# MODFLOW6的输入

Li

# 引言

MODFLOW6没有GUI界面，命令行输入，读取ASCII文本文件或二进制文件。
计算结果保存为ASCII文本文件或二进制文件。

MODFLOW6的数值方法和原理见单独的报告文件(Hughes and others, 2017; Langevin and others, 2017; Provost and others, 2017; Langevin and others, 2020; Hughes and others, 2022a; Langevin and others, 2022; Hughes and others, 2022b)。

# 运行模拟

需要mfsim.nam文件，否则终止运行。

可以查询运行参数：
mf6.exe –h

运行过程创建输出文件mfsim.lst

# 输入指令的形式

为了增加输入文件的可读性，使用blocks and keywords

BEGIN

END

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [PRINT_INPUT]
  [MAXIMUM_ITERATION <maxsfrit>]
END OPTIONS
```

本报告的举例样式：

一个有效输入格式：

```
#This is my options block
BEGIN OPTIONS
  AUXILIARY temperature salinity
  MAXIMUM_ITERATION 10
END OPTIONS
```

另一个有效输入格式：

```
#This is an alternative options block
BEGIN OPTIONS
  # Assign two auxiliary variables
  AUXILIARY temperature salinity
  # Specify the maximum iteration
  MAXIMUM_ITERATION 10
  #specify the print input option
  PRINT_INPUT
END OPTIONS
#done with the options block
```

# 在OPEN/CLOSE文件中定义块信息

大部分的块信息可以从单独的文本文件读取，但有的块不支持OPEN/CLOSE功能（见附录A）。

```
#This is an alternative options block
BEGIN OPTIONS
  OPEN/CLOSE myoptblock.txt
END OPTIONS
```

# 附录A Blocks列表（OPEN/CLOSE表示块信息是否能在单独的文件定义）

| Component | FTYPE | Blockname | OPEN/CLOSE |
|---|---|---|---|
| SIM | NAM | OPTIONS | yes |
| SIM | NAM | TIMING | yes |
| SIM | NAM | MODELS | yes |
| SIM | NAM | EXCHANGES | yes |
| SIM | NAM | SOLUTIONGROUP | yes |
| SIM | TDIS | OPTIONS | yes |
| SIM | TDIS | DIMENSIONS | yes |
| SIM | TDIS | PERIODDATA | yes |
| EXG | GWFGWF | OPTIONS | yes |
| EXG | GWFGWF | DIMENSIONS | yes |
| EXG | GWFGWF | EXCHANGEDATA | yes |
| EXG | GWTGWT | OPTIONS | yes |
| EXG | GWTGWT | DIMENSIONS | yes |
| EXG | GWTGWT | EXCHANGEDATA | yes |
| SLN | IMS | OPTIONS | yes |
| SLN | IMS | NONLINEAR | yes |
| SLN | IMS | LINEAR | yes |
| GWF | NAM | OPTIONS | yes |
| GWF | NAM | PACKAGES | yes |
| GWF | DIS | OPTIONS | yes |
| GWF | DIS | DIMENSIONS | yes |
| GWF | DIS | GRIDDATA | no |
| GWF | DISV | OPTIONS | yes |
| GWF | DISV | DIMENSIONS | yes |
| GWF | DISV | GRIDDATA | no |
| GWF | DISV | VERTICES | yes |
| GWF | DISV | CELL2D | yes |
| GWF | DISU | OPTIONS | yes |
| GWF | DISU | DIMENSIONS | yes |
| GWF | DISU | GRIDDATA | no |
| GWF | DISU | CONNECTIONDATA | yes |
| GWF | DISU | VERTICES | yes |
| GWF | DISU | CELL2D | yes |
| GWF | IC | GRIDDATA | no |
| GWF | NPF | OPTIONS | yes |
| GWF | NPF | GRIDDATA | no |
| GWF | BUY | OPTIONS | yes |
| GWF | BUY | DIMENSIONS | yes |
| GWF | BUY | PACKAGEDATA | yes |

| Component | FTYPE | Blockname | OPEN/CLOSE |
|---|---|---|---|
| GWF | STO | OPTIONS | yes |
| GWF | STO | GRIDDATA | no |
| GWF | STO | PERIOD | yes |
| GWF | CSUB | OPTIONS | yes |
| GWF | CSUB | DIMENSIONS | yes |
| GWF | CSUB | GRIDDATA | no |
| GWF | CSUB | PACKAGEDATA | yes |
| GWF | CSUB | PERIOD | yes |
| GWF | HFB | OPTIONS | yes |
| GWF | HFB | DIMENSIONS | yes |
| GWF | HFB | PERIOD | yes |
| GWF | CHD | OPTIONS | yes |
| GWF | CHD | DIMENSIONS | yes |
| GWF | CHD | PERIOD | yes |
| GWF | WEL | OPTIONS | yes |
| GWF | WEL | DIMENSIONS | yes |
| GWF | WEL | PERIOD | yes |
| GWF | DRN | OPTIONS | yes |
| GWF | DRN | DIMENSIONS | yes |
| GWF | DRN | PERIOD | yes |
| GWF | RIV | OPTIONS | yes |
| GWF | RIV | DIMENSIONS | yes |
| GWF | RIV | PERIOD | yes |
| GWF | GHB | OPTIONS | yes |
| GWF | GHB | DIMENSIONS | yes |
| GWF | GHB | PERIOD | yes |
| GWF | RCH | OPTIONS | yes |
| GWF | RCH | DIMENSIONS | yes |
| GWF | RCH | PERIOD | yes |
| GWF | RCHA | OPTIONS | yes |
| GWF | RCHA | PERIOD | yes |
| GWF | EVT | OPTIONS | yes |
| GWF | EVT | DIMENSIONS | yes |
| GWF | EVT | PERIOD | yes |
| GWF | EVTA | OPTIONS | yes |
| GWF | EVTA | PERIOD | yes |
| GWF | MAW | OPTIONS | yes |
| GWF | MAW | DIMENSIONS | yes |
| GWF | MAW | PACKAGEDATA | yes |
| GWF | MAW | CONNECTIONDATA | yes |
| GWF | MAW | PERIOD | yes |
| GWF | SFR | OPTIONS | yes |

| Component | FTYPE | Blockname | OPEN/CLOSE |
|---|---|---|---|
| GWF | SFR | DIMENSIONS | yes |
| GWF | SFR | PACKAGEDATA | yes |
| GWF | SFR | CROSSSECTIONS | yes |
| GWF | SFR | CONNECTIONDATA | yes |
| GWF | SFR | DIVERSIONS | yes |
| GWF | SFR | PERIOD | yes |
| GWF | LAK | OPTIONS | yes |
| GWF | LAK | DIMENSIONS | yes |
| GWF | LAK | PACKAGEDATA | yes |
| GWF | LAK | CONNECTIONDATA | yes |
| GWF | LAK | TABLES | yes |
| GWF | LAK | OUTLETS | yes |
| GWF | LAK | PERIOD | yes |
| GWF | UZF | OPTIONS | yes |
| GWF | UZF | DIMENSIONS | yes |
| GWF | UZF | PACKAGEDATA | yes |
| GWF | UZF | PERIOD | yes |
| GWF | MVR | OPTIONS | yes |
| GWF | MVR | DIMENSIONS | yes |
| GWF | MVR | PACKAGES | yes |
| GWF | MVR | PERIOD | yes |
| GWF | GNC | OPTIONS | yes |
| GWF | GNC | DIMENSIONS | yes |
| GWF | GNC | GNCDATA | yes |
| GWF | OC | OPTIONS | yes |
| GWF | OC | PERIOD | yes |
| GWF | API | OPTIONS | yes |
| GWF | API | DIMENSIONS | yes |
| GWT | ADV | OPTIONS | yes |
| GWT | DSP | OPTIONS | yes |
| GWT | DSP | GRIDDATA | no |
| GWT | CNC | OPTIONS | yes |
| GWT | CNC | DIMENSIONS | yes |
| GWT | CNC | PERIOD | yes |
| GWT | DIS | OPTIONS | yes |
| GWT | DIS | DIMENSIONS | yes |
| GWT | DIS | GRIDDATA | no |
| GWT | DISV | OPTIONS | yes |
| GWT | DISV | DIMENSIONS | yes |
| GWT | DISV | GRIDDATA | no |
| GWT | DISV | VERTICES | yes |
| GWT | DISV | CELL2D | yes |

# 附录A Blocks列表（OPEN/CLOSE表示块信息是否能在单独的文件定义）

| Component | FTYPE | Blockname | OPEN/CLOSE |
|---|---|---|---|
| GWT | DISU | OPTIONS | yes |
| GWT | DISU | DIMENSIONS | yes |
| GWT | DISU | GRIDDATA | no |
| GWT | DISU | CONNECTIONDATA | yes |
| GWT | DISU | VERTICES | yes |
| GWT | DISU | CELL2D | yes |
| GWT | IC | GRIDDATA | no |
| GWT | NAM | OPTIONS | yes |
| GWT | NAM | PACKAGES | yes |
| GWT | OC | OPTIONS | yes |
| GWT | OC | PERIOD | yes |
| GWT | SSM | OPTIONS | yes |
| GWT | SSM | SOURCES | yes |
| GWT | SSM | FILEINPUT | yes |
| GWT | SRC | OPTIONS | yes |
| GWT | SRC | DIMENSIONS | yes |
| GWT | SRC | PERIOD | yes |
| GWT | MST | OPTIONS | yes |
| GWT | MST | GRIDDATA | no |
| GWT | IST | OPTIONS | yes |
| GWT | IST | GRIDDATA | no |
| GWT | SFT | OPTIONS | yes |
| GWT | SFT | PACKAGEDATA | yes |
| GWT | SFT | PERIOD | yes |
| GWT | LKT | OPTIONS | yes |
| GWT | LKT | PACKAGEDATA | yes |
| GWT | LKT | PERIOD | yes |
| GWT | MWT | OPTIONS | yes |
| GWT | MWT | PACKAGEDATA | yes |
| GWT | MWT | PERIOD | yes |
| GWT | UZT | OPTIONS | yes |
| GWT | UZT | PACKAGEDATA | yes |
| GWT | UZT | PERIOD | yes |
| GWT | FMI | OPTIONS | yes |
| GWT | FMI | PACKAGEDATA | yes |
| GWT | MVT | OPTIONS | yes |
| GWT | API | OPTIONS | yes |
| GWT | API | DIMENSIONS | yes |
| UTL | SPC | OPTIONS | yes |
| UTL | SPC | DIMENSIONS | yes |
| UTL | SPC | PERIOD | yes |
| UTL | SPCA | OPTIONS | yes |
| UTL | SPCA | PERIOD | yes |
| UTL | OBS | OPTIONS | yes |
| UTL | OBS | CONTINUOUS | yes |
| UTL | LAKTAB | DIMENSIONS | yes |
| UTL | LAKTAB | TABLE | yes |
| UTL | SFRTAB | DIMENSIONS | yes |
| UTL | SFRTAB | TABLE | yes |
| UTL | TS | ATTRIBUTES | yes |
| UTL | TS | TIMESERIES | yes |
| UTL | TAS | ATTRIBUTES | yes |
| UTL | TAS | TIME | no |
| UTL | ATS | DIMENSIONS | yes |
| UTL | ATS | PERIODDATA | yes |
| UTL | TVK | OPTIONS | yes |
| UTL | TVK | PERIOD | yes |
| UTL | TVS | OPTIONS | yes |
| UTL | TVS | PERIOD | yes |

## 文件名输入

Windows不区分文件名大小写，但Linux系统区分。

如MODEL.DIS

## 模拟名称文件

mfsim.nam

文件包含如下的输入块，必须以一定顺序列出。

options块是可选的，还需要其他的一些块。

# 模拟名称文件(Simulation Name File)

## 块的结构

