR^{kiter-1}_{turb_{j+1/2,i,k}} \left|h_{j,i,k} - h_{j+1,i,k}\right|^{kiter-1}}}, \tag{43}$$

$$CR^{kiter}_{turb_{j-1/2,i,k}} = CR_{j-1/2,i,k} \sqrt{\frac{CR_{j-1/2,i,k} \left|\Delta h_{crit_{j-1/2,i,k}}\right|}{CR^{kiter-1}_{turb_{j-1/2,i,k}} \left|h_{j,i,k} - h_{j-1,i,k}\right|^{kiter-1}}}, \tag{44}$$

$$CC^{kiter}_{turb_{j,i+1/2,k}} = CC_{j,i+1/2,k} \sqrt{\frac{CC_{j,i+1/2,k} \left|\Delta h_{crit_{j,i+1/2,k}}\right|}{CC^{kiter-1}_{turb_{j,i+1/2,k}} \left|h_{j,i,k} - h_{j,i+1,k}\right|^{kiter-1}}}, \tag{45}$$

$$CC^{kiter}_{turb_{j,i-1/2,k}} = CC_{j,i-1/2,k} \sqrt{\frac{CC_{j,i-1/2,k} \left|\Delta h_{crit_{j,i-1/2,k}}\right|}{CC^{kiter-1}_{turb_{j,i-1/2,k}} \left|h_{j,i,k} - h_{j,i-1,k}\right|^{kiter-1}}}. \tag{46}$$

where $CR_{j+1/2,i,k}$, $CR_{j-1/2,i,k}$, $CC_{j,i+1/2,k}$, and $CC_{j,i-1/2,k}$ are the traditional MODFLOW conductances [L^2T^{-1}] along rows and columns, respectively; and Δh_{crit} is the critical head difference between, for example, model cells *j,i,k* and *j+1,i,k* for flow to the right, and model cells *j,i,k* and *j,i+1,k* for flow to the front. Turbulent conductances are used in MODFLOW solvers to compute heads in conduit layer model cells. Iterative solvers, such as SIP or PCG, are required when conduit layers are active in a simulation. Direct solvers for MODFLOW, such as DE4 cannot be applied if CFPM2 is active. Because turbulent conductances are used for the preferential flow layers, these layers should not dewater, even though it is required that these be specified as convertible layers.

A limitation of the CFPM2 approach is apparent for two- and three-dimensional problems due to the finite-difference solution scheme implemented in MODFLOW. MODFLOW computes flow through all horizontal faces of a MODFLOW cell using the head gradient between the side defined by the two adjacent cells and the cell row or column conductance.

The resultant velocity vector for each MODFLOW cell is not computed by MODFLOW; however, resultant velocity vectors generally are computed by post-processors using the cell-by-cell flow budget file, and plotted on maps as flow vectors. The CFPM2 is designed like MODFLOW in that the check for horizontal turbulent versus laminar flow is calculated for each vertical cell face. It is possible for one side of a cell to be under turbulent flow conditions and another side to be under laminar flow conditions. An indicator code (turbulence code) is printed in a separate CFPM2 output file that documents whether the flow components on each conduit layer model cell face is laminar or turbulent.

3.2.6. Conduit Flow Process (CFP) Budget Routines

Several water-budget subroutines were created or integrated into MODFLOW-2005 for the CFP. These subroutines were used to modify: (1) water budgets at constant-head boundaries that contain a conduit pipe; (2) adjacent flows computed by the BCF, LPF, and HUF packages; and (3) volumetric water budgets for the MODFLOW cells. Subroutines also were integrated into MODFLOW-2005 for the CFP to compute conduit-pipe and node water budgets.

3.2.6.1. Modification of Flows to Constant-Head Boundaries and Laminar Flow Model Budgets

When conduit pipes are active, modifications are made to MODFLOW volumetric budgets to account for: (1) Q_{tex}, the total volumetric exchange of ground water between the pipes and laminar flow model; and (2) the volume of water exchanged between the pipes and laminar flow model constant-head boundary cells. At constant-head boundaries that contain a conduit pipe, the budget subroutines of the BCF, LPF, and HUF packages were modified to account for the exchange of ground water between the pipe and boundary cell. This exchange is computed as the product of the pipe conductance, $\alpha_{j,i,k}$, and head difference between the pipe and constant head boundary. The exchange flow is positive if flow is from the pipe into the boundary, and conversely, is negative if the flow is from the boundary into the pipe. Ultimately, these modifications adjust flow reported in conventional MODFLOW water budgets in the Listing or Output File.

3.2.6.2. Modifications to BCF, LPF, or HUF Adjacent Flows

When conduit layers are active (CFPM2) and flow through the face of a conduit layer model cell is turbulent, turbulent conductances are used to compute adjacent flows in the budget subroutines of the BCF, LPF, or HUF packages. Flow to the right is computed as:

$$Q_{j+1/2,i,k} = CR_{turb\,j+1/2,i,k}(h_{j,i,k} - h_{j+1,i,k})\,. \tag{47}$$

Flow to the left is computed as:

$$Q_{j-1/2,i,k} = CR_{turb\,j-1/2,i,k}(h_{j,i,k} - h_{j-1,i,k})\,. \tag{48}$$

Flow to the front is computed as:

$$Q_{j,i+1/2,k} = CC_{turb_{j,i+1/2,k}}(h_{j,i,k} - h_{j,i+1,k})\,. \tag{49}$$

Flow to the back is computed as:

$$Q_{j,i-1/2,k} = CC_{turb_{j,i-1/2,k}}(h_{j,i,k} - h_{j,i-1,k})\,. \tag{50}$$

3.2.6.3. Conduit Pipe and Node Water Budgets

Subroutines were integrated into MODFLOW-2005 to track water budgets for conduit pipes and nodes. For conduit pipes, sources and sinks of ground water in conduit pipe budgets include matrix exchange, direct recharge, flows to and from constant-head boundaries, and storage changes when pipes are partially full. Water budgets also are tracked for each node when conduit pipes are active. This includes flows from nodes with constant heads, direct recharge, matrix exchange, storage changes, and total flow into and out of the node from connecting conduit pipes.

3.2.7. Conduit Flow Process (CFP) Output Control Routines

Subroutines were integrated into MODFLOW-2005 to output CFP results into text files. CFP results written to text files include node heads and water budgets. Some conduit flow results automatically are written to the conventional MODFLOW Listing File. Specifically, at the end of each time step whether flow in each pipe is laminar or turbulent, the flow rate within each pipe, Reynolds number, and residence time are automatically written to the conventional MODFLOW Listing File. The residence time in each pipe is calculated as the distance it takes for a particle to travel along the center of the pipe divided by the mean pipe flow velocity. The particle travel distance is simply the product of the pipe length and tortuosity, and the mean pipe flow velocity is calculated as the volumetric flow rate divided by the flow cross-sectional area.

If time-series data are desired in a format useful for plotting and graphing programs, the Conduit Flow Process Output Control (COC) File is needed. The COC File holds the total number of nodes, specific node numbers, and time-step frequency for writing node heads. In addition to writing node heads, water-budget information also is reported including direct recharge, matrix exchange, flows to and from connecting pipes, changes in storage for partially saturated cases, and flows from constant-head boundaries. The COC File also includes the total number of pipes, specific pipe numbers, and time-step frequency for writing the flow rate in the pipe to an output file. In addition to pipe flow rates, the internally computed R_e also is written in time-series format.

An additional output file named turblam.txt also is written when preferential flow layers are active. This file lists an integer for each conduit layer model cell that indicates whether flow to the right, front, or both is laminar or turbulent at the end of each stress period. If the integer equals 0, flow to the right and front are both laminar. If the integer equals 1, flow to the right is turbulent, whereas flow to the front is laminar. If the integer equals 2, flow to the front is turbulent, whereas flow to the right is laminar. If the integer equals 3, flow to the front and right both are turbulent. These integers can be plotted with visualization programs to show locations in the aquifer where flow is laminar or turbulent.

3.3. MODFLOW-2005 Compatibility

Because the CFP was created with MODFLOW-2005 version 1.2.01, dated January 17, 2007, CFP compatibility is restricted to programming available with that version of MODFLOW (table 1). Note that CFP is not compatible with the parameter estimation and sensitivity processes of MODFLOW-2000 (Harbaugh and others, 2000). Parameter estimation, however, could be accomplished with CFP using programs such as UCODE (Poeter and Hill, 1998) or PEST (Doherty, 2002), which operate independently from MODFLOW. Further programming efforts to increase compatibility of CFP with other MODFLOW developments may be done in the future.

Chapter 4. Conduit Flow Process (CFP) Input Instructions

Input for running the CFP are contained within the Name File, CFP Input File, CRCH Package, and (or) COC File. Input data are read in either free or fixed format for each input variable or array. Free format variables and arrays do not need to occupy a fixed number of columns in a row. Conversely, variables and arrays read with a fixed format must occupy specific columns in a row. Comments can be put into any of the input files by starting the comment with a “#.” All text on the line after this symbol is ignored. Recall iterative solvers, such as SIP or PCG, are required when conduit layers are active in a simulation. Direct solvers for MODFLOW, such as DE4 cannot be applied if CFPM2 is active.

4.1. Name File

The Name File (Harbaugh, 2005) is used to specify that the CFP will be used in a simulation. This file requires the Package initials, the Fortran unit number that Files and Packages will be read from or written to, and the names of the files that either hold input data or will receive model output data in text or binary form. An example of a line within the Name File that activates the CFP would be “CFP 16 test1c.cfp,” where “CFP” means the Conduit Flow Process will be active during this simulation, “16” is the assigned Fortran unit number, and “test1c.cfp” is the name of the file with the input data for the CFP. The Name File is read by MODFLOW on unit 99, and is constructed as follows:

FOR EACH SIMULATION

1. Ftype Nunit Fname

Ftype—is the file type. Ftype may be entered in any combination of upper and lower case letters. Below are the Ftype names specific to the CFP application. All the other Ftypes in a Name File are existing MODFLOW Ftypes.

CFP for the Conduit Flow Process (required for all CFP modes)

CRCH for the Conduit Recharge Package (only used with modes 1 or 3)

COC for the Conduit Output Control Option (only used with modes 1 or 3)

Nunit—is the Fortran unit number assigned to the file (an integer). Each Ftype must have a unique Nunit integer. The Name File is read on unit 99.

Fname—is the file name, which is a character value. Pathnames may be specified as part of Fname.

An example of a Name file for using the CFP is given below. In this example, there are comment lines starting with a # symbol, and all the functionality available in the CFP is active; specifically, the CFP Input File and the CRCH and COC Packages.

```
Example Name File:
#output files
LIST 6 test1.OUT
DATA(BINARY) 50 test1.cbb
DATA(BINARY) 51 test1.hds
#input files
BAS6 1 test1.ba6
DIS 2 test1.DIS
LPF 11 test1.LPF
RCH 13 test1.rch
CRCH 14 test1.crch
PCG 15 test1.pcg
CFP 16 test1.cfp
OC 22 test1.oc
COC 23 test1.coc
```

Table 1. Conduit Flow Process compatibility with other MODFLOW processes and packages.

Process	Package	File type	Source
GLO	List	LIST	Harbaugh (2005)
	Discretization	DIS	Harbaugh (2005)
	Multiplier	MULT	Harbaugh (2005)
	Zone	ZONE	Harbaugh (2005)
GWF	Basic	BAS6	Harbaugh (2005)
	Block-Centered Flow	BCF6	Harbaugh (2005)
	Layer-Property Flow	LPF	Harbaugh (2005)
	Hydrogeologic-Unit Flow	HUF	Anderman and Hill (2000)
	Drain	DRN	Harbaugh (2005)
	Drain Return Flow	DRT	Banta (2000)
	River	RIV	Harbaugh (2005)
	Gaging Station	GAGE	Prudic and others (2004); Merrit and Konikow (2000)
	Stream Flow Routing	SFR	Prudic and others (2004)
	Lake	LAK	Merrit and Konikow (2000)
	Reservoir	RES	Fenske and others (1996)
	General-Head Boundary	GHB	Harbaugh (2005)
	Evapotranspiration	EVT	Harbaugh (2005)
	Segmented Evapotranspiration	ETS	Banta (2000)
	Well	WEL	Harbaugh (2005)
	Multi-node Well	MNW	Halford and Hanson (2002)
	Recharge	RCH	Harbaugh (2005)
	Flow and Head Boundary	FHB	Leake and Lilly (1997)
	Hydraulic Flow Barrier	HFB	Hsieh and Freckleton (1993)
	Stream	STR	Prudic (1989)
	Strongly Implicit Procedure	SIP	Harbaugh (2005)
	Preconditioned Conjugate Gradient	PCG	Harbaugh (2005)
	Output Control	OC	Harbaugh (2005)
OBS	Hydraulic Head Observation	OBS	Hill and others (2000)
	Drain Observation	DROB	Hill and others (2000)
	General-Head Boundary Observation	GBOB	Hill and others (2000)
	River Observation	RVOB	Hill and others (2000)
CFP	Conduit Flow	CFP	This report
	Conduit Direct Recharge	CRCH	This report
	Conduit Output Control	COC	This report

4.2. Conduit Flow Process (CFP) Input File

Input for the CFP is read from the file type "CFP" in the Name File. The CFP Input File contains most of the information needed to simulate dual porosity flow. For example, the location, geometry, and hydraulic properties of conduit flow pipes are held in the CFP Input File, as well as mean void diameters of preferential flow layers.