```
BEGIN OPTIONS
  [CONTINUE]
  [NOCHECK]
  [MEMORY_PRINT_OPTION <memory_print_option>]
  [MAXERRORS <maxerrors>]
END OPTIONS

BEGIN TIMING
  TDIS6 <tdis6>
END TIMING

BEGIN MODELS
  <mtype> <mfname> <mname>
  <mtype> <mfname> <mname>
  ...
END MODELS

BEGIN EXCHANGES
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  ...
END EXCHANGES

BEGIN SOLUTIONGROUP <group_num>
  [MXITER <mxiter>]
  <slntype> <slnfname> <slnmnames(:)>
  <slntype> <slnfname> <slnmnames(:)>
  ...
END SOLUTIONGROUP
```

多个GWF模型之间的交换（拓扑关系文件）

# 变量解释

```
BEGIN OPTIONS
  [CONTINUE]
  [NOCHECK]
  [MEMORY_PRINT_OPTION <memory_print_option>]
  [MAXERRORS <maxerrors>]
END OPTIONS
```

```
BEGIN TIMING
  TDIS6 <tdis6>
END TIMING
```

```
BEGIN MODELS
  <mtype> <mfname> <mname>
  <mtype> <mfname> <mname>
  ...
END MODELS
```

```
BEGIN EXCHANGES
  <exgtype> <exgfile> <exgmnam
  <exgtype> <exgfile> <exgmnam
  ...
END EXCHANGES
```

```
BEGIN SOLUTIONGROUP <group_num>
  [MXITER <mxiter>]
  <slntype> <slnfname> <slnmnames(:)>
  <slntype> <slnfname> <slnmnames(:)>
  ...
END SOLUTIONGROUP
```

**Block: OPTIONS**

CONTINUE—keyword flag to indicate that the simulation should continue even if one or more solutions do not converge.

NOCHECK—keyword flag to indicate that the model input check routines should not be called prior to each time step. Checks are performed by default.

memory_print_option—is a flag that controls printing of detailed memory manager usage to the end of the simulation list file. NONE means do not print detailed information. SUMMARY means print only the total memory for each simulation component. ALL means print information for each variable stored in the memory manager. NONE is default if MEMORY_PRINT_OPTION is not specified.

maxerrors—maximum number of errors that will be stored and printed.

# 变量解释

```
BEGIN OPTIONS
  [CONTINUE]
  [NOCHECK]
  [MEMORY_PRINT_OPTION <memory_print_option>]
  [MAXERRORS <maxerrors>]
END OPTIONS
```

```
BEGIN TIMING
  TDIS6 <tdis6>
END TIMING
```

**Block: TIMING**

**tdis6**—is the name of the Temporal Discretization (TDIS) Input File.

```
BEGIN MODELS
  <mtype> <mfname> <mname>
  <mtype> <mfname> <mname>
  ...
END MODELS
```

**Block: MODELS**

**mtype**—is the type of model to add to simulation.

**mfname**—is the file name of the model name file.

**mname**—is the user-assigned name of the model. The model name cannot exceed 16 characters and mu
not have blanks within the name. The model name is case insensitive; any lowercase letters are co
verted and stored as upper case letters.

```
BEGIN EXCHANGES
  <exgtype> <exgfile> <exgmna
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  ...
END EXCHANGES
```

```
BEGIN SOLUTIONGROUP <group_num>
  [MXITER <mxiter>]
  <slntype> <slnfname> <slnmnames(:)>
  <slntype> <slnfname> <slnmnames(:)>
  ...
END SOLUTIONGROUP
```

**Table 2.** Model types available in Version mf6.3.0.

| Mtype | Type of Model |
|---|---|
| GWF6 | Groundwater Flow Model |
| GWT6 | Groundwater Transport Model |

# 变量解释

## Block: EXCHANGES

```
BEGIN OPTIONS
  [CONTINUE]
  [NOCHECK]
  [MEMORY_PRINT_OPTION <m
  [MAXERRORS <maxerrors>]
END OPTIONS

BEGIN TIMING
  TDIS6 <tdis6>
END TIMING

BEGIN MODELS
  <mtype> <mfname> <mname>
  <mtype> <mfname> <mname>
  ...
END MODELS

BEGIN EXCHANGES
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  ...
END EXCHANGES

BEGIN SOLUTIONGROUP <group_num>
  [MXITER <mxiter>]
  <slntype> <slnfname> <slnmnames(:)>
  <slntype> <slnfname> <slnmnames(:)>
  ...
END SOLUTIONGROUP
```

**exgtype**—is the exchange type.

**exgfile**—is the input file for the exchange.

**exgmnamea**—is the name of the first model that is part of this exchange.

**exgmnameb**—is the name of the second model that is part of this exchange.

**Table 3.** Exchange types available in Version mf6.3.0.

| Exgtype | Type of Exchange |
|---|---|
| GWF6-GWF6 | Exchange between two Groundwater Flow Models. Input for this file is described in a dedicated section in this guide. |
| GWF6-GWT6 | Exchange between a Groundwater Flow Model and a Groundwater Transport Model. In the present version, a filename is required for this exchange and the file must exist, however, nothing is read from this file. |
| GWT6-GWT6 | Exchange between two Groundwater Transport Models. Input for this file is described in a dedicated section in this guide. |

# 变量解释

```
BEGIN OPTIONS
  [CONTINUE]
  [NOCHECK]
  [MEMORY_PRINT_OPTION <mem
  [MAXERRORS <maxerrors>]
END OPTIONS

BEGIN TIMING
  TDIS6 <tdis6>
END TIMING

BEGIN MODELS
  <mtype> <mfname> <mname>
  <mtype> <mfname> <mname>
  ...
END MODELS

BEGIN EXCHANGES
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  <exgtype> <exgfile> <exgmnamea> <exgmnameb>
  ...
END EXCHANGES

BEGIN SOLUTIONGROUP <group_num>
  [MXITER <mxiter>]
  <slntype> <slnfname> <slnmnames(:)>
  <slntype> <slnfname> <slnmnames(:)>
  ...
END SOLUTIONGROUP
```

**Block: SOLUTIONGROUP**

**group_num**—is the group number of the solution group. Solution groups must be numbered sequentially, starting with group number one.

**mxiter**—is the maximum number of outer iterations for this solution group. The default value is 1. If there is only one solution in the solution group, then MXITER must be 1.

**slntype**—is the type of solution. The Integrated Model Solution (IMS6) is the only supported option in this version.

**slnfname**—name of file containing solution input.

**slnmnames**—is the array of model names to add to this solution. The number of model names is determined by the number of model names the user provides on this line.

# 输入文件举例

```
# This block is optional
BEGIN OPTIONS
END OPTIONS

# Simulation timing information
BEGIN TIMING
  TDIS6 simulation.tdis
END TIMING

# List of models in the simulation
BEGIN MODELS
  #modeltype     namefile     modelname
        GWF6    model1.nam   GWF_Model_1
        GWF6    model2.nam   GWF_Model_2
END MODELS

# List of exchanges in the simulation
BEGIN EXCHANGES
  GWF6-GWF6 simulation.exg GWF_Model_1 GWF_Model_2
END EXCHANGES

# Models are part of the same numerical solution
BEGIN SOLUTIONGROUP 1
  IMS6 simulation.ims GWF_Model_1 GWF_Model_2
END SOLUTIONGROUP
```

**GWF模型之间交换的文件(.exg)**

# 时间离散（TDIS）软件包

TDIS软件的输入都是从模拟名称文件中的TIMING输入块的TDIS文件读取。

```
BEGIN OPTIONS
  [TIME_UNITS <time_units>]
  [START_DATE_TIME <start_date_time>]
  [ATS6 FILEIN <ats6_filename>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NPER <nper>
END DIMENSIONS
```

```
BEGIN PERIODDATA
  <perlen> <nstp
  <perlen> <nstp
  ...
END PERIODDATA
```

**Block: OPTIONS**

`time_units`—is the time units of the simulation. This is a text string that is used as a label within model output files. Values for time_units may be “unknown”, “seconds”, “minutes”, “hours”, “days”, or “years”. The default time unit is “unknown”.

`start_date_time`—is the starting date and time of the simulation. This is a text string that is used as a label within the simulation list file. The value has no effect on the simulation. The recommended format for the starting date and time is described at https://www.w3.org/TR/NOTE-datetime.

`ATS6`—keyword to specify that record corresponds to an adaptive time step (ATS) input file. The behavior of ATS and a description of the input file is provided separately.

`FILEIN`—keyword to specify that an input filename is expected next.

`ats6_filename`—defines an adaptive time step (ATS) input file defining ATS controls. Records in the ATS file can be used to override the time step behavior for selected stress periods.

# 时间离散（TDIS）软件包

TDIS软件的输入都是从模拟名称文件中的TIMING输入块的TDIS文件读取。

```
BEGIN OPTIONS
  [TIME_UNITS <time_units>]
  [START_DATE_TIME <start_date_time>]
  [ATS6 FILEIN <ats6_filename>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NPER <nper>
END DIMENSIONS
```

```
BEGIN PERIODDATA
  <perlen> <nstp> <tsmult>
  <perlen> <nstp> <tsmult>
  ...
END PERIODDATA
```

**Block: DIMENSIONS**

nper—is the number of stress periods for the simulation.

**Block: PERIODDATA**

perlen—is the length of a stress period.

nstp—is the number of time steps in a stress period.

tsmult—is the multiplier for the length of successive time steps. The length of a time step is calculated by multiplying the length of the previous time step by TSMULT. The length of the first time step, $\Delta t_1$, is related to PERLEN, NSTP, and TSMULT by the relation $\Delta t_1 = perlen\frac{tsmult-1}{tsmult^{nstp}-1}$.

# 时间离散（TDIS）软件包

输入文件举例：

```
# Comment for this TDIS input file

BEGIN OPTIONS
  TIME_UNITS DAYS
END OPTIONS

BEGIN DIMENSIONS
  NPER 2
END DIMENSIONS

BEGIN PERIODDATA
   365.00 1 1.0  Items: PERLEN NSTP TSMULT
   365.00 10 1.2 Items: PERLEN NSTP TSMULT
END PERIODDATA
```

# 自适应时间步(ATS)功能

在TDIS输入文件中激活定义ATS6选项，在TDIS软件包中使用Adaptive Time Step (ATS)工具。

在PERIODDATA块中激活对任意stress periods的自适应时间步长功能。如果是自适应的，则nstp和tsmult参数对时间步长推进没有影响。否则，使用定义的ATS设置控制时间推进。

目前实施ATS的限制是：无法在驱动周期内显式定义保存输出的次数。在周期结束时获得输出，在周期内根据Output Control时间步设置。Output Control设置保存结果是基于FIRST, LAST, FREQUENCY和STEPS选项。

# 自适应时间步(ATS)功能

## 块结构