Note that the data requirements in the CFP Input File change depending upon the desired CFP Mode. The CFPM1 only requires input for conduit flow pipes and nodes, whereas the CFPM2 only requires input for preferential flow layers. The CFPM3 requires input for conduit flow pipes, nodes, and layers. When conduit flow pipes and nodes are active, it is helpful to have already mapped out the conduit network in a sketch of nodes and pipes in each model layer so that all of the nodes and pipes have assigned numbers prior to developing the CFP dataset. Additionally, the length (LENUNI) and time units (ITMUNI) in the Discretization (DIS) File of MODFLOW-2005 are used within CFP to calculate conduit flow terms, such as gravitational acceleration (g) and viscosity (μ). Thus, if using units of feet and day in the MODFLOW laminar flow model datasets, then units used in the CFP Input File also must be entered in feet and day.

For each simulation, all variables, characters, and arrays in the CFP Input File are read using free format. The required comment lines are dimensioned for 80 characters.

0. #Required comment line
1. MODE (read 1 line)
2. #Required comment line
3. #Required comment line
4. NNODES NPIPES NLAYERS (read 1 line)
5. #Required comment line
6. TEMPERATURE (read 1 line)
7. # Required comment line
8. NO_N MC MR ML NB1 NB2 NB3 NB4 NB5 NB6 PB1 PB2 PB3 PB4 PB5 PB6 (read in a total of NNODES lines)
9. #Required comment line
10. #Required comment line
11. #Required comment line
12. GEOHEIGHT. (Option 1: NO_N, ELEVATION; read in a total of NNODES lines of input data) or (Option 2: NNODES, ELEVATION; 1-line)
13. #Required comment line
14. SA_EXCHANGE (read 1 integer value, either 0 or 1)
15. #Required comment line
16. EPSILON (read 1 real value)
17. #Required comment line
18. NITER (read 1 integer value)
19. #Required comment line
20. RELAX (read 1 real value)
21. #Required comment line
22. P_NR (read 1 integer value, either 0 or 1)
23. #Required comment line
24. #Required comment line
25. NO_P DIAMETER TORTUOSITY RHEIGHT LCRITREY_P TCRITREY_P (Read 1 integer and 5 real values for each pipe)
26. #Required comment line
27. NO_N N_HEAD (read in a total of NNODES lines of input data)
28. #Required comment line
29. NO_N K_EXCHANGE (read in a total of NNODES lines of input data)
30. #Required comment line
31. #Required comment line
32. NCL (one integer value)
33. #Required comment line
34. CL (one line read, number of values equal to NCL)
35. #Required comment line
36. LTEMP (one real value)
37. #Required comment line
38. #Required comment line
39. VOID LCRITREY_L TCRITREY_L (read one line for each NCL layer)

The description of CFP data input variables is given below:

0. The first line of the input file is a required text comment line. A blank line could be inserted, but it is best to put some sort of run documentation in this line. For example “#Test Run 1 of Shoemaker and Kuniansky for Wakulla Springs.” Below is a detailed description of all of the numbered data types in the CFP Input File.

1. MODE—is an integer value controlling the activation of conduit pipes and (or) layers.

   If MODE=1, only conduit pipes are active (CFPM1).

   If MODE=2, only conduit layers are active (CFPM2).

   If MODE=3, both conduit pipes and layers are active (CFPM3).

   If MODE=1, items 0–29 have to be specified. If MODE=2, only items 0, 1, and 30–39 have to be specified. If MODE=3, items 0–39 have to be specified.

2–3. Two comment lines. These are required text comment lines for further description of the model run.

4. NNODES NPIPES NLAYERS

   NNODES—is an integer value for the total number of nodes in the conduit pipe network. Each node is located at the center of a model cell in plan view.

   NPIPES—is an integer value for the total number of pipes in the conduit network.

   NLAYERS—is an integer value for the total number of model layers.

5. One required text comment line.

6. TEMPERATURE—is a real number in degrees Celsius, representing the average temperature of ground water in the conduit pipes.

7. One required text comment line.

8. NO_N MC MR ML NB1 NB2 NB3 NB4 NB5 NB6 PB1 PB2 PB3 PB4 PB5 PB6—are integer values that describe how the nodes are connected to the MODFLOW model cells, and how node and pipe connections are formed. There should be one line for each node. It is best to have developed a sketch of the model grid with the pipes and nodes numbered before starting this dataset.

   NO_N (column 1)—is the node number. Nodes are at the center of a model cell and define the connections of the pipe network.

   MC, MR, and ML (columns 2 to 4)—are the MODFLOW cell column, row, and layer numbers within which node NO_N is located, respectively.

   NB1 to NB6 (columns 5 to 10)—are neighbor node numbers, which are connected by pipes to node NO_N (specified in the first column). As many as six adjacent or diagonal neighboring nodes in three dimensions can be listed. If there are less than six neighbor nodes connected, insert zeros for the remaining columns. There will always be at least one node connected to the specified node NO_N.

   PB1 to PB6 (columns 11 to 16)—are the pipe numbers connected to the node NO_N. Each node should be connected to at least 1 pipe, but can be connected to as many as 6 pipes. Pipe numbers represent the pipe segment between nodes. If there are less than six neighboring pipes connected to the node (NO_N), insert zeros in the remaining columns. Again, a zero in the rest of the columns means there are no more pipe connections.

9–11. Required comment lines used to describe model input data.

12. GEOHEIGHT—is the absolute elevation of the pipe nodes. There are two options for entering node elevations. With option 1 the number of lines read is equal to the total number of nodes (NNODES data line 4). With option 2 only 1 line is read and the first number must equal NNODES in data line 4.

   Option 1—NO_N ELEVATION— Each line will contain an integer node number (NO_N) and the respective node elevation (ELEVATION) with respect to the user’s model datum. A CFPM1 simulation with 10 nodes, for example, would require 10 lines, each containing an integer node number and the respective node elevation. If the first node number is less than the total number of nodes, this indicates that option 1 has been selected and the CFP program will expect to read in a total number of lines of data equal to the total number of nodes as specified in data line 4.

   Option 2—NNODES ELEVATION— only one line is read describing the elevation of each pipe node within finite-difference cells. If the first integer in this section of input equals the total number of nodes as specified in data line 4, then only one line will be read. This option allows for the CFPM1 to set each node elevation to the vertical center of the encompassing model cell, if ELEVATION is assigned a value of 0.0. Assigning values other than 0.0 for ELEVATION will raise or lower the node elevations a distance equal to ELEVATION above or below the vertical center of the encompassing model cell.

13. Required comment lines used to describe model input data.

14. SA_EXCHANGE—is an integer that equals either 0 or 1

If SA_EXCHANGE = 0, the user assigns the pipe conductance for each node in the CFP Input File.

If SA_EXCHANGE = 1, the user assigns the conduit wall permeability, and the CFP will compute the surface areas of pipes when assembling pipe conductances for ground-water exchange between pipes and porous media.

15. Required comment lines used to describe model input data.

16. EPSILON—is a real number for the convergence criterion of the Newton-Raphson iteration for pipe flow equations. Use a very small number, such as 0.000001.

17. Required comment lines used to describe model input data.

18. NITER—is an integer number for the maximum number of Newton-Raphson iterations. If convergence cannot be achieved, the program will stop and a warning will be printed in the MODFLOW listing file.

19. Required comment lines used to describe model input data.

20. RELAX—is a real number of relaxation that determines the step length of the Newton-Raphson iterations. Changing RELAX to a value slightly less than 1.0 may facilitate convergence of the pipe flow equations.

21. Required comment lines used to describe model input data.

22. P_NR—is an integer print flag for Newton Raphson iterations. If P_NR equals 0, results from these iterations are not printed. If P_NR equals 1, results from these iterations are printed, including the MODFLOW iteration number, Newton Raphson iteration number, and the maximum node head change.

23–24. Two required comment lines used to describe model input data.

25. NO_P DIAMETER TORTUOSITY RHEIGHT LCRITREY_P TCRITREY_P—are the pipe numbers and hydraulic properties. The total lines of input data are equal to the total number of pipes defined by NPIPES in data line 4.

    NO_P (column 1)—is the integer pipe number.

    DIAMETER (column 2)—is a real number for the pipe diameter.

    TORTUOSITY (column 3)—is a real number for the pipe tortuosity. A value of 1.0 indicates a straight pipe. If a pipe length greater than the straight-line distance between the two nodes of the pipe is required, a tortuosity greater than 1 can be used.

    RHEIGHT (column 4)—is a real number for the internal calculation of roughness and is the mean height of the micro-topography of the conduit (pipe) wall.

    LCRITREY_P (column 5)—is the lower critical Reynolds number (turbulent to laminar).

    TCRITREY_P (column 6)—is the upper critical Reynolds number (laminar to turbulent).

26. Required comment lines used to describe model input data.

27. NO_N N_HEAD—are integer node numbers and either node constant heads or a flag that activates a CFP solution for node head. Use one line for each node.

    NO_N (column 1)—is the integer node number.

    N_HEAD (column 2)—is the piezometric heads for nodes with fixed head (positive values) or a flag (-1) that indicates that heads will be calculated during simulation.

28. Required comment lines used to describe model input data.

29. NO_N K_EXCHANGE—are integer node numbers and real numbers for either conduit wall permeabilities (when SA_EXCHANGE=1) or pipe conductances (when SA_EXCHANGE=0), one line for each node.

    NO_N (column 1)—is the node number (in increasing order).

    K_EXCHANGE (column 2)—is the pipe conductance (SA_EXCHANGE=0) or conduit wall permeability (SA_EXCHANGE=1).

30–31. Two required text comment lines read only if MODE equals 2 or 3 from data line 1. The following variables are required in the CFP Input File for conduit layers. It is important to remember that when MODE equals 2, the following variables are the only variables needed in the CFP Input File. Also, conduit layers must be convertible layers. For example, conduit layers must be specified as LAYCON = 3 in the BCF Package, LAYTYPE > 0 in the LPF Package, or LTHUF > 0 in the HUF Package.

32. NCL—is an integer equal to the total number of conduit layers.

33. Required comment lines used to describe model input data.

34. CL—is a one-dimensional integer array entered on a single line of the CFP Input File. This array holds the MODFLOW layer numbers that are conduit layers.

35. Required comment line used to describe model input data.

36. LTEMP—is the mean water temperature in degrees Celsius of all conduit layers.

37. Required text comment line.

38. Required text comment line

39. VOID LCRITREY_L TCRITREY_L—are real numbers for each conduit flow layer.

    VOID—is the mean void diameter. This value is used in calculating the critical head difference from the specified lower and upper critical Reynolds numbers.

    LCRITREY_L—is the lower critical Reynolds number for switching from turbulent to laminar flow.

    TCRITREY_L—is the upper critical Reynolds number for transitioning from laminar to turbulent flow.

Repeat lines 38 and 39 for each conduit flow layer.

4.3. Conduit Output Control (COC) File

Many results from CFP calculations are written to the MOFLOW-2005 Listing File. Some results not written to the Listing File can be output to separate files for post processing and error checking. The COC File is used to specify the output desired in addition to the output written to the Listing File. For example, node heads and flow terms can be written to separate files for post processing, such as plotting and calculations. Pipe flow rates and Reynolds numbers also can be written to separate files for post processing. The COC File specifies the pipe and node numbers for output as well as the output interval. This file is only used for CFP Modes 1 and 3.

For each simulation, all variables, characters, and arrays in the COC Input file are read using free format.

0. #Required comment line
1. #Required comment line
2. NNODES (one integer value, one line)
3. #Required comment line
4. NODE_NUMBERS (one integer value per line, multiple lines equal to NNODES in data line 2).
5. #Required comment line
6. N_NTS
7. #Required comment line
8. NPIPES
9. #Required comment line
10. PIPE_NUMBERS

    Item 10 are pipe numbers for output to files other than the Listing File. List one pipe number per line.

11. #Required comment line
12. T_NTS

The description of COC data input variables is given below:

0, 1, 3, 5, 7, 9, 11—are required comment lines.