```
BEGIN DIMENSIONS
  MAXATS <maxats>
END DIMENSIONS

BEGIN PERIODDATA
  <iperats> <dt0> <dtmin> <dtmax
  <iperats> <dt0> <dtmin> <dtmax
  ...
END PERIODDATA
```

**Block: DIMENSIONS**

`maxats`—is the number of records in the subsequent perioddata block that will be used for adaptive time stepping.

**Block: PERIODDATA**

`iperats`—is the period number to designate for adaptive time stepping. The remaining ATS values on this line will apply to period iperats. iperats must be greater than zero. A warning is printed if iperats is greater than nper.

`dt0`—is the initial time step length for period iperats. If dt0 is zero, then the final step from the previous stress period will be used as the initial time step. The program will terminate with an error message if dt0 is negative.

`dtmin`—is the minimum time step length for this period. This value must be greater than zero and less than dtmax. dtmin must be a small value in order to ensure that simulation times end at the end of stress periods and the end of the simulation. A small value, such as 1.e-5, is recommended.

`dtmax`—is the maximum time step length for this period. This value must be greater than dtmin.

`dtadj`—is the time step multiplier factor for this period. If the number of outer solver iterations are less than the product of the maximum number of outer iterations (OUTER_MAXIMUM) and ATS_OUTER_MAXIMUM_FRACTION (an optional variable in the IMS input file with a default value of 1/3), then the time step length is multipled by dtadj. If the number of outer solver iterations are greater than the product of the maximum number of outer iterations and ATS_OUTER_MAXIMUM_FRACTION, then the time step length is divided by dtadj. dtadj must be zero, one, or greater than one. If dtadj is zero or one, then it has no effect on the simulation. A value between 2.0 and 5.0 can be used as an initial estimate.

`dtfailadj`—is the divisor of the time step length when a time step fails to converge. If there is solver failure, then the time step will be tried again with a shorter time step length calculated as the previous time step length divided by dtfailadj. dtfailadj must be zero, one, or greater than one. If dtfailadj is zero or one, then time steps will not be retried with shorter lengths. In this case, the program will terminate with an error, or it will continue of the CONTINUE option is set in the simulation name file. Initial tests with this variable should be set to 5.0 or larger to determine if convergence can be achieved.

# 自适应时间步(ATS)功能

输入文件举例：

```
# ATS input file

BEGIN dimensions
  MAXATS 2
END dimensions
```

```
BEGIN perioddata
# per    dt0   dtmin   dtmax  dtadj  dtfailadj
    2  100.0 1.0E-5 1000.0   2.0        5.0
    7   10.0 1.0E-5  100.0   1.7        2.0
END perioddata
```

# GWF模型输入

# GWF模型输入

GWF模型在模拟名称文件的MODELS块的GWF入口加入模拟（主程序）。

MODFLOW6的GWF模型有3种空间离散方法：DIS (结构网格), DISV (非结构网格), DISU (USG模型没有的）。

**给MODFLOW用户的建议**

MODFLOW6 包含了 MOFLOW-2005，MODFLOW-NWT，MODFLOW-USG 和 MODFLOW-LGR的大部分功能，但MODFLOW6的一些软件包分解、改名和删除了，一些功能由于内存限制不再使用了，如GWF模型quasi-3d confining units。

列举了MODFLOW6与之前版本的MODFLOW的主要差别(Langevin and others (2017))：P16~19，共20条。

输入输出文件和用户与计算机交互方面也有一些不同，见P19~20，共9条。

# GWF模型输入

## 数组输入(READARRAY)

**DATA块的输入指令：**

```
BEGIN DATA
  ARRAY1
    <array1(nval)> -- READARRAY
END DATA
```

**READARRAY控制行**

1. CONSTANT <constant>

With CONSTANT, all values in the array are set equal to `constant`.

2. INTERNAL [FACTOR <factor>] [IPRN <iprn>]

With INTERNAL, the individual array elements will be read from the same file that contains the control line.

3. OPEN/CLOSE <fname> [FACTOR <factor>] [(BINARY)] [IPRN <iprn>]

With OPEN/CLOSE, the array will be read from the file whose name is specified by `fname`. This file will be opened just prior to reading the array and closed immediately after the array is read. A file that is read using this control line can contain only a single array.

# 数组输入(READARRAY)

## READARRAY变量描述

`<constant>`—is a real number constant for real arrays and an integer constant for integer arrays. The `constant` value is assigned to the entire array.

`FACTOR <factor>`—are a keyword and a real number factor for real arrays and an integer factor for integer arrays. The individual elements of the array are multiplied by `factor` after they are read. If `factor` is specified as 0, then it is changed to 1.

`(BINARY)`—is an option that indicates the OPEN/CLOSE file contains array data in binary (unformatted) form. A binary file that can be read by MODFLOW may be created in only two ways. The first way is to use MODFLOW to create the file by saving heads in a binary file. This is commonly done when the user desires to use computed heads from one simulation as initial heads for a subsequent simulation. The other way to create a binary file is to write a special program that generates a binary file. "(BINARY)" can be specified only when the control line is OPEN/CLOSE.

`IPRN <iprn>`—are a keyword and a flag that indicates whether the array being read should be written to the Listing File after the array has been read and a code for indicating the format that should be used when the array is written. The format codes are the same as for MODFLOW-2005. IPRN is set to zero when the specified value exceeds those defined. If IPRN is less than zero or if the keyword and flag are omitted, the array will not be printed.

## READARRAY变量描述

表4 IPRN代码及对应的数组读取格式，这些编码决定了用户提供的数组是如何写到列表文件的。

| IPRN | Real | Integer |
|---|---|---|
| 0 | 10G11.4 | 10I11 |
| 1 | 11G10.3 | 60I1 |
| 2 | 9G13.6 | 40I2 |
| 3 | 15F7.1 | 30I3 |
| 4 | 15F7.2 | 25I4 |
| 5 | 15F7.3 | 20I5 |
| 6 | 15F7.4 | 10I11 |
| 7 | 20F5.0 | 25I2 |
| 8 | 20F5.1 | 15I4 |
| 9 | 20F5.2 | 10I6 |

| IPRN | Real | Integer |
|---|---|---|
| 10 | 20F5.3 | |
| 11 | 20F5.4 | |
| 12 | 10G11.4 | |
| 13 | 10F6.0 | |
| 14 | 10F6.1 | |
| 15 | 10F6.2 | |
| 16 | 10F6.3 | |
| 17 | 10F6.4 | |
| 18 | 10F6.5 | |
| 19 | 5G12.5 | |
| 20 | 6G11.4 | |
| 21 | 7G9.2 | |

# 数组输入(READARRAY)

IC软件的GRIDDATA块

```
BEGIN GRIDDATA
  STRT [LAYERED]
    <strt(nodes)> -- READARRAY
END GRIDDATA
```

如果有可选的LAYERED关键词，则分层给定初始水头条件，适用于DIS和DISV，DISU不设置分层初始值。

```
STRT LAYERED
   CONSTANT 10.0 #layer 1
   CONSTANT 10.0 #layer 2
   CONSTANT 10.0 #layer 3
   CONSTANT 10.0 #layer 4
```

对整个str数组设置初始值，则：

```
STRT
   CONSTANT 10.0 #applies to all cells in the grid
```

# 数组输入(READARRAY)

还有一些数组输入的说明如下：

- List输入
- 长度和时间单位（英制单位和国际单位）
- 恒定态模拟
- 水体收支
- Cell-by-Cell流动

# GWF模型名称文件

## Structure of Blocks

```
BEGIN OPTIONS
  [LIST <list>]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [NEWTON [UNDER_RELAXATION]]
END OPTIONS

BEGIN PACKAGES
  <ftype> <fname> [<pname>]
  <ftype> <fname> [<pname>]
  ...
END PACKAGES
```

## 变量介绍：

**Block: OPTIONS**

`list`—is name of the listing file to create for this GWF model. If not specified, then the name of the list file will be the basename of the GWF model name file and the '.lst' extension. For example, if the GWF name file is called "my.model.nam" then the list file will be called "my.model.lst".

`PRINT_INPUT`—keyword to indicate that the list of all model stress package information will be written to the listing file immediately after it is read.

`PRINT_FLOWS`—keyword to indicate that the list of all model package flow rates will be printed to the listing file for every stress period time step in which "BUDGET PRINT" is specified in Output Control. If there is no Output Control option and "PRINT_FLOWS" is specified, then flow rates are printed for the last time step of each stress period.

`SAVE_FLOWS`—keyword to indicate that all model package flow terms will be written to the file specified with "BUDGET FILEOUT" in Output Control.

`NEWTON`—keyword that activates the Newton-Raphson formulation for groundwater flow between connected, convertible groundwater cells and stress packages that support calculation of Newton-Raphson terms for groundwater exchanges. Cells will not dry when this option is used. By default, the Newton-Raphson formulation is not applied.

`UNDER_RELAXATION`—keyword that indicates whether the groundwater head in a cell will be under-relaxed when water levels fall below the bottom of the model below any given cell. By default, Newton-Raphson UNDER_RELAXATION is not applied.

# GWF模型名称文件

## Structure of Blocks