2. NNODES—is an integer number equal to the number of nodes for which flow and head values are desired in separate output files.

4. NODE_NUMBERS—are integer values of the node numbers for which flow and head values are desired in separate output files. List one node number per line.

6. N_NTS—is an integer value equal to the time step interval for output of node head and flow values. As an example, N_NTS equal to 2 activates node output at each NODE_NUMBERS every 2 time steps.

8. NPIPES—is an integer value equal to the number of pipes for which flow rates and Reynolds numbers are desired in separate output files.

10. PIPE_NUMBERS—are integer values of the pipe numbers for which flow and head values are desired in separate output files. List one pipe number per line.

12. T_NTS—is an integer value equal to the time step interval for output of pipe flow rates and Reynolds numbers. As an example, T_NTS equal to 2 activates node output at each PIPE_NUMBERS every 2 time steps.

4.4. Conduit Recharge (CRCH) Package

The CRCH Package is a mechanism for routing a fraction of the diffuse areal recharge (assigned in the traditional MODFLOW RCH Package) into nodes of conduit pipes. This functionality may be useful in scenarios where rainfall runs directly into karst features, such as sinkholes or swallets. If activated for CFPM1 or CFPM3 simulations, the CRCH Package can route a fraction or all of the diffuse areal recharge (entered in the standard MODFLOW-2005 RCH package) into a specified conduit node. The routed direct recharge will be subtracted from the diffuse recharge for MODFLOW water-budget calculations. Note the RCH Package must be active when the CRCH Package is active. This file is only used for CFP Modes 1 and 3.

For each simulation and stress period, all variables, characters, and arrays in the CRCH Package are read using free format.

0. #Required comment line
1. IFLAG_CRCH
2. NODE_NUMBERS P_CRCH

The description of CRCH data input variables is given below:

0. #Required comment line

1. IFLAG_CRCH—is an integer value that activates or deactivates the reading of CRCH data.

   If IFLAG_CRCH is not equal to -1, NODE_NUMBERS and P_CRCH values are read for the total number of nodes (NNODES) in the simulation. Each node must be listed with NODE_NUMBERS and P_CRCH values.

   If IFLAG_CRCH equals -1, NODE_NUMBERS and P_CRCH from the last stress period are used for the current stress period.

2. NODE_NUMBERS—is an integer value indicating the node number. P_CRCH is a real number equal to a fraction of diffuse areal recharge (entered in the MODFLOW-2005 RCH Package) partitioned directly into the conduit node NODE_NUMBERS. If the user, for example, wants the direct conduit recharge to equal the diffuse recharge rate assigned for the model cell in which the pipe is located, the user would enter a value of 1.0 for P_CRCH. In this case, the diffuse areal recharge for the model cell would equal zero in MODFLOW water-budget calculations.

Chapter 5. Guidance on Assignment of Conduit Flow Process (CFP) Parameters

Simulating turbulent flow with the CFP requires more data than traditional MODFLOW simulations. For CFPM1, additional data requirements include the conduit (pipe) locations, elevations, lengths, and connections of subsurface voids conceptualized as conduit flow pipes. Also required are the temperature of ground water, conduit diameters, tortuosity, roughness, and lower and upper critical Reynolds numbers. Finally, pipe conductances must be computed for the exchange of water between the MODFLOW cell and the pipe network node. When CFPM1 internally computes pipe conductances, the additional required data are conduit wall permeability terms. For CFPM2, additional data requirements include the temperature of ground water in the preferential flow layers, mean void diameters, and lower and upper critical Reynolds numbers. Assigning values for these data requirements can be challenging. Additionally, one cannot use a quasi-three-dimensional modeling approach for CFPM2 preferential flow layers because the top and bottom of the preferential flow layer must be assigned in the DIS Package for use with the BCF, LPF, or HUF Packages.

5.1. Parameter Guidance for the Conduit Flow Process Mode 1 (CFPM1)

Subsurface caverns, such as Wakulla Springs in northern Florida, Mammoth Cave in Kentucky, and Jewel and Wind Caves in the Black Hills of South Dakota, are commonly mapped in National or State Parks. These maps could be used to assign CFPM1 conduit pipe locations, elevations, lengths, and inter-connections. Ground-water temperature is a relatively easy quantity to measure using water temperature probes. Maps of subsurface caverns also could be used to assign pipe diameters and tortuosity. Measurements of pipe roughness can be obtained by sampling and averaging the height of the conduit wall microtopography. The height of the mean microtopography is used to calculate the roughness. As stated earlier, the tortuosity is set to adjust the straight-line length between the nodes as defined from the centroid of the MODFLOW cells to the total length of the actual cave or submerged cave passage. As discussed below, N_{Re} and mean conduit wall permeability terms (K) are the most difficult values to assign for CFPM1 simulations because both can only be known through controlled laboratory or field measurements.

The typical range in the lower N_{Re} for determining when flow in a pipe becomes turbulent is 2,000 to 4,000 (Vennard and Street, 1975, chap. 7–9). What is known from numerous studies of circular and other pipe shapes is that the N_{Re} decreases as the: (1) pipe walls become rougher, (2) pipe or conduit is more tortuous, and (3) more abrupt changes in diameter or shape occur between conduit passages. Because natural cave conduits are very rough and of irregular shape, the lower N_{Re} for determining turbulent flow could be at the lower range of the value for pipes (Dreybrodt, 1988). The N_{Re} for natural conduits could be set to at least an order of magnitude higher than that of porous media; specifically, greater than 1,000 if the conduits are pipe like (not small interconnected vugs as shown in fig. 6B). Unfortunately, the exact value of the N_{Re} used will be uncertain. The previous versions of CAVE set the lower N_{Re} to about 2,000 for large conduits. The upper N_{Re} can be experimented with during calibration. Upper N_{Re} for circular and smooth pipes can be as high as 12,000 (Vennard and Street, 1975, p. 301).

The order of magnitude for the mean conduit wall permeability term likely is determined by the hydraulic conductivity of the geologic material encompassing CFPM1 pipes (that is, the rock matrix, not the conduits). More complicated formulations for pipe conductance can be derived; for example, Bauer and others (2000; 2003) concluded that the pipe conductance should be a function of: (1) the hydraulic conductivity of the lower permeability diffuse-flow rock at the location of the cell, (2) the surface area of the conduits within the cell, and (3) a geometry factor representing the fracture spacing in the rock adjacent to the conduit in the cell. The pipe conductance limits the exchange of water and can effectively decouple the laminar flow model from the pipe network when values are assigned close to zero. In these cases, the exchanged water is much smaller than the volume of water in the conduit system, and thus, little influence is found when varying pipe conductance. If the pipe conductance is increased, then the exchanged water becomes similar to the volume of water in the pipe network and the system becomes sensitive to the pipe conductance. For values of the conduit wall permeability term similar to the hydraulic conductivity of the porous media, the flow resistance in the laminar flow model limits exchange between the two systems and exchange flow may not be sensitive to the exchange coefficient. Bauer and others (2003, fig. 8) presented these findings and also reported convergence problems for large values of the pipe conductance. This potentially results from the fact that very large conductance values tend to force the head in MODFLOW to equal the head in the pipe and allow a large exchange volume.

5.2. Parameter Guidance for the Conduit Flow Process Mode 2 (CFPM2)

For CFPM2, additional data for turbulent simulations include the top and bottom elevations of preferential flow layers specified as part of the MODFLOW-2005 discretization file input, as well as the temperature of ground water, mean void diameters, and lower and upper critical Reynolds numbers (N_{Re}) for all the preferential flow layers. The laminar horizontal hydraulic conductivities, K_{lam}, also need to be assigned in the BFC, HUF, of LPF packages. Top and bottom elevations for preferential flow layers can be determined using geologic mapping, well lithology data, and borehole and (or) surface geophysics. Again, the temperature of ground water should be relatively easy to quantify using water temperature probes. Mean void diameters could be assigned based on visual inspection of geological cores or images of the borehole. The upper and lower critical Reynolds numbers and laminar horizontal hydraulic conductivity, K_{lam}, are the most difficult properties to assign for CFPM2 simulations.

Values for upper and lower critical Reynolds numbers and laminar horizontal hydraulic conductivities depend on the geology of each preferential flow zone. When borehole flow meter logging is used in conjunction with traditional aquifer tests, an estimate of laminar hydraulic conductivity can be obtained for the preferential flow layers (Paillet, 1998; Reese and Cunningham, 2000; Cunningham and others, 2006). These data can be used as a starting point for estimating K_{lam}. For the gray limestone aquifer in southern Florida, Reese and Cunningham (2000) estimated K_{lam} as large as 3,700 to 15,000 m/d in preferential flow zones of 1.5 to 3 m thick. If the thickness of the preferential flow zone is determined from flow meter logs, then the estimate of K_{lam} for that preferential layer can be estimated as described in Paillet (1998).

If the CFPM2 preferential flow zone is similar to that of the Biscayne aquifer (fig. 6B), then upper and lower critical Reynolds numbers could be closer to that for porous media (perhaps greater than 50 but probably less than 1,000 and would have a high degree of uncertainty). Until laboratory experimentation with physical models of the secondary vuggy porosity of the Biscayne aquifer can be accomplished, the lower and upper critical Reynolds numbers will remain uncertain model parameters. If the preferential flow layer resembles a pipe, then the lower N_{Re} would be closer to values of 2,000 for a very rough pipe.

The laminar hydraulic conductivity, K_{lam}, of a circular conduit can be estimated from the resistance terms in the Darcy-Weisbach as:

$$K_{lam} = \frac{gd^2}{32\nu}, \tag{51}$$

where

- g is the gravitational acceleration constant,
- d is the diameter of the conduit,

and

- ν is the kinematic viscosity.

With very large diameter conduits, the estimated hydraulic conductivity is millions of L/T (for example, meters per day) and may at a certain point seem unreasonably high when compared to results from traditional aquifer tests. Traditional aquifer tests, however, generally assume that ground water flows uniformly along the entire thickness of the aquifer, when in reality flow can be concentrated within preferential flow zones that are present within 10 percent or less of the entire thickness of the aquifer. The K_{lam} estimated with equation 51 will be more consistent with results from aquifer tests that assume flow is restricted to a percentage of the total thickness of the aquifer.

Extremely large changes in hydraulic conductivity may cause some numerical problems for MODFLOW convergence (Kuniansky and Danskin, 2003). Thus, the final value used in the model may be less than the original estimate. Another method to estimate the laminar horizontal hydraulic conductivity for carbonate rock aquifers with preferential layers like the Biscayne aquifer, in which interconnected vugs create the preferential flow layer, uses the average pore diameter of the vugs and the effective porosity (interconnected void space). Additionally, the average temperature should be known in order to calculate viscosity. This estimate is based on the assumption that the interconnected vugs are circular and the hydraulic conductivity at laminar flow can be computed from equation 51; that the hydraulic conductivity of the rock matrix is essentially zero; and that the vugs are evenly distributed through the rock matrix of the preferential flow layer. In a given cross section of unit width and height, the effective porosity provides the estimate of the thickness of the pipes in the preferential flow layer and one minus the porosity provides the estimate of the thickness of the rock matrix. In equation 52 below, it is assumed that the rock matrix has a hydraulic conductivity of approximately zero. The horizontal hydraulic conductivity for the layer is computed with the following simplified equation for calculation of horizontal hydraulic conductivity modified from Bouwer (1979):

$$K_{lam} = \frac{gd^2}{32\nu}\eta, \quad (52)$$

where K_{lam} is the estimated laminar hydraulic conductivity for the preferential flow layer and η is the effective porosity (percent volume of interconnected vugs). For example, K.J. Cunningham (U.S. Geological Survey, written commun., 2007) examined 240 core samples from 23 wells that penetrated preferential flow layers in the Biscayne aquifer of southern Florida, and estimated a mean vug diameter of 0.9 cm and a mean effective porosity of about 12 percent. Using equation 52, the mean laminar hydraulic conductivity equals about 200,000 m/d. This simple equation for estimating laminar hydraulic conductivity compared favorably with results from a more complex method of lattice Boltzmann modeling of fluid flow through a specimen of the Biscayne aquifer (Alvarez, 2007).

When using CFPM2, even for steady-state simulation, specify multiple time steps in the steady-state stress period. Set the head closure criteria for the solvers to less than 0.0001 to ensure that both the head converges and the F_{adj} converges. If the iterative solver does not converge initially, then try changing some of the solver parameters. For example with the pre-conditioned conjugate gradient solver package (PCG2), frequently the dampening factor must be set to 0.98 or less rather than leaving it at the default value of 1.0. Additionally, remember that preferential flow layers should not dewater, even though it is required that these be specified as convertible layers.

Chapter 6. Conduit Flow Process (CFP) Example Problems

An example problem is presented that consists of four model rows and columns, three model layers, and five stress periods. Model cells are 10 m in length and width and 1 m thick. A constant-head boundary is assigned along column 1 for all three model layers, with a value equal to 20.0 m above an arbitrary datum. No-flow boundaries exist around the outer edges of the model domain. Each layer was assigned as a convertible layer in the LPF package with constant horizontal and vertical hydraulic conductivity equal to 1,000 and 200 m/d, respectively. Specific yield and storage coefficient for each layer were assigned values of 0.2 and 0.00002, respectively. Head and velocity vectors were computed with the CFP inactive (fig. 15), so the effects of activating the CFP are clear.

6.1. Conduit Flow Process Mode 1 (CFPM1) Example Problem

For the CFPM1 example problem, conduit pipes were activated, and five nodes and four pipes (10 m in length) were designed with the geometry and connections presented in figure 16. Mean pipe water temperature was assigned a value of 25°C. The elevations of the pipes were assigned as 4.5 m above the datum, and SA_EXCHANGE was set to 1 so the CFP would use the surface area of pipes in cells to compute the pipe conductance. Solution controls for the pipe flow equations included a value of 0.000001 for convergence criterion (EPSILON), 100 maximum iterations (NITER), and the relaxation parameter (RELAX) equal to 1.0. Each pipe diameter, tortuosity, and roughness heights were set to constant values of 0.1, 1.0, and 0.01 m, respectively. The critical Reynolds numbers (N_{Re}) for each pipe were assigned constant values equal to 10.0 for LCRITREY_P and 20.0 for TCRITREY_P. These values may be lower than generally expected for straight, nontortuous, and rough pipes; nevertheless, they were chosen so that both laminar and turbulent pipe flow equations could be demonstrated using this example problem. A constant node head equal to 20.0 m above the datum was assigned for node 2, and a solution was desired for the remaining node heads. Conduit wall permeabilities were assigned values of 5.0 m/d. The transient model boundary is net recharge. Values of 0.003, 0.001, 0.002, 0.003, 0.002 m/d were assigned for net recharge to the uppermost active model layer for stress periods 1 through 5, respectively, using the RCH Package. The CRCH Package was employed, which routed all of the net recharge assigned to MODFLOW cell 3,3,1 (j,i,k) into conduit node 4.

6.2. Conduit Flow Process Mode 1 (CFPM1) Input Files for Example Problem

The example CFP input files include the CFP Input File, CRCH Package, and COC File. The example CFPM1 input files and the results plotted for the CFPM1 example problem are presented herein.

6.2.1. Example CFPM1 Input File

```
# mode
1
#data for mode 1 conduit pipe system
#number of nodes / pipes / layers
5 4 3
#mean ground-water temperature in pipes
25.
#No mc mr ml Nb1 Nb2 Nb3 Nb4 Nb5 Nb6 pb1 pb2 pb3 pb4 pb5 pb6
 1  2  2  1  3   0   0   0   0   0   1   0   0   0   0   0
 2  1  3  1  3   0   0   0   0   0   2   0   0   0   0   0
 3  2  3  1  1   2   4   0   0   0   1   2   3   0   0   0
 4  3  3  1  3   5   0   0   0   0   3   4   0   0   0   0
 5  4  3  1  4   0   0   0   0   0   4   0   0   0   0   0
#node elevations, two possibilities
#1. node #, elevation (1 line for each node)
#2. number of nodes, distance from vertical centroid (only one line used to assign constant value)
```

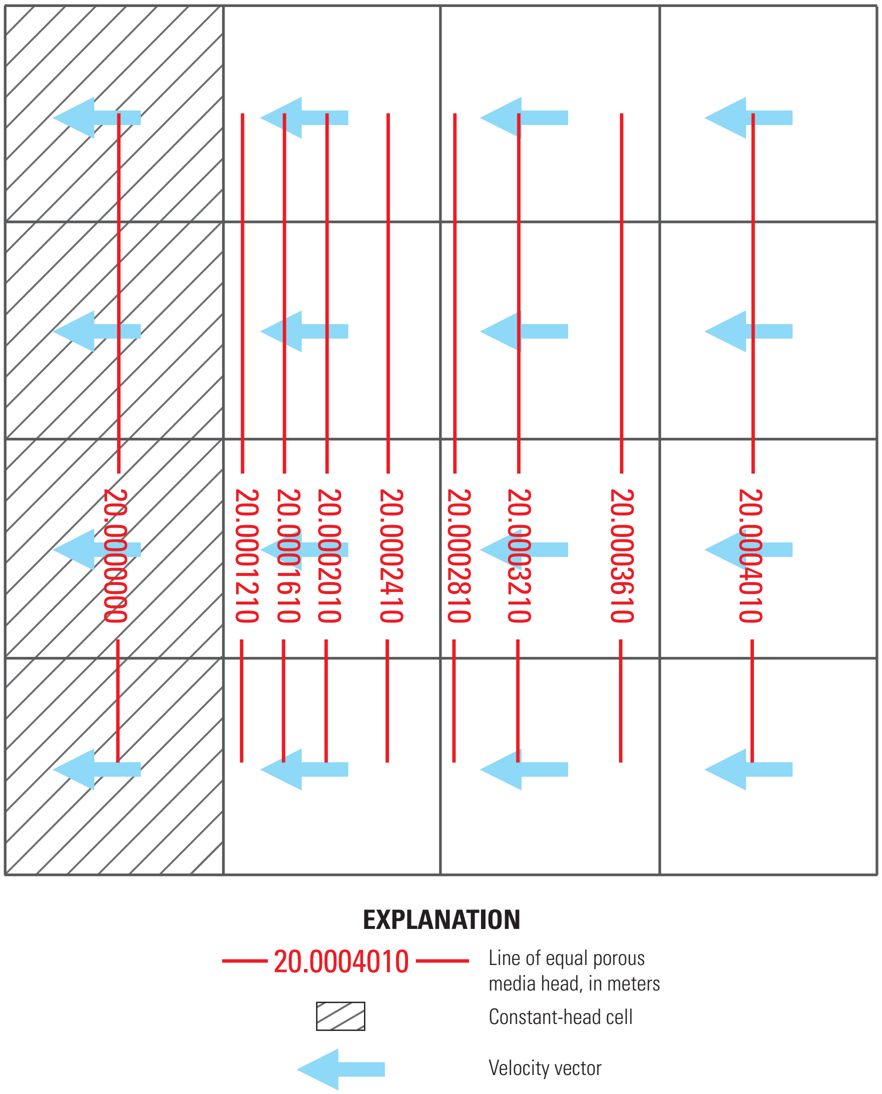


Figure 15. Heads and velocity vectors for example problem without the Conduit Flow Process.

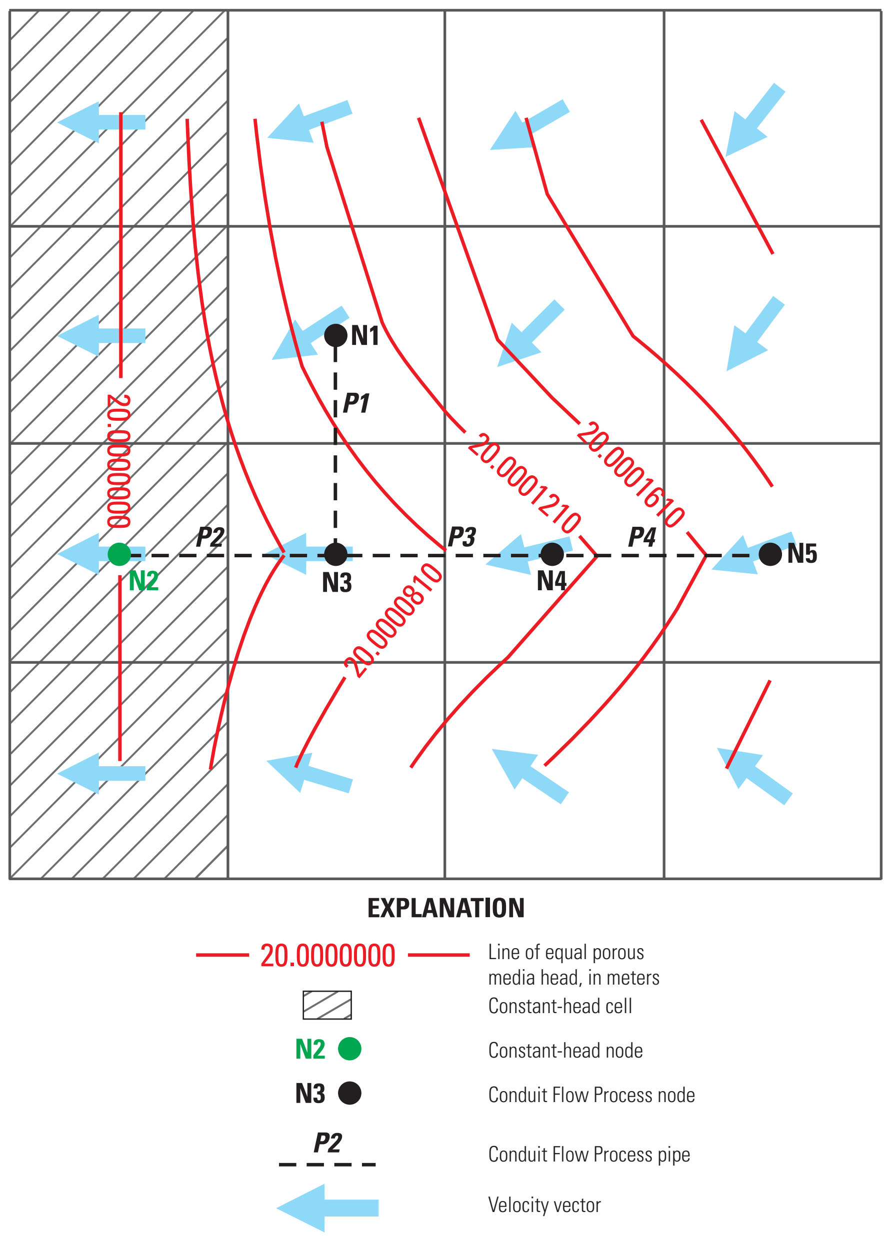


Figure 16. Heads and velocity vectors for example problem with CFPM1 active.

```
1 4.5
2 4.5
3 4.5
4 4.5
5 4.5
#SA_EXCHANGE, CFP pipe conductance (set 1) or user
computes pipe conductance (set 0)
1
#criterion for convergence
1.0D-6
#maximum number of newton raphson iterations
100
#parameter of relaxation
1.0
#newton raphson print flag
1
#pipe parameters:
#no.  diameter  tortuosity  roughness  lReynolds  tReynolds
  1     0.1        1.0        0.01       10.0       20.0
  2     0.1        1.0        0.01       10.0       20.0
  3     0.1        1.0        0.01       10.0       20.0
  4     0.1        1.0        0.01       10.0       20.0
#node heads (if head unequal -1, then head is fixed)
  1     -1
  2     20.0
  3     -1
  4     -1
  5     -1
#pipe conductance (SA_EXCHANGE=0) or conduit wall
exchange term (SA_EXCHANGE=1).
  1     5.0
  2     5.0
  3     5.0
  4     5.0
  5     5.0
```

6.2.2. Example CFPM1 Conduit Recharge (CRCH) Package

```
#Conduit Recharge Package
  1
   1    .00
   2    .00
   3    .00
   4    1.00 (all diffuse net recharge assigned in the RCH
        Package will be routed into node 4)
   5    .00
#recharge data for stress period:2
   -1
#recharge data for stress period:3
   -1
#recharge data for stress period:4
   -1
#recharge data for stress period:5
   -1
```

6.2.3. Example CFPM1 Conduit Output Control (COC) File

```
#Mode 1 time series output
#Number of nodes for output
5
#Node numbers, one per line
1
2
3
4
5
#Output each n time steps
1
#Number of pipes for output
4
#Pipe numbers, one per line
1
2
3
4
#Output each n time steps
1
```

6.2.4. Results for CFPM1 Example Problem

Results plotted for the CFPM1 example scenario include porous media heads and velocity vectors and volumetric water budgets, as well as pipe node heads, Reynolds numbers (R_e), pipe flow rates, and pipe water budgets. As expected, porous media heads were affected by the presence of conduit pipes; porous media heads in cells surrounding and adjacent to the conduit pipes suggest that the pipes are a drain on the aquifer system (fig. 16). An exception is pipe 1; specifically, porous media heads were less affected in the vicinity of pipe 1. Ground-water matrix exchange flow occurred between pipe 1 and the porous media; however, this matrix exchange flow was not substantial enough to considerably alter porous media heads. Experimental simulations were performed in which the pipe 1 conductance was increased. Porous media heads were more considerably affected as the pipe conductance was increased.