```
BEGIN OPTIONS
  [LIST <list>]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [NEWTON [UNDER_RELAXATION]]
END OPTIONS

BEGIN PACKAGES
  <ftype> <fname> [<pname>]
  <ftype> <fname> [<pname>]
  ...
END PACKAGES
```

## 变量介绍：

**Block: PACKAGES**

`ftype`—is the file type, which must be one of the following character values shown in table 21. Ftype may be entered in any combination of uppercase and lowercase.

`fname`—is the name of the file containing the package input. The path to the file should be included if the file is not located in the folder where the program was run.

`pname`—is the user-defined name for the package. PNAME is restricted to 16 characters. No spaces are allowed in PNAME. PNAME character values are read and stored by the program for stress packages only. These names may be useful for labeling purposes when multiple stress packages of the same type are located within a single GWF Model. If PNAME is specified for a stress package, then PNAME will be used in the flow budget table in the listing file; it will also be used for the text entry in the cell-by-cell budget file. PNAME is case insensitive and is stored in all upper case letters.

# GWF模型名称文件

| Ftype | Input File Description | Pname |
|---|---|---|
| DIS6 | Rectilinear Discretization Input File | |
| DISV6 | Discretization by Vertices Input File | |
| DISU6 | Unstructured Discretization Input File | |
| IC6 | Initial Conditions Package | |
| OC6 | Output Control Option | |
| NPF6 | Node Property Flow Package | |
| STO6 | Storage Package | |
| CSUB6 | Compaction and Subsidence Package | |
| BUY6 | Buoyancy Package | |
| HFB6 | Horizontal Flow Barrier Package | |
| CHD6 | Time-Variant Specified Head Option | * |
| WEL6 | Well Package | * |
| DRN6 | Drain Package | * |
| RIV6 | River Package | * |
| GHB6 | General-Head Boundary Package | * |
| RCH6 | Recharge Package | * |
| EVT6 | Evapotranspiration Package | * |
| MAW6 | Multi-Aquifer Well Package | * |
| SFR6 | Streamflow Routing Package | * |
| LAK6 | Lake Package | * |
| UZF6 | Unsaturated Zone Flow Package | * |
| MVR6 | Water Mover Package | |
| GNC6 | Ghost-Node Correction Package | |
| OBS6 | Observations Option | |

表5 Ftype值，Pname列表示软件包名称是否以名称文件提供。

## Example Input File

```
# This block is optional
BEGIN OPTIONS
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
END OPTIONS

# List of packages. List can be listed in any order.
BEGIN PACKAGES
  IC6          bcf2ss.ic
  NPF6         bcf2ss.npf
  WEL6         bcf2ss.wel WEL-COUNTY
  RIV6         bcf2ss.riv
  RCH6         bcf2ss.rch
  OC6          bcf2ss.oc
  DIS6         bcf2ss.dis
END PACKAGES
```

# 结构网格离散(DIS)输入

## Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
END OPTIONS

BEGIN DIMENSIONS
  NLAY <nlay>
  NROW <nrow>
  NCOL <ncol>
END DIMENSIONS

BEGIN GRIDDATA
  DELR
    <delr(ncol)> -- READARRAY
  DELC
    <delc(nrow)> -- READARRAY
  TOP
    <top(ncol, nrow)> -- READARRAY
  BOTM [LAYERED]
    <botm(ncol, nrow, nlay)> -- READARRAY
  [IDOMAIN [LAYERED]
    <idomain(ncol, nrow, nlay)> -- READARRAY]
END GRIDDATA
```

## 变量介绍

**Block: OPTIONS**

`length_units`—is the length units used for this model. Values can be "FEET", "METERS", or "CENTIMETERS". If not specified, the default is "UNKNOWN".

`NOGRB`—keyword to deactivate writing of the binary grid file.

`xorigin`—x-position of the lower-left corner of the model grid. A default value of zero is assigned if not specified. The value for XORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`yorigin`—y-position of the lower-left corner of the model grid. If not specified, then a default value equal to zero is used. The value for YORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`angrot`—counter-clockwise rotation angle (in degrees) of the lower-left corner of the model grid. If not specified, then a default value of 0.0 is assigned. The value for ANGROT does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

# 结构网格离散(DIS)输入

## Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
END OPTIONS

BEGIN DIMENSIONS
  NLAY <nlay>
  NROW <nrow>
  NCOL <ncol>
END DIMENSIONS

BEGIN GRIDDATA
  DELR
    <delr(ncol)> -- READARRAY
  DELC
    <delc(nrow)> -- READARRAY
  TOP
    <top(ncol, nrow)> -- READARRAY
  BOTM [LAYERED]
    <botm(ncol, nrow, nlay)> -- READARRAY
  [IDOMAIN [LAYERED]
    <idomain(ncol, nrow, nlay)> -- READARRAY]
END GRIDDATA
```

## 变量介绍

**Block: DIMENSIONS**

`nlay`—is the number of layers in the model grid.

`nrow`—is the number of rows in the model grid.

`ncol`—is the number of columns in the model grid.

**Block: GRIDDATA**

`delr`—is the column spacing in the row direction.

`delc`—is the row spacing in the column direction.

`top`—is the top elevation for each cell in the top model layer.

`botm`—is the bottom elevation for each cell.

`idomain`—is an optional array that characterizes the existence status of a cell. If the IDOMAIN array is not specified, then all model cells exist within the solution. If the IDOMAIN value for a cell is 0, the cell does not exist in the simulation. Input and output values will be read and written for the cell, but internal to the program, the cell is excluded from the solution. If the IDOMAIN value for a cell is 1 or greater, the cell exists in the simulation. If the IDOMAIN value for a cell is -1, the cell does not exist in the simulation. Furthermore, the first existing cell above will be connected to the first existing cell below. This type of cell is referred to as a "vertical pass through" cell.

# 结构网格离散(DIS)输入

- 结构网格的离散信息从定义为DIS6的文件读取。
- 一个模型仅可以定义一种离散输入文件(DISU6, DISV6或者DIS6)。

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
  LENGTH_UNITS METERS
END OPTIONS

#The DIMENSIONS block is required
BEGIN DIMENSIONS
  NLAY 10
  NROW 1
  NCOL 21
END DIMENSIONS

#The GRIDDATA block is required
BEGIN GRIDDATA
  DELR
    INTERNAL FACTOR 1.
        .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 .1 0.01
  DELC
    CONSTANT 1.0
  TOP LAYERED
    CONSTANT 1.
  BOTM LAYERED
    CONSTANT 0.9
    CONSTANT 0.8
    CONSTANT 0.7
    CONSTANT 0.6
    CONSTANT 0.5
    CONSTANT 0.4
    CONSTANT 0.3
    CONSTANT 0.2
    CONSTANT 0.1
    CONSTANT 0.0
END GRIDDATA
```