Porous media water budgets also were affected by the conduit pipes; specifically, about 5 percent of the total recharge entering the porous media was drained into the conduit pipes (table 2). The remaining net recharge that infiltrates the porous media exited the constant-head boundary on column 1. Greater amounts of net recharge could be routed into the conduit pipes by increasing conduit wall permeability.

Minor spatial variability existed in the CFPM1 node head solution (table 3). With the exception of node 2, surrounding porous media heads were always greater than pipe node heads, forcing matrix exchange flow from the porous media system into the conduit pipe system. Note that in cases where node heads are greater than heads in the surrounding porous media, matrix exchange flow would be forced from the conduit pipes into the porous media system.

Table 2. Volumetric budget for the Conduit Flow Process Mode 1 (CFPM1) example problem.

[End of time step 10, stress period 5. Percent discrepancy is 0.0]

Parameter	Cumulative volumes (L^3)	Parameter	Time-step rate (L^3/T)
IN			
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	0.0000	CONSTANT HEAD	0.0000
RECHARGE	1,210.0000	RECHARGE	2.2000
PIPES	0.0000	PIPES	0.0000
TOTAL IN	1,210.0000	TOTAL IN	2.2000
OUT			
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	1,142.4295	CONSTANT HEAD	2.0771
RECHARGE	0.0000	RECHARGE	0.0000
PIPES	67.5670	PIPES	0.1229
TOTAL OUT	1,210.000	TOTAL OUT	2.2000
IN-OUT	0.0000	IN-OUT	0.0000

In stress period 5, flow in conduit pipes 1 to 4 ranged from about 0.014 to -0.32 m^3/d (table 4). Positive pipe flow indicates the head difference between node *in* and the connecting node is positive, while negative pipe flow indicates the head difference between node *in* and the connecting node is negative. Specifically, the head difference between nodes 1 and 3 positive, creating positive flow in pipe 1 from node 1 to node 3 (fig. 16). The head difference between nodes 2 and 3 is negative, creating negative pipe flow in pipe 2 from node 3 to node 2 (fig. 16). Flow in conduit pipes 2 and 3 was turbulent, indicating that the Darcy-Weisbach pipe flow equation was employed. Reynolds numbers for pipes 1 to 4 ranged from about 2 to 53. Note the volumetric flow rate in pipe 2 is -0.32287 m^3/d (table 4). Pipe 2 is the final drainage location for all the matrix exchange (table 3) and net recharge routed directly into node 4. Specifically, the total matrix exchange rate is about 0.12 m^3/d (table 3) and net recharge routed directly into node 4 equals 0.2 m^3/d (table 3). The sum of the total matrix exchange and direct recharge equals the volumetric flow rate in pipe 2, specifically, about 0.32 m^3/d.

Pipe water budgets also were computed with CFPM1 simulations (table 5). For this example case, about 60 percent of the ground water discharging at the pipe constant-node boundary was recharge routed directly into node 4, whereas the remaining 40 percent was diffuse matrix exchange from the porous media system (MATRIX EXCHANGE). Finally, node water budgets were computed with CFPM1 simulations. These water budgets identify the relative magnitudes of sources and sinks of water for each node in the CFPM1 example simulation (table 6).

Table 3. Node head and ground-water exchange for the Conduit Flow Process Mode 1 (CFPM1) example problem.

[Stress period 5]

Node number	Pipe head (meters)	Cell head (meters)	Matrix exchange (cubic meters per day)	Direct conduit recharge (cubic meters per day)
1	20.00000	20.00018	-0.0140	0.000000
2	20.00000 (fixed)	20.00000	0.0000	0.000000
3	20.00000	20.00017	-0.0401	0.000000
4	20.00000	20.00027	-0.0417	0.2000000
5	20.00000	20.00034	-0.027	0.000000

Table 4. Pipe flow and Reynolds numbers for the Conduit Flow Process Mode 1 (CFPM1) example problem.

[Flow in pipe 1 was laminar; flow in pipes 2 to 4 was turbulent; stress period 5]

Pipe number	Volumetric flow rate (cubic meters per day)	Reynolds number
1	0.014	2.317
2	-0.323	53.27
3	-0.2687	44.33
4	-0.027	4.46

Table 5. Water budget for the conduit pipes.

[End of time step 10, in stress period 5. Total simulation time is 500 seconds]

Parameter	Cumulative volumes (L^3)	Parameter	Time-step rate (L^3/T)
		IN	
CONSTANT HEAD	0.0000	CONSTANT HEAD	0.0000
PIPE RECHARGE	110.00	PIPE RECHARGE	0.2000
MATRIX EXCHANGE	67.5669	MATRIX EXCHANGE	0.1228
STORAGE	0.0000	STORAGE	0.0000
		TOTAL IN	0.3228
		OUT	
CONSTANT HEAD	177.5669	CONSTANT HEAD	0.3229
MATRIX EXCHANGE	0.0000	MATRIX EXCHANGE	0.0000
STORAGE	0.0000	STORAGE	0.0000
		TOTAL OUT	0.3229
IN-OUT	0.0000	IN-OUT	0.0000
PERCENT ERROR	0.0000	PERCENT ERROR	0.0000

Table 6. Water budget for the conduit nodes.

[Time step 10, in stress period 5. Total simulation time is 500 seconds. Flow to the node, in cubic meters per day]

Node number	Fixed head	Recharge	Matrix exchange	Storage	Tube in	Tube out	In-out
1	0.0000000	0.0000000	0.014	0.0000000	0.0000000	-0.014	0.0000000
2	-0.3229	0.0000000	0.0000000	0.0000000	0.3229	0.0000000	0.0000000
3	0.0000000	0.0000000	0.04012	0.0000000	0.2827	-0.3229	0.0000000
4	0.0000000	0.20000000	0.04169	0.0000000	0.02701	-0.2687	0.0000000
5	0.0000000	0.0000000	0.02701	0.0000000	0.0000000	-0.27014	0.0000000

6.3. Conduit Flow Process Mode 2 (CFPM2) Example Problem

Preferential flow layers were demonstrated using CFPM2. Each CFPM2 model layer was assigned as a preferential flow layer, with mean water temperature equal to 25.0°C. Mean void diameters for each conduit layer were set to a constant value of 0.5 m, and critical Reynolds numbers (N_{Re}) for each conduit layer also were assigned constant values equal to 1.0 for LCRITREY_L and 2.0 for TCRITREY_L. These relatively small N_{Re} create conditions that demonstrate both laminar and turbulent flow calculations for this example problem. Note the CRCH Package was deactivated in the Name File, because no pipes existed in this example problem for recharge partitioning. The example CFPM2 Input File and the results plotted for the CFPM2 example problem are presented herein.

6.3.1. Example CFPM2 Input File

```
# mode
2
#data for mode 2 conduit layer system
#number of conduit layers
3
#conduit layer #'s
1 2 3
#water temperature, in degrees Celsius
25.0
#mean void diameter and lower and upper critical Reynolds numbers for conduit layers
#conduit layer 1
 0.50 1.0 2.0
#conduit layer 2
 0.50 1.0 2.0
#conduit layer 3
 0.50 1.0 2.0
```

6.3.2. Results for CFPM2 Example Problem

Plotted results for the CFPM2 example scenario include conduit layer heads, velocity vectors, and tabulated water budgets (table 7). As expected, lines of equal conduit layer head in layer 1 stress period 1 were parallel with the constant-head boundary condition assigned along column 1 (fig. 17). The net recharge assigned to the uppermost active layer mounds ground-water heads farther above the vertical datum with distance away from the constant-head boundary. Turbulence codes for CFPM2 simulations are automatically written to an output file named turblam.txt. For each conduit layer during each stress period, these codes indicate whether flows at the right and front sides of conduit layer model cells are laminar or turbulent. These codes can be color coded during post-processing to visualize the distribution of laminar and turbulent flow within preferential flow layers (fig. 17).

Turbulent conduit layer heads (fig. 17) are greater than laminar heads calculated with CFPM2 inactive (fig. 15). In column three, for example, turbulent head differences from laminar elevations are about 0.003 m. Head differences can be explained by the approach for simulating turbulent flow, which reduces laminar conductances to turbulent conductances. Specifically, when net flows to a cell are positive, reduced

Table 7. Volumetric budget for the Conduit Flow Process Mode 2 (CFPM2) example problem.

[End of time step 10, in stress period 5. Percentage discrepancy is 0.00]

Parameter	Cumulative volumes (L^3)	Parameter	Time-step rate (L^3/T)
IN			
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	0.0000	CONSTANT HEAD	0.0000
RECHARGE	1,320.0000	RECHARGE	2.4000
TOTAL IN	1,320.0002	TOTAL IN	2.4000
OUT			
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	1,320.0000	CONSTANT HEAD	2.4000
RECHARGE	0.0000	RECHARGE	0.0000
TOTAL OUT	1,320.0000	TOTAL OUT	2.4000
IN-OUT	0.0000	IN-OUT	0.0000

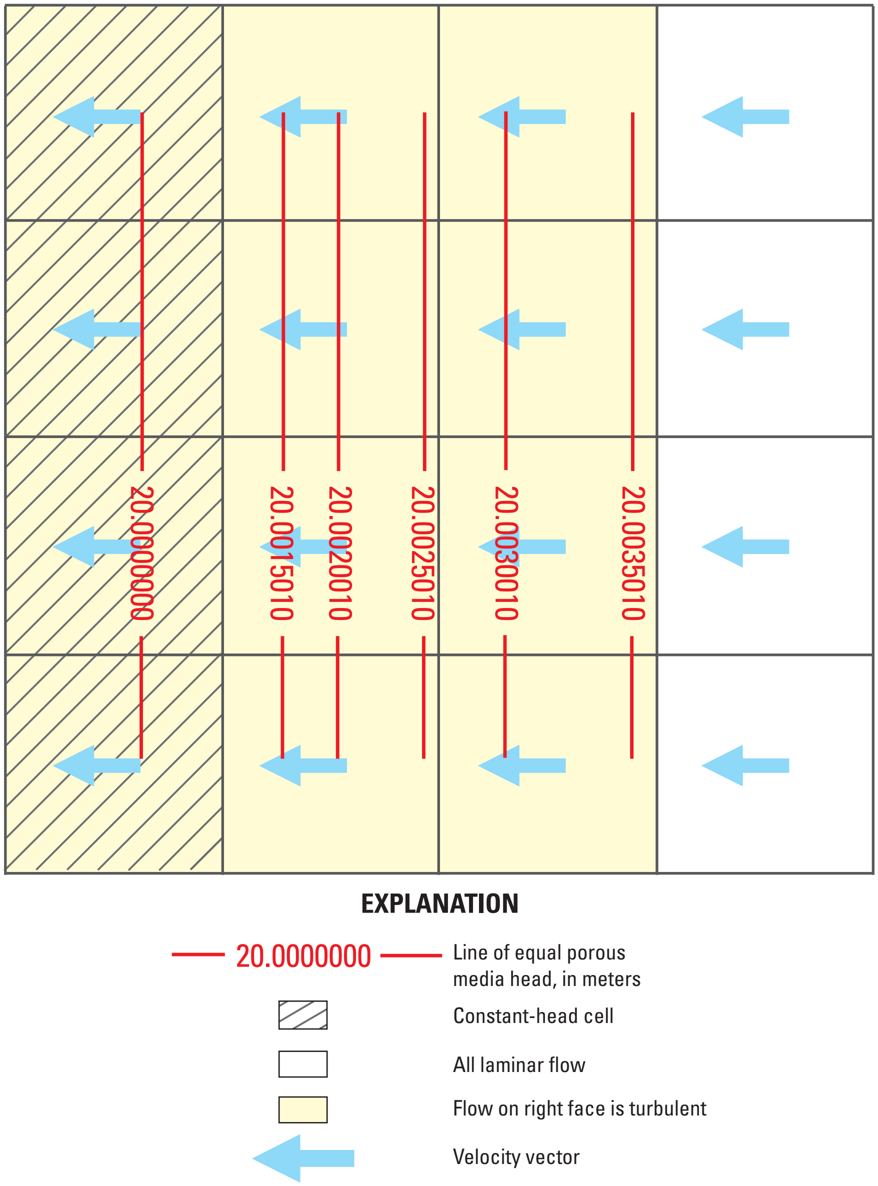


Figure 17. Heads, velocity vectors, and turbulence codes for example problem with CFPM2 active.

conductances create greater positive head differences to balance the net volume of water entering the cell. Conversely, when net flows to a cell are negative, reduced conductances create greater negative head differences to balance the net volume of water leaving the cell. Similar results were observed in experimental CFPM2 simulations of the Biscayne aquifer, where head differences from laminar elevations ranged from about -0.26 to +0.95 m.