# 节点离散（DISV）输入

结构网格的离散信息从定义为DISV6的文件读取，一个模型仅可以定义一种离散输入文件(DISU6, DISV6或者DIS6)。

DISV软件包的单元和单元节点的编号如图1。单元的节点列表必须是顺时针顺序。第1个节点号增加到节点列表的最后面，封闭多边形。因此，用户可决定是否封闭单元多边形。

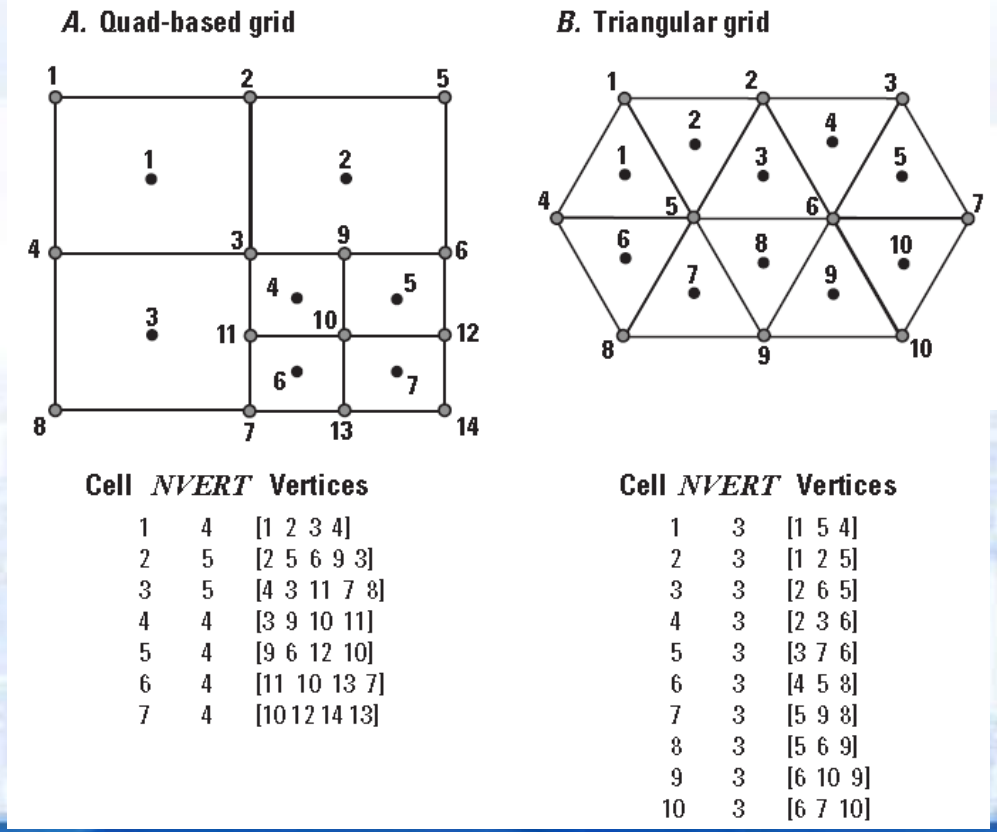


图1 使用节点离散软件包定义的节点和单元的编号示意图

# Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NLAY <nlay>
  NCPL <ncpl>
  NVERT <nvert>
END DIMENSIONS
```

```
BEGIN GRIDDATA
  TOP
    <top(ncpl)> -- READARRAY
  BOTM [LAYERED]
    <botm(nlay, ncpl)> -- READARRAY
  [IDOMAIN [LAYERED]
    <idomain(nlay, ncpl)> -- READARRAY]
END GRIDDATA
```

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES
```

```
BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  ...
END CELL2D
```

# 节点离散（DISV）输入

**Block: OPTIONS**

`length_units`—is the length units used for this model. Values can be "FEET", "METERS", or "CENTIMETERS". If not specified, the default is "UNKNOWN".

`NOGRB`—keyword to deactivate writing of the binary grid file.

`xorigin`—x-position of the origin used for model grid vertices. This value should be provided in a real-world coordinate system. A default value of zero is assigned if not specified. The value for XORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`yorigin`—y-position of the origin used for model grid vertices. This value should be provided in a real-world coordinate system. If not specified, then a default value equal to zero is used. The value for YORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`angrot`—counter-clockwise rotation angle (in degrees) of the model grid coordinate system relative to a real-world coordinate system. If not specified, then a default value of 0.0 is assigned. The value for ANGROT does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

# Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NLAY <nlay>
  NCPL <ncpl>
  NVERT <nvert>
END DIMENSIONS

BEGIN GRIDDATA
  TOP
    <top(ncpl)> -- READARRAY
  BOTM [LAYERED]
    <botm(nlay, ncpl)> -- READARRAY
  [IDOMAIN [LAYERED]
    <idomain(nlay, ncpl)> -- READARRAY]
END GRIDDATA
```

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES

BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  ...
END CELL2D
```

# 节点离散（DISV）输入

**Block: DIMENSIONS**

nlay—is the number of layers in the model grid.

ncpl—is the number of cells per layer. This is a constant value for the grid and it applies to all layers.

nvert—is the total number of (x, y) vertex pairs used to characterize the horizontal configuration of the model grid.

**Block: GRIDDATA**

top—is the top elevation for each cell in the top model layer.

botm—is the bottom elevation for each cell.

idomain—is an optional array that characterizes the existence status of a cell. If the IDOMAIN array is not specified, then all model cells exist within the solution. If the IDOMAIN value for a cell is 0, the cell does not exist in the simulation. Input and output values will be read and written for the cell, but internal to the program, the cell is excluded from the solution. If the IDOMAIN value for a cell is 1 or greater, the cell exists in the simulation. If the IDOMAIN value for a cell is -1, the cell does not exist in the simulation. Furthermore, the first existing cell above will be connected to the first existing cell below. This type of cell is referred to as a "vertical pass through" cell.

# 节点离散（DISV）输入

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xo
  [YORIGIN <yo
  [ANGROT <ang
END OPTIONS
```

```
BEGIN DIMENSIONS
  NLAY <nlay>
  NCPL <ncpl>
  NVERT <nvert>
END DIMENSIONS
```

```
BEGIN GRIDDATA
  TOP
    <top(ncpl)> -- READARRAY
  BOTM [LAYERED]
    <botm(nlay, ncpl)> -- READAR
  [IDOMAIN [LAYERED]
    <idomain(nlay, ncpl)> -- REA
END GRIDDATA
```

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES
```

```
BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  ...
END CELL2D
```

**Block: VERTICES**

`iv`—is the vertex number. Records in the VERTICES block must be listed in consecutive order from 1 to NVERT.

`xv`—is the x-coordinate for the vertex.

`yv`—is the y-coordinate for the vertex.

**Block: CELL2D**

`icell2d`—is the CELL2D number. Records in the CELL2D block must be listed in consecutive order from the first to the last.

`xc`—is the x-coordinate for the cell center.

`yc`—is the y-coordinate for the cell center.

`ncvert`—is the number of vertices required to define the cell. There may be a different number of vertices for each cell.

`icvert`—is an array of integer values containing vertex numbers (in the VERTICES block) used to define the cell. Vertices must be listed in clockwise order. Cells that are connected must share vertices.

# 节点离散（DISV）输入

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
  LENGTH_UNITS METERS
END OPTIONS

#The DIMENSIONS block is required
BEGIN DIMENSIONS
  NCPL 4
  NLAY 3
  NVERT 9
END DIMENSIONS

#The GRIDDATA block is required
BEGIN GRIDDATA
  TOP
    CONSTANT 3.0
  BOTM LAYERED
    CONSTANT 2.0
    CONSTANT 1.0
    CONSTANT 0.0
  IDOMAIN LAYERED
    INTERNAL FACTOR 1
      1 1 1 0
    CONSTANT 1
    CONSTANT 1
END GRIDDATA

#The VERTICES block is required
BEGIN VERTICES
  1 0. 1.
  2 .5 1.
  3 1. 1.
  4 0 .5
  5 .5 .5
  6 1. .5
  7 0. 0.
  8 .5 0.
  9 1. 0.
END VERTICES