6.4. Conduit Flow Process Mode 3 (CFPM3) Example Problem

The CFPM3 example problem combines conduit pipes and conduit layers into a single simulation. The CFP conduit pipes and layers can be assigned within independent model cells or within the same model cell. For example, conduit layers can exist in model cells that do not contain conduit pipes, and likewise, conduit pipes can exist in model cells that are not conduit layers. This example problem demonstrates a case where all conduit pipes occur in model cells that are assigned as conduit layers. The CFPM3 Input File is presented below; however, the COC File and CRCH Package are not presented because their formats are identical to the formats presented for the CFPM1 example problem. The conduit pipe and layer input data for CFPM3 are identical to the conduit pipe and layer input data for CFPM1 and CFPM2. The example CFPM3 Input File and the results plotted for the CFPM3 example problem are present herein.

6.4.1. Example CFPM3 Input File

```
# mode
3
#data for mode 1 conduit pipe system
#number of nodes / pipes / layers
5 4 3
#mean ground-water temperature in pipes
25.
#No mc mr ml Nb1 Nb2 Nb3 Nb4 Nb5 Nb6 pb1 pb2 pb3 pb4 pb5 pb6
  1  2  2  1  3   0   0   0   0   0   1   0   0   0   0   0
  2  1  3  1  3   0   0   0   0   0   2   0   0   0   0   0
  3  2  3  1  1   2   4   0   0   0   1   2   3   0   0   0
  4  3  3  1  3   5   0   0   0   0   3   4   0   0   0   0
  5  4  3  1  4   0   0   0   0   0   4   0   0   0   0   0
#node elevations, two possibilities
#1. node #, elevation (1 line for each node)
#2. number or nodes, distance from vertical centroid (only one line used to assign constant value)
1 4.5
2 4.5
3 4.5
4 4.5
5 4.5
#SA_EXCHANGE, CFP computes pipe conductance (set 1) or user computes pipe conductance (set 0)
1
#criterion for convergence
1.0D-6
#maximum number of newton raphson iterations
100
#parameter of relaxation
1.0
#newton raphson print flag
1
#pipe parameters:
#no.  diameter  tortuosity  roughness  lReynolds  tReynolds
  1     0.1       1.0         0.01       10.0       20.0
  2     0.1       1.0         0.01       10.0       20.0
  3     0.1       1.0         0.01       10.0       20.0
  4     0.1       1.0         0.01       10.0       20.0
#node heads (if head unequal -1, then head is fixed)
  1     -1
  2     20.0
  3     -1
  4     -1
  5     -1
#pipe conductance (SA_EXCHANGE=0) or conduit wall exchange term (SA_EXCHANGE=1).
  1     5.0
  2     5.0
  3     5.0
  4     5.0
  5     5.0
#data for mode 2 conduit layer system
#number of conduit layers
3
#conduit layer #'s
1 2 3
#water temperature, in degrees Celsius
25.0
#mean void diameter and critical Reynolds numbers for conduit layers
#conduit layer 1
 0.50  1.0  2.0
#conduit layer 2
 0.50  1.0  2.0
#conduit layer 3
 0.50  1.0  2.0
```

6.4.2. Results for CFPM3 Example Problem

Results for the CFPM3 example scenario include conduit layer heads, velocity vectors, turbulence codes, and volumetric water budgets as well as pipe heads, Reynolds numbers (R_e), flow rates, and pipe water budgets. Conduit layer heads were affected by the presence of conduit pipes (fig. 18). Conduit layer heads in this example are partly a function of reduced conductances along rows and columns when flow switches to turbulent.

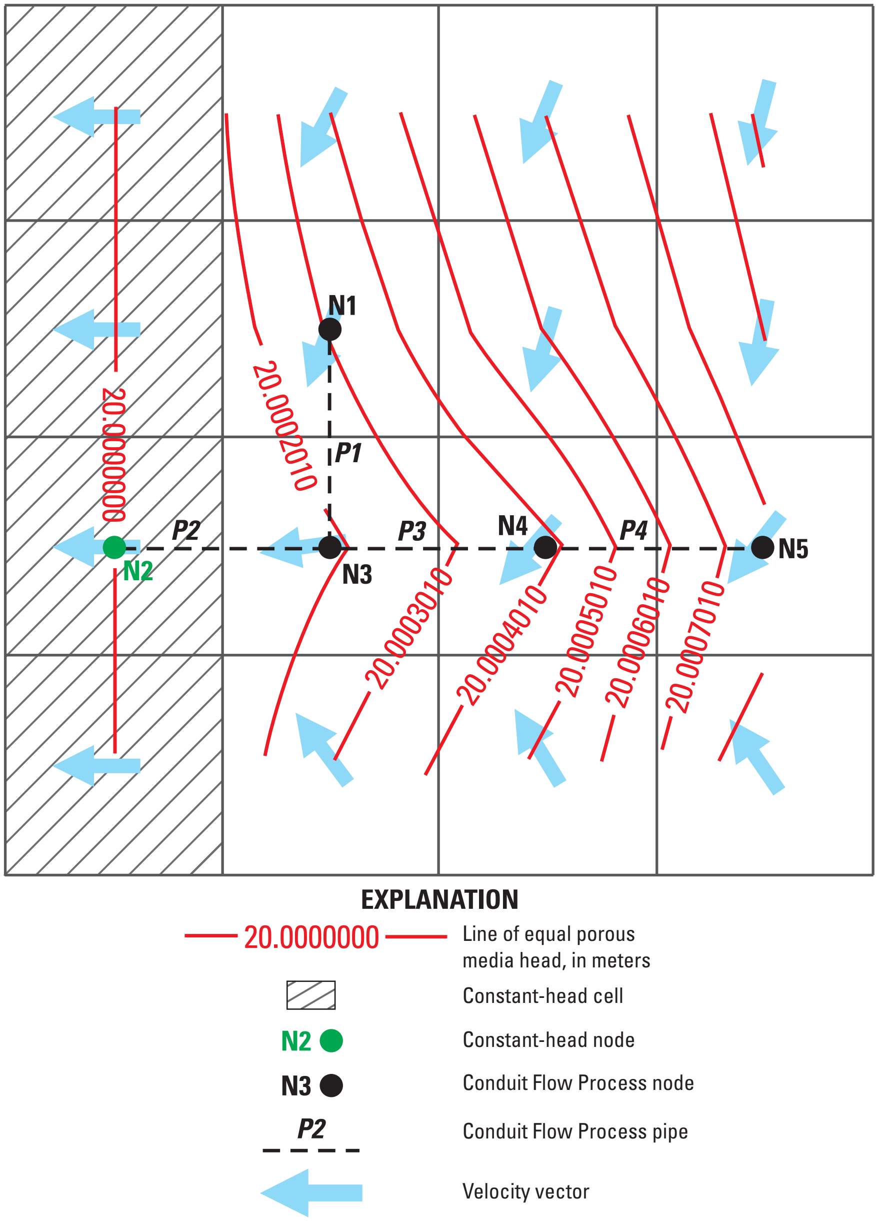


Figure 18. Heads and velocity vectors for example problem with CFPM3 active.

Conduit layer water budgets were affected by conduit pipes. Specifically, about 6 percent of the cumulative volume of recharge entering the CFPM3 simulation was drained into the conduit pipes (see cumulative values in table 8). Ultimately, this drainage exited at the constant node boundary. The 94 percent of cumulative recharge remaining in the porous media exits the constant head boundary assigned along the entire length of column one.

Minor spatial variability existed in the CFPM3 node head solution (table 9). With the exception of node 2, surrounding porous media heads were always greater than pipe node heads, forcing matrix exchange flow from the porous media into the conduit pipe system. Flow in conduit pipes 1 to 4 ranged from about 0.013 to -0.322 m^3/d (table 10). As previously mentioned, positive pipe flow indicates the head difference between node *in* and the connecting node is positive, while negative pipe flow indicates the head difference between node *in* and the connecting node is negative. Note the volumetric flow rate in pipe 2 is -0.3223 m^3/d (table 10). Pipe 2 is the final drainage location for all the matrix exchange and net recharge routed directly into node 4 (table 9). Flow in conduit pipes 2 and 3 was turbulent, indicating that the Darcy-Weisbach equation was employed. The Reynolds numbers (R_e) for pipes 1 to 4 ranged from about 2 to 53.

Table 8. Volumetric budget for the Conduit Flow Process Mode 3 (CFPM3) example problem.

[End of time step 10, in stress period 5. Percentage discrepancy is 0.00]

Parameter	Cumulative volumes (L^3)	Parameter	Time-step rate (L^3/T)
		IN	
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	0.0000	CONSTANT HEAD	0.0000
RECHARGE	1,210.0000	RECHARGE	2.2000
PIPES	0.0000	PIPES	0.0000
TOTAL IN	1,210.0000	TOTAL IN	2.2000
		OUT	
STORAGE	0.0000	STORAGE	0.0000
CONSTANT HEAD	1,134.6423	CONSTANT HEAD	2.0777
RECHARGE	0.0000	RECHARGE	0.0000
PIPES	75.352	PIPES	0.1223
TOTAL OUT	1,210.0013	TOTAL OUT	2.2000
IN-OUT	-0.0161	IN-OUT	0.0000

Table 9. Node head and ground-water exchange for the Conduit Flow Process Mode 3 (CFPM3) example problem.

Node number	Pipe head (meters)	Cell head (meters)	Matrix exchange flow (cubic meters per day)	Direct recharge (cubic meters per day)
1	20.00000	20.00018	-0.0139	0.000000
2	20.00000 (fixed)	20.00000	0.0000	0.000000
3	20.00000	20.00017	-0.0399	0.000000
4	20.00000	20.00027	-0.0415	0.200000
5	20.00000	20.00034	-0.0269	0.000000

Table 10. Pipe flow and Reynolds numbers for the Conduit Flow Process Mode 3 (CFPM3) example problem.

[Flow in pipe 1 was laminar; flow in pipes 2–4 was turbulent]

Pipe number	Volumetric flow (L^3T^{-1})	Reynolds number
1	0.014	2.3
2	-0.322	53.18
3	-0.268	44.28
4	-0.027	4.44

Pipe water budgets also were computed with CFPM3 simulations (table 11). For this example case, about 60 percent of the ground water discharging at the pipe constant-node boundary (fig. 18) was recharge routed directly into node 4, whereas the remaining 40 percent was diffuse matrix exchange (MATRIX EXCHANGE) from the porous media system (table 11). Finally, node water budgets were computed with CFPM3 simulations (table 12). These water budgets summarize the sources and sinks of water for each node in the CFPM3 example simulation.

Turbulence codes also were automatically written to an output file named turblam.txt for CFPM3 simulations. Turbulence codes for conduit layer 1 were color coded to visualize the distribution of laminar and turbulent flow (fig. 19). Flow at the right face for all cells in columns one to three was turbulent.

Table 11. Pipe water budget for the Conduit Flow Process Mode 3 (CFPM3) example problem.

[Time step 10, in stress period 5. Total simulation time is 500 seconds]

Parameter	Cumulative volumes (cubic meters)	Parameter	Time-step rate (cubic meters per day)
		IN	
CONSTANT HEAD	0.0000	CONSTANT HEAD	0.0000
PIPE RECHARGE	110.0	PIPE RECHARGE	0.2000
MATRIX EXCHANGE	75.352	MATRIX EXCHANGE	0.12
STORAGE	0.0000	STORAGE	0.0000
		TOTAL IN	0.32
		OUT	
CONSTANT HEAD	185.35	CONSTANT HEAD	0.322
MATRIX EXCHANGE	0.0000	RECHARGE	0.0000
STORAGE	0.0000	TOTAL OUT	0.322
IN-OUT	0.0000	IN-OUT	0.0000
PERCENT ERROR	0.0000	PERCENT ERROR	0.0000

Table 12. Node water budget for the Conduit Flow Process Mode 3 (CFPM3) example problem.

[Time step 10, in stress period 5. Total simulation time is 500 seconds. Flow to the node, in cubic meters per day]

Node number	Fixed head	Recharge	Matrix exchange	Storage	Tube in	Tube out	In-out
1	0.0000000	0.0000000	-0.1398	0.0000000	0.0000000	-0.01398	0.0000000
2	-0.322	0.0000000	0.0000000	0.0000000	0.322	0.0000000	0.0000000
3	0.0000000	0.0000000	-0.0399	0.0000000	0.2823	-0.322	0.0000000
4	0.0000000	0.20000000	-0.04148	0.0000000	-0.0269	-0.2684	0.0000000
5	0.0000000	0.0000000	-0.0269	0.0000000	0.0000000	-0.027	0.0000000

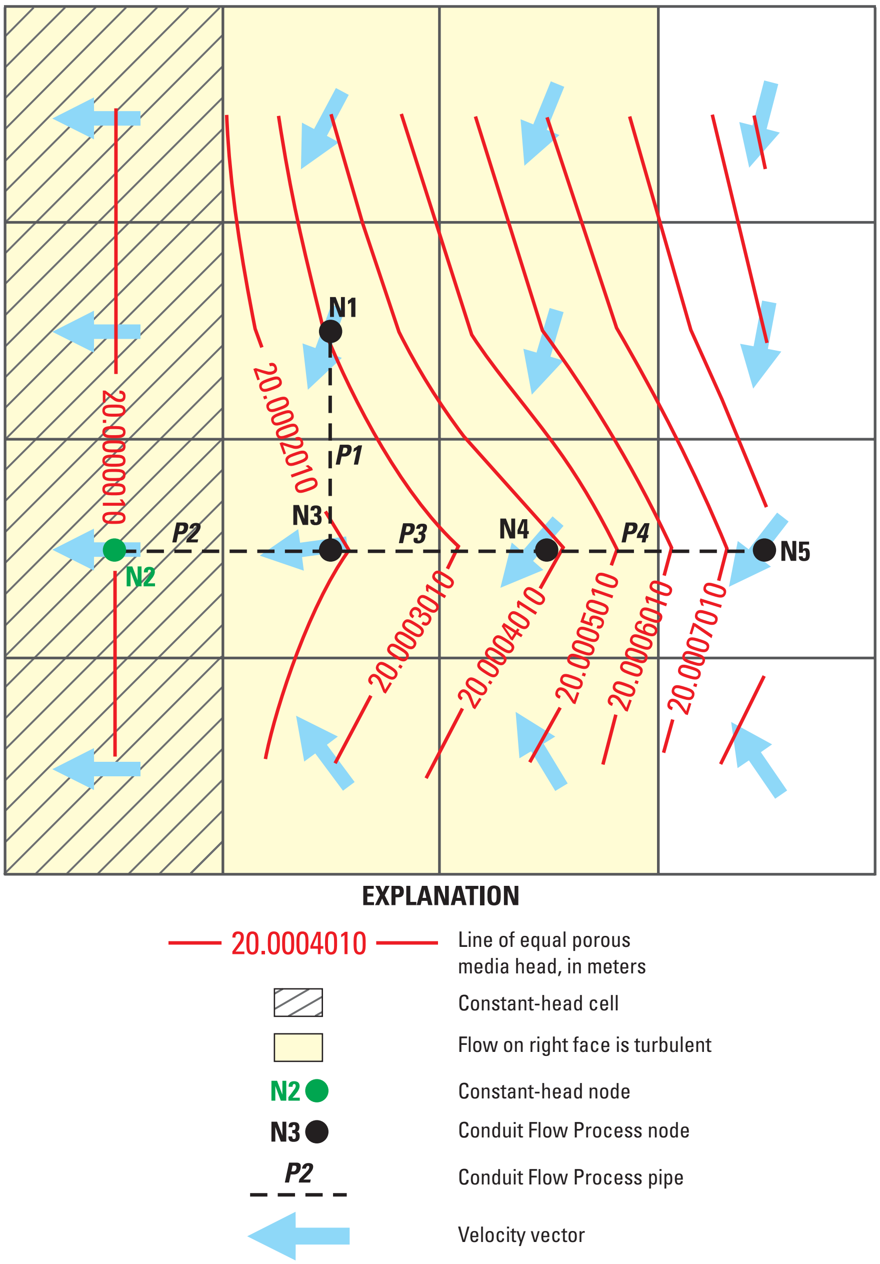


Figure 19. Heads, velocity vectors, and turbulence codes for example problem with CFPM3 active.

Chapter 7. Benchmark Testing

Because MODFLOW is a widely used computer program for ground-water flow simulation, benchmark testing was not performed for the MODFLOW parts of the CFP. The conduit pipe flow solutions were tested using simple benchmark test problems with known solutions to help verify CFP accuracy.

A simple test problem was designed to test the accuracy of CFP laminar flow calculations. This test problem consisted of a CFP pipe that was 1.0 m diameter, 10 m in length, with internal roughness height equal to 0.00001 m, and water temperature equal to 10°C. The test pipe was decoupled from the porous media system by setting the pipe conductance equal to zero. Head differences along the test pipe varied from relatively small values of -5.0×10^{-8} to $+5.0 \times 10^{-8}$ m to create laminar flow. Laminar flows computed by the CFP and the Hagen-Poiseuille equation were equal (table 13), which supports the accuracy of laminar pipe flow computations in the CFP.

Table 13. Comparison of laminar pipe flow computed with the Conduit Flow Process (CFP) and Hagan-Poiseuille equation.

[Values shown in cubic meters per second, except where noted]

Head difference (meters)	CFPM1 (laminar)	Hagen-Poiseuille
-5.00×10^{-8}	-9.20×10^{-4}	-9.20×10^{-4}
-4.00×10^{-8}	-7.36×10^{-4}	-7.36×10^{-4}
-3.00×10^{-8}	-5.52×10^{-4}	-5.52×10^{-4}
-2.00×10^{-8}	-3.68×10^{-4}	-3.68×10^{-4}
-1.00×10^{-8}	-1.84×10^{-4}	-1.84×10^{-4}
1.00×10^{-8}	1.84×10^{-4}	1.84×10^{-4}
2.00×10^{-8}	3.68×10^{-4}	3.68×10^{-4}
3.00×10^{-8}	5.52×10^{-4}	5.52×10^{-4}
4.00×10^{-8}	7.36×10^{-4}	7.36×10^{-4}
5.00×10^{-8}	9.20×10^{-4}	9.20×10^{-4}

Another test problem was designed to test CFP turbulent flow computations. This test problem also consisted of a CFP pipe that was 1.0 m diameter, 10 m in length, with internal roughness height equal to 0.00001 m, and water temperature equal to 10°C. The test pipe was decoupled from the porous media system by setting the pipe conductance equal to zero. Head differences along the test pipe varied from relatively large values of -10 to +10.1 m to create turbulent flow. Turbulent flows computed by the CFP and the Darcy-Weisbach equation were equal (table 14), which supports the accuracy of turbulent pipe flow computations in the CFP.

A final test problem was designed to test the accuracy of CFP exchange flow computations. Cases were tested for conditions in which the pipe conductance was internally computed by the CFP (SA_EXCHANGE=1). Also tested were cases when the pipe conductance is externally computed by users (SA_EXCHANGE=0). The surface area of the test pipe was computed as the product of a pipe with a diameter equal to 1 m and a length equal to 2.5 m in the encompassing MODFLOW cell, giving a surface area of about 15.71 m^2. The pipe was set as fully saturated, and the head difference between the porous media and pipe varied from -0.5 to +0.5 m. In both cases, the exchange flows computed by the CFP equaled the flows computed with the linear exchange equation (table 15), which supports the accuracy of these CFP calculations.

Table 14. Comparison of turbulent pipe flow computed with the Conduit Flow Process (CFP) and the Darcy-Weisbach equation.

[Values shown in cubic meters per second, except where noted]

Head difference (meters)	CFPM1 (turbulent)	Darcy-Weisbach
-10	-38.01	-38.01
-8	-33.93	-33.93
-6	-29.30	-29.30
-4	-23.81	-23.81
-2	-16.68	-16.68
0.1	3.48	3.48
2.1	17.10	17.10
4.1	24.12	24.12
6.1	29.55	29.55
8.1	34.14	34.14
10.1	38.20	38.20

Table 15. Comparison of exchange flows computed with the Conduit Flow Process (CFP) and the linear exchange equation.

[Values shown in cubic meters per second]

Head difference (meters)	Exchange flow user-defined pipe conductance	CFP exchange flow user-defined pipe conductance	Exchange flow CFP-defined pipe conductance	CFP exchange flow CFP-defined pipe conductance
-0.5	-5.00×10^{-5}	-5.00×10^{-5}	-7.85×10^{-4}	-7.85×10^{-4}
-0.4	-4.00×10^{-5}	-4.00×10^{-5}	-6.28×10^{-4}	-6.28×10^{-4}
-0.3	-3.00×10^{-5}	-3.00×10^{-5}	-4.71×10^{-4}	-4.71×10^{-4}
-0.2	-2.00×10^{-5}	-2.00×10^{-5}	-3.14×10^{-4}	-3.14×10^{-4}
-0.1	-1.00×10^{-5}	-1.00×10^{-5}	-1.57×10^{-4}	-1.57×10^{-4}
0.1	1.00×10^{-5}	1.00×10^{-5}	1.57×10^{-4}	1.57×10^{-4}
0.2	2.00×10^{-5}	2.00×10^{-5}	3.14×10^{-4}	3.14×10^{-4}
0.3	3.00×10^{-5}	3.00×10^{-5}	4.71×10^{-4}	4.71×10^{-4}
0.4	4.00×10^{-5}	4.00×10^{-5}	6.28×10^{-4}	6.28×10^{-4}
0.5	5.00×10^{-5}	5.00×10^{-5}	7.85×10^{-4}	7.85×10^{-4}

Chapter 8. Selected References

Adams, R., and Parkin, G., 2002, Development of a surface-groundwater-pipe network model for the sustainable management of karstic groundwater: Berlin, Environmental Geology, v. 42, no. 5, p. 513–517.

Alvarez, P.F., 2007, Lattice Boltzmann modeling of fluid flow to determine the permeability of a karst specimen: Master of Science Thesis, Miami, Florida International University, 96 p.

Banta, E.R., 2000, MODFLOW-2000, the U.S. Geological Survey modular ground-water model—Documentation of packages for simulating evapotranspiration with a segmented function (ETS1) and drains with return flow (DRT1): U.S. Geological Survey Open-File Report 00–466, 127 p.

Barenblatt, G.I., Zheltov, I.P., and Kochina, I.N., 1960, Basic concepts in the theory of seepage of homogeneous liquids in fissured rocks: Journal of Applied Mathematics and Mechanics, v. 24, 1,286–1,303 p.

Bauer, Sebastian, 2002, Simulation of the genesis of karst aquifers in carbonate rocks: Vol. 62 of Tübinger Geowissenschaftliche Arbeiten: Tübingen, Germany, Reihe C. Institut und Museum für Geologie und Paläontologie der Universität Tübingen.

Bauer, Sebastian, Liedl, Rudolf, and Sauter, Martin, 2000, Modelling of karst development considering conduit-matrix exchange flow, *in* Stauffer, F., Kinzelbach, W., Kovar, K., and Hoehn, E., eds., Calibration and reliability in groundwater modelling: Coping with uncertainty—Proceedings of the ModelCARE'99 Conference: International Association of Hydrological Sciences Publication 265, p. 10–15.

Bauer, Sebastian, Liedl, Rudolf, and Sauter, Martin, 2003, Modeling of karst aquifer genesis: Influence of exchange flow: Water Resources Research, v. 39, no. 10, 1,285 p.

Bear, Jacob, 1979, Hydraulics of groundwater: New York, McGraw-Hill, 567 p.

Birk, Steffen, 2002, Characterization of karst systems by simulating aquifer genesis and spring responses: Model development and application to gypsum karst: Vol. 60 of Tübinger Geowissenschaftliche Arbeiten: Tübingen, Germany, Reihe C. Institut und Museum für Geologie und Paläontologie der Universität Tübingen. Online at *http://w210.ub.uni-tuebingen.de/ dbt/volltexte/2002/558/*

Bouwer, Herman, 1978, Groundwater hydrology: New York, McGraw-Hill, 480 p.

Chin, D.A., 2000, Water-resources engineering: Upper Saddle River, N.J., Prentice Hall, 750 p.

Clemens, Torsten, 1998, Simulation der Entwicklung von Karstaquiferen: Ph.D. dissertation submitted to Eberhard-Karls Universitat Tübingen, Germany.

Clemens, Torsten, Hückinghaus, Dirk, Sauter, Martin, Liedl, Rudolf, and Teutsch, Georg, 1996, A combined continuum and discrete network reactive transport model for the simulation of karst development, *in* Calibration and reliability in groundwater modelling—Proceedings of the ModelCARE 96 Conference, Golden, Colo., September 1996, International Association of Hydrological Sciences Publication 237, p. 309–318.

Colebrook, C.F., and White, C.M., 1937, The reduction of carrying capacity of pipes with age: London, Journal Institute of Civil Engineers, v. 7, p. 99.

Craft, B.C., Holden, W.R., and Graves, E.D., Jr., 1962, Well design: Drilling and production: Englewood Cliffs, N.J., Prentice-Hall, 571 p.

Cunningham, K.J., Wacker, M.A., Robinson, E., Dixon, J.F., and Wingard, G.L., 2006, A cyclostratigraphic and borehole geophysical approach to development of the three-dimensional conceptual hydrogeologic model of the karstic Biscayne aquifer, southeastern Florida: U.S. Geological Survey Scientific Investigations Report 2005–5235, 69 p.