BEGIN CELL2D
  1 .25 .75 4 1 2 5 4
  2 .75 .75 4 2 3 6 5
  3 .25 .25 4 4 5 8 7
  4 .75 .25 4 5 6 9 8
END CELL2D
```

# 非结构离散（DISU）输入

结构网格的离散信息从定义为DISU6的文件读取，一个模型仅可以定义一种离散输入文件(DISU6, DISV6或者DIS6)。
使用节点定义各单元的形状和位置。该信息是可选的，仅当DIMENSIONS块中定义了节点数(NVERT)才读取（上述信息），分配给大于零的值。如果文件中提供了节点和2D单元信息，则该信息也写出到二进制网格文件。提供该信息可能对后处理程序读取二进制网格文件有用。
DISU软件不支持分层的概念，与MODFLOW-USG的DISU不同。
在MODFLOW6中，模型的所有网格输入和输出都使用DISU软件包，作为节点数规模的1D数组进入和写出。
所有模拟都不需要DISU VERTICES和CELL2D块。如果在NPF软件包中定义了SAVE_SPECIFIC_DISCHARGE或者使用XT3D，则需要这些块。通常建议包括VERTICES和CELL2D块。

# 非结构离散（DISU）输入

## Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
  [VERTICAL_OFFSET_TOLERANCE <vertical_offset_tolerance>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NODES <nodes>
  NJA <nja>
  [NVERT <nvert>]
END DIMENSIONS
```

```
BEGIN GRIDDATA
  TOP
    <top(nodes)> -- READARRAY
  BOT
    <bot(nodes)> -- READARRAY
  AREA
    <area(nodes)> -- READARRAY
  [IDOMAIN
    <idomain(nodes)> -- READARRAY]
END GRIDDATA
```

```
BEGIN CONNECTIONDATA
  IAC
    <iac(nodes)> -- READARRAY
  JA
    <ja(nja)> -- READARRAY
  IHC
    <ihc(nja)> -- READARRAY
  CL12
    <cl12(nja)> -- READARRAY
  HWVA
    <hwva(nja)> -- READARRAY
  [ANGLDEGX
    <angldegx(nja)> -- READARRAY]
END CONNECTIONDATA
```

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES
```

```
BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  ...
END CELL2D
```

# 非结构离散（DISU）输入

## Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
  [VERTICAL_OFFSET_TOLERANCE <vertical_c
END OPTIONS
```

```
BEGIN DIMENSIONS
  NODES <nodes>
  NJA <nja>
  [NVERT <nvert>]
END DIMENSIONS
```

```
BEGIN GRIDDATA
  TOP
    <top(nodes)> -- READARRAY
  BOT
    <bot(nodes)> -- READARRAY
  AREA
    <area(nodes)> -- READARRAY
  [IDOMAIN
    <idomain(nodes)> -- READARRAY]
END GRIDDATA
```

**Block: OPTIONS**

`length_units`—is the length units used for this model. Values can be "FEET", "METERS", or "CENTIMETERS". If not specified, the default is "UNKNOWN".

`NOGRB`—keyword to deactivate writing of the binary grid file.

`xorigin`—x-position of the origin used for model grid vertices. This value should be provided in a real-world coordinate system. A default value of zero is assigned if not specified. The value for XORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`yorigin`—y-position of the origin used for model grid vertices. This value should be provided in a real-world coordinate system. If not specified, then a default value equal to zero is used. The value for YORIGIN does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`angrot`—counter-clockwise rotation angle (in degrees) of the model grid coordinate system relative to a real-world coordinate system. If not specified, then a default value of 0.0 is assigned. The value for ANGROT does not affect the model simulation, but it is written to the binary grid file so that postprocessors can locate the grid in space.

`vertical_offset_tolerance`—checks are performed to ensure that the top of a cell is not higher than the bottom of an overlying cell. This option can be used to specify the tolerance that is used for checking. If top of a cell is above the bottom of an overlying cell by a value less than this tolerance, then the program will not terminate with an error. The default value is zero. This option should generally not be used.

# 非结构离散（DISU）输入

## Structure of Blocks

```
BEGIN OPTIONS
  [LENGTH_UNITS <length_units>]
  [NOGRB]
  [XORIGIN <xorigin>]
  [YORIGIN <yorigin>]
  [ANGROT <angrot>]
  [VERTICAL_OFFSET_TOLERANCE <ve
END OPTIONS
```

```
BEGIN DIMENSIONS
  NODES <nodes>
  NJA <nja>
  [NVERT <nvert>]
END DIMENSIONS
```

```
BEGIN GRIDDATA
  TOP
    <top(nodes)> -- READARRAY
  BOT
    <bot(nodes)> -- READARRAY
  AREA
    <area(nodes)> -- READARRAY
  [IDOMAIN
    <idomain(nodes)> -- READARRAY]
END GRIDDATA
```

**Block: DIMENSIONS**

`nodes`—is the number of cells in the model grid.

`nja`—is the sum of the number of connections and NODES. When calculating the total number of connections, the connection between cell n and cell m is considered to be different from the connection between cell m and cell n. Thus, NJA is equal to the total number of connections, including n to m and m to n, and the total number of cells.

`nvert`—is the total number of (x, y) vertex pairs used to define the plan-view shape of each cell in the model grid. If NVERT is not specified or is specified as zero, then the VERTICES and CELL2D blocks below are not read. NVERT and the accompanying VERTICES and CELL2D blocks should be specified for most simulations. If the XT3D or SAVE_SPECIFIC_DISCHARGE options are specified in the NPF Package, then this information is required.

**Block: GRIDDATA**

`top`—is the top elevation for each cell in the model grid.

`bot`—is the bottom elevation for each cell.

`area`—is the cell surface area (in plan view).

`idomain`—is an optional array that characterizes the existence status of a cell. If the IDOMAIN array is not specified, then all model cells exist within the solution. If the IDOMAIN value for a cell is 0, the cell does not exist in the simulation. Input and output values will be read and written for the cell, but internal to the program, the cell is excluded from the solution. If the IDOMAIN value for a cell is 1 or greater, the cell exists in the simulation. IDOMAIN values of -1 cannot be specified for the DISU Package.

# 非结构离散（DISU）输入

```
BEGIN CONNECTIONDATA
  IAC
    <iac(nodes)> -- READARRAY
  JA
    <ja(nja)> -- READARRAY
  IHC
    <ihc(nja)> -- READARRAY
  CL12
    <cl12(nja)> -- READARRAY
  HWVA
    <hwva(nja)> -- READARRAY
  [ANGLDEGX
    <angldegx(nja)> -- READARRAY]
END CONNECTIONDATA
```

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES
```

```
BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(n
  <icell2d> <xc> <yc> <ncvert> <icvert(n
  ...
END CELL2D
```

**Block: CONNECTIONDATA**

`iac`—is the number of connections (plus 1) for each cell. The sum of all the entries in IAC must be equal to NJA.

`ja`—is a list of cell number (n) followed by its connecting cell numbers (m) for each of the m cells connected to cell n. The number of values to provide for cell n is IAC(n). This list is sequentially provided for the first to the last cell. The first value in the list must be cell n itself, and the remaining cells must be listed in an increasing order (sorted from lowest number to highest). Note that the cell and its connections are only supplied for the GWF cells and their connections to the other GWF cells. Also note that the JA list input may be divided such that every node and its connectivity list can be on a separate line for ease in readability of the file. To further ease readability of the file, the node number of the cell whose connectivity is subsequently listed, may be expressed as a negative number, the sign of which is subsequently converted to positive by the code.

`ihc`—is an index array indicating the direction between node n and all of its m connections. If IHC = 0 then cell n and cell m are connected in the vertical direction. Cell n overlies cell m if the cell number for n is less than m; cell m overlies cell n if the cell number for m is less than n. If IHC = 1 then cell n and cell m are connected in the horizontal direction. If IHC = 2 then cell n and cell m are connected in the horizontal direction, and the connection is vertically staggered. A vertically staggered connection is one in which a cell is horizontally connected to more than one cell in a horizontal connection.

`cl12`—is the array containing connection lengths between the center of cell n and the shared face with each adjacent m cell.

`hwva`—is a symmetric array of size NJA. For horizontal connections, entries in HWVA are the horizontal width perpendicular to flow. For vertical connections, entries in HWVA are the vertical area for flow. Thus, values in the HWVA array contain dimensions of both length and area. Entries in the HWVA array have a one-to-one correspondence with the connections specified in the JA array. Likewise, there is a one-to-one correspondence between entries in the HWVA array and entries in the IHC array, which specifies the connection type (horizontal or vertical). Entries in the HWVA array must be symmetric; the program will terminate with an error if the value for HWVA for an n to m connection does not equal the value for HWVA for the corresponding n to m connection.

`angldegx`—is the angle (in degrees) between the horizontal x-axis and the outward normal to the face between a cell and its connecting cells. The angle varies between zero and 360.0 degrees, where zero degrees points in the positive x-axis direction, and 90 degrees points in the positive y-axis direction. ANGLDEGX is only needed if horizontal anisotropy is specified in the NPF Package, if the XT3D option is used in the NPF Package, or if the SAVE_SPECIFIC_DISCHARGE option is specifed in the NPF Package. ANGLDEGX does not need to be specified if these conditions are not met. ANGLDEGX is of size NJA; values specified for vertical connections and for the diagonal position are not used. Note that ANGLDEGX is read in degrees, which is different from MODFLOW-USG, which reads a similar variable (ANGLEX) in radians.

# 非结构离散（DISU）输入