Daily, J.W., and Harleman, D.R.F., 1966, Fluid dynamics: Reading, Mass., Addison Wesley, 454 p.

Darcy, H., 1856, Les Fontances publiques de la ville de Dijon: Paris, Victor Dalmont.

Davis, H., 1996, Hydrogeologic investigation and simulation of ground-water flow in the Upper Floridan aquifer of north-central Florida and delineation of contributing areas for selected City of Tallahassee, Florida, water-supply wells: U.S. Geological Survey Water-Resources Investigations Report 95–4296, 55 p.

Doherty, J., 2002, PEST—Model independent parameter estimation, version 6. Queensland, Australia: Watermark Numerical Computing.

Dreybrodt, W., 1988, Processes in karst systems—Physics, chemistry, and geology: Berlin, Springer-Verlag, 288 p.

Ewen, J., Parkin, G., and O'Connell, P.E., 2000, SHETRAN: A coupled surface/subsurface modeling system for 3D water flow and sediment and solute transport in river basins: American Society of Civil Engineering, Journal of Hydrologic Engineering, v. 5, no. 3, p. 50–258.

Fenske, J.P., Leake, S.A., and Prudic, D.E., 1996, Documentation of a computer program (RES1) to simulate leakage from reservoirs using the modular finite-difference ground-water flow model (MODFLOW): U.S. Geological Survey Open-File Report 96–364, 51 p.

Ford, D.C., 2003, Perspectives in karst hydrogeology and cavern genesis: Speleogenesis and evolution of karst aquifers: The Virtual Scientific Journal, v. 1, 12 p. Online at *http://www.speleogenesis.info*

Ford, D.C., and Williams, P.W., 1989, Karst geomorphology and hydrology: London, Unwin Hyman.

Freeze, R.A., and Cherry, J.A., 1979, Groundwater: Englewood Cliffs, N.J., Prentice-Hall, 604 p.

French, R.H., 1985, Open-channel hydraulics: New York, McGraw Hill, 739 p.

Halford, K.J., 2000, Simulation and interpretation of borehole flowmeter results under laminar and turbulent flow conditions: Seventh International Symposium on Logging for Minerals and Geotechnical Applications, Golden, Colo., October 24–26, 2000, p. 157–168.

Halford, K.J., and Hanson, R.T., 2002, User guide for the drawdown-limited, multi-node well (MNW) package for the U.S. Geological Survey's modular three-dimensional finite-difference ground-water flow model, versions MODFLOW-96 and MODFLOW-2000: U.S. Geological Survey Open-File Report 02–293, 33 p.

Harbaugh, A.W., 2005, MODFLOW-2005, the U.S. Geological Survey modular ground-water model—The ground-water flow process: U.S. Geological Survey Techniques and Methods 6-A16, variously paginated.

Harbaugh, A.W., Banta, E.R., Hill, M.C., and McDonald, M.G., 2000, MODFLOW-2000, The U.S. Geological Survey modular ground-water model—User's guide to modularization concepts and the ground-water flow process: U.S. Geological Survey Open-File Report 00–92, 121 p.

Harbaugh, A.W., and McDonald, M.G., 1996, User's documentation for MODFLOW-96, an update to the U.S. Geological Survey modular finite-difference ground-water flow model: U.S. Geological Survey Open-File Report 96–485, 56 p.

Henderson, F.M., 1966, Open channel flow: New York, Macmillan Series in Civil Engineering, 522 p.

Hill, M.C., Banta, E.R., Harbaugh, A.W., and Anderman, E.R., 2000, MODFLOW-2000, the U.S. Geological Survey modular ground-water model—User guide to the Observation, Sensitivity, and Parameter-Estimation Processes and three post-processing programs: U.S. Geological Survey Open-File Report 00–184, 209 p.

Horlacher, H.B., and Lüdecke, H.J., 1992, Strömungsberechnung für Rohrsysteme [Current calculation for tube systems]: Ehningen, Germany, Springer-Verlag.

Hsieh, P.A., and Freckleton, J.R., 1993, Documentation of a computer program to simulate horizontal flow barriers using the U.S. Geological Survey's modular three-dimensional finite-difference ground-water flow model: U.S. Geological Survey Open-File Report 92–477, 32 p.

Hückinghaus, Dirk, 1998, Simulation der Aquifergenese und des Wärmetransports in Karstaquiferen: Vol. 42 of Tübinger Geowissenschaftliche Arbeiten: Tübingen, Germany, Reihe C. Institut und Museum für Geologie und Paläontologie der Universität Tübingen. Online at *http://w210.ub.uni-tuebingen.de/dbt/volltexte/2005/2030/*

Katz, B.G., 2004, Sources of nitrate contamination and age of water in large karstic springs of Florida, Environmental Geology, v. 46, p. 689–706, DOI 10.1007/s00254-004-1061-9.

Kiraly, L., 1998, Modelling karst aquifers by the combined discrete channel and continuum approach: Bulletin d'Hydrogeologie, v. 16, p. 77–98.

Klimchouk, A.B., Ford, D.C., Palmer, A.N., Dreybrodt, Wolfgang, eds., 2000, Speleogenesis: Evolution of karst aquifers: Huntsville, Ala., National Speleogical Society, 527 p.

Knochenmus, L.A., and Robinson, J.L., 1996, Descriptions of anisotropy and heterogeneity and their effect on groundwater flow and areas of contribution to public supply wells in a karst carbonate aquifer system: U.S. Geological Survey Water-Supply Paper 2475, 47 p.

Kuniansky, E.L., and Danskin, W.R., 2003, Models gone bad—Common modeling problems and how to solve them, *in* Poeter, Zheng, Hill, and Doherty, eds., MODFLOW and More 2003: Understanding through Modeling–Conference Proceedings: Golden, Colo., Colorado School of Mines, p. 356–360.

Kuniansky, E.L., Fahlquist, Lynne, and Ardis, A.F., 2001, Travel times along selected flow paths of the Edwards aquifer, central Texas, *in* Kuniansky, E.L., ed., U.S. Geological Survey Karst Interest Group Proceedings, St. Petersburg, Florida, February 13–16, 2001: U.S. Geological Survey Water-Resources Investigations Report 01–4011, p. 69–77.

Kuniansky, E.L., and Holligan, K.Q., 1994, Simulations of flow in the Edwards-Trinity aquifer system and contiguous hydraulically units, west-central Texas: U.S. Geological Survey Water-Resources Investigations Report 93–4093, 40 p.

Lang, U., 1995, Simulation regionaler Strömungs—Und Transportvorgänge in Karstaquiferen mit Hilfe des Doppelkontinuumansatzes: Stuttgart, Germany, Methodenentwicklung und Parameterstudie. Ph.D. dissertation, Institut für Wasserbau, Universität Stuttgart.

McDonald, M.G., and Harbaugh, A.W., 1988, A modular three-dimensional finite-difference ground-water flow model: U.S. Geological Survey Techniques of Water-Resources Investigations, book 6, chap. A1, 548 p.

Means, G.H., and Scott, T.M., 2005, Swallets in Florida: Contaminate pathways: Geological Society of America, Abstracts with Programs, v. 37, no. 7, p. 435.

Mehl, Steffen, 2006, Use of picard and newton iteration for solving nonlinear groundwater flow equations: Ground Water, v. 44, no. 4, p. 583–594.

Merritt, M.L., and Konikow, L.F., 2000, Documentation of a computer program to simulate lake-aquifer interaction using the MODFLOW ground-water flow model and the MOC3D solute-transport model: U.S. Geological Survey Water-Resources Investigations Report 00–4167, 146 p.

Mohrlok, U., 1996, Parameter-identifikation in doppel-kontinuum-modellen am beispiel von karst-aquiferen: Tübingen, Germany, Reihe C. Institut und Museum für Geologie und Paläontologie der Universität Tübingen, Tübinger Geowissenschaftliche Arbeiten, v. 31.

Morris, H.M., 1963, Applied hydraulics in engineering: New York, Ronald Press, 455 p.

Munson, B.R., Young, D.F., and Okiishi, T.H.,1994, Fundamentals of fluid mechanics (2d ed.): New York, John Wiley, 14 p.

Paillet, F.L., 1998, Flow modeling and permeability estimation using borehole flow logs in heterogeneous fractured formations: Water Resources Research, v. 34, no. 5, p. 997–1,010.

Poeter, E.P., and Hill, M.C., 1998, Documentation of UCODE, a computer code for universal inverse modeling: U.S. Geological Survey Water-Resources Investigations Report 98–4080.

Prudic, D.E., Konikow, L.F., and Banta, E.R., 2004, A new Streamflow-Routing (SFR1) Package to simulate stream-aquifer interaction with MODFLOW-2000: U.S. Geological Survey Open-File Report 2004–1042, 95 p.

Raudkivi, A.J., and Callander, R.A., 1976, Analysis of groundwater flow: London, Edward Arnold, 214 p.

Renken, R.A., Cunningham, K.J., Zygnerski, M.R., Wacker, M.A., Shapiro, A.M., Harvey, R.W., Metge, D.W., Osborn, C.L., and Ryan, J.N., 2005, Assessing the vulnerability of a municipal wellfield to contamination in a karst aquifer: Environmental and Engineering Geoscience, v. XI, no. 4, p. 319–331.

Rouse, Hunter, ed., 1950, Engineering hydraulics: Proceedings of the Fourth Hydraulics Conference, Iowa Institute of Hydraulic Research, June 12–15, 1949: New York, John Wiley, 1,039 p.

Sauter, Martin, 1993, Double porosity models in karstified limestone aquifers: Field validation and data provision, *in* Hydrogeological Process in Karst Terranes, International Association of Hydrological Sciences, v. 207, p. 261–279.

Scanlon, B.R., Mace, R.E., Barrett, M.E., Smith, Brian, 2003, Can we simulate regional groundwater flow in a karst system using equivalent porous media models? Case study, Barton Springs Edwards aquifer, United States of America: Journal of Hydrology, v. 276, p. 137–158.

Schneebeli, G., 1955, Experiences sur la limite de validite de la loi de Darcy et l'apparition de la turbulence dans un ecoulement de filtration: La Houille Blanche, v. 10, 141–149 p.

Sepulveda, Nicasio, 2002, Simulation of ground-water flow in the intermediate and Floridan aquifer systems in peninsular Florida: U.S. Geological Survey Water-Resources Investigations Report 02–4009, 130 p.

Shoemaker, W.B., O'Reilly, A.M., Sepulveda, N., Williams, S.A., Motz, L.H., and Suiv, Q., 2003, Comparison of estimated areas contributing recharge to selected springs in north-central Florida using multiple ground-water flow models: U.S. Geological Survey Open-File Report 03–448, 31 p.

Svensson, U., 2001, A continuum representation of fracture networks, Part I: Method and basic test cases: Journal of Hydrology, v. 250, p. 170–186.

Teutsch, Georg, 1993, An extended double-porosity concept as a practical modeling approach for a karstified terraine, *in* Gultekin, Johnson, and Back, eds., Hydrogeological processes in karst terraines: Proceedings of the Antalya Symposium and Field Seminar, October 1990: International Association of Hydrological Sciences, v. 207, p. 281–292.

Teutsch, G., and Sauter, Martin, 1998, Distributed parameter modelling approaches in karst hydrological investigations: Bulletin d'Hydrogeologie, v. 16, p. 99–110.

Thrailkill, J., 1986, Models and methods for shallow conduit-flow aquifers, *in* Proceedings of the Environmental Problems in Karst Terraines and their Solutions Conference: Bowling Green, Ky., National Water Well Association, p. 17–31.

Vennard, J.K., and Street, R.L., 1975, Elementary fluid mechanics (5th ed.): New York, John Wiley, 740 p.

Weast, R.C., 1979, CRC Handbook of chemistry and physics: A ready-reference book of chemical and physical data (60th ed.): Boca Raton, Fla., Chemical Rubber Press.

White, F.M., 1994, Fluid mechanics (3d ed.): New York, McGraw-Hill, 736 p.

White, W.B., 1969, Conceptual models for carbonate aquifers: Ground Water, v. 7, no. 3, p. 15–21.

White, W.B., 1993, Analysis of karst aquifers, *in* Alley, W.M., ed., Regional ground-water quality: New York, Van Nostrand Reinhold, p. 471–489.

White, W.B., 2003, Conceptual models of karstic aquifers: Speleogenesis and evolution of karst aquifers: The Virtual Scientific Journal, v. 1, 6 p. Online at *http://www.speleogenisis.info*

Wolfe, W.J., Haugh, C.J., Webbers, Ank, and Diehl, T.H., 1997, Preliminary conceptual models of the occurrence, fate, and transport of chlorinated solvents in karst regions of Tennessee: U.S. Geological Survey Water-Resources Investigations Report 97–4097, 80 p.