```
BEGIN CONNECTIONDATA
  IAC
    <iac(nodes)> -- READARRAY
  JA
    <ja(nja)> -- READARRAY
  IHC
    <ihc(nja)> -- READARRAY
  CL12
    <cl12(nja)> -- READARRAY
  HWVA
    <hwva(nja)> -- READARRAY
  [ANGLDEGX
    <angldegx(nja)> -- READARR
END CONNECTIONDATA
```

**Block: VERTICES**

`iv`—is the vertex number. Records in the VERTICES block must be listed in consecutive order from 1 to NVERT.

`xv`—is the x-coordinate for the vertex.

`yv`—is the y-coordinate for the vertex.

**Block: CELL2D**

`icell2d`—is the cell2d number. Records in the CELL2D block must be listed in consecutive order from 1 to NODES.

`xc`—is the x-coordinate for the cell center.

`yc`—is the y-coordinate for the cell center.

`ncvert`—is the number of vertices required to define the cell. There may be a different number of vertices for each cell.

`icvert`—is an array of integer values containing vertex numbers (in the VERTICES block) used to define the cell. Vertices must be listed in clockwise order.

```
BEGIN VERTICES
  <iv> <xv> <yv>
  <iv> <xv> <yv>
  ...
END VERTICES

BEGIN CELL2D
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  <icell2d> <xc> <yc> <ncvert> <icvert(ncvert)>
  ...
END CELL2D
```

# 非结构离散（DISU）输入

```
BEGIN OPTIONS
  LENGTH_UNITS METERS
END OPTIONS

BEGIN DIMENSIONS
  NODES 9
  NJA 33
END DIMENSIONS

BEGIN GRIDDATA
  TOP
    CONSTANT 0.
  BOT
    CONSTANT -10
  AREA
    INTERNAL FACTOR 1
      10000 10000 10000 10000 10000 10000 10000 10000 10000
END GRIDDATA

BEGIN CONNECTIONDATA
  IHC
    CONSTANT 1
  IAC
    INTERNAL FACTOR 1
    3 4 3 4 5 4 3 4 3
  JA
    INTERNAL FACTOR 1
    1 2 4
    2 1 3 5
    3 2 6
    4 1 5 7
    5 2 4 6 8
    6 3 5 9
    7 4 8
    8 5 7 9
    9 6 8
  CL12
    INTERNAL FACTOR 1
    0 50 50
    0 50 50 50
    0 50 50
    0 50 50 50
    0 50 50 50 50
    0 50 50 50
    0 50 50
    0 50 50 50
    0 50 50
  HWVA
    INTERNAL FACTOR 1
    0 100 100
    0 100 100 100
    0 100 100
    0 100 100 100
    0 100 100 100 100
    0 100 100 100
    0 100 100
    0 100 100 100
    0 100 100
END CONNECTIONDATA
```

# 初始条件(IC)软件包

使用IC6定义的文件类型读取IC软件包信息。一个GWF模型仅定义一个IC软件。

## Structure of Blocks

```
BEGIN GRIDDATA
  STRT [LAYERED]
    <strt(nodes)> -- READARRAY
END GRIDDATA
```

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
END OPTIONS

#The GRIDDATA block is required
BEGIN GRIDDATA
  STRT LAYERED
    CONSTANT  0.0 Initial Head layer 1
    CONSTANT  0.0 Initial Head layer 2
END GRIDDATA
```

Explanation of Variables

**Block: GRIDDATA**

`strt`—is the initial (starting) head—that is, head at the beginning of the GWF Model simulation. STRT must be specified for all simulations, including steady-state simulations. One value is read for every model cell. For simulations in which the first stress period is steady state, the values used for STRT generally do not affect the simulation (exceptions may occur if cells go dry and (or) rewet). The execution time, however, will be less if STRT includes hydraulic heads that are close to the steady-state solution. A head value lower than the cell bottom can be provided if a cell should start as dry.

# Output Control (OC)选项

类型OC6的名称文件定义输出控制选项。如果没有OC6文件，使用默认输出控制。输出控制决定如何以及何时将水头输出到列表文件或写到单独的二进制输出文件。水头的PRINT和SAVE选项，不再有指定单个分层的选项。只要这些数组的一个打印或保存，所有层都打印或保存。

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BUDGET FILEOUT <budgetfile>]
  [BUDGETCSV FILEOUT <budgetcsvfile>]
  [HEAD FILEOUT <headfile>]
  [HEAD PRINT_FORMAT COLUMNS <columns> WIDTH <width> DIGITS <digits> <format>]
END OPTIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  [SAVE <rtype> <ocsetting>]
  [PRINT <rtype> <ocsetting>]
END PERIOD
```

# Output Control (OC)选项

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BUDGET FILEOUT <budgetfile>]
  [BUDGETCSV FILEOUT <budgetcsvfil
  [HEAD FILEOUT <headfile>]
  [HEAD PRINT_FORMAT COLUMNS <colu
END OPTIONS
```

**Block: OPTIONS**

`BUDGET`—keyword to specify that record corresponds to the budget.

`FILEOUT`—keyword to specify that an output filename is expected next.

`budgetfile`—name of the output file to write budget information.

`BUDGETCSV`—keyword to specify that record corresponds to the budget CSV.

`budgetcsvfile`—name of the comma-separated value (CSV) output file to write budget summary information. A budget summary record will be written to this file for each time step of the simulation.

`HEAD`—keyword to specify that record corresponds to head.

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  [SAVE <rtype> <ocsetting>]
  [PRINT <rtype> <ocsetting>]
END PERIOD
```

`headfile`—name of the output file to write head information.

`PRINT_FORMAT`—keyword to specify format for printing to the listing file.

`columns`—number of columns for writing data.

`width`—width for writing each number.

`digits`—number of digits to use for writing a number.

`format`—write format can be EXPONENTIAL, FIXED, GENERAL, or SCIENTIFIC.

# Output Control (OC)选项

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BUDGET FILEOUT <budgetfile>]
  [BUDGETCSV FILEOUT <budgetcsvfile>]
  [HEAD FILEOUT <headfile>]
  [HEAD PRINT_FORMAT COLUMNS <columns> WIDT
END OPTIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  [SAVE <rtype> <ocsetting>]
  [PRINT <rtype> <ocsetting>]
END PERIOD
```

**Block: PERIOD**

**iper**—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

**SAVE**—keyword to indicate that information will be saved this stress period.

**PRINT**—keyword to indicate that information will be printed this stress period.

**rtype**—type of information to save or print. Can be BUDGET or HEAD.

**ocsetting**—specifies the steps for which the data will be saved.

```
ALL
FIRST
LAST
FREQUENCY <frequency>
STEPS <steps(<nstp)>
```

**ALL**—keyword to indicate save for all time steps in period.

**FIRST**—keyword to indicate save for first step in period. This keyword may be used in conjunction with other keywords to print or save results for multiple time steps.

**LAST**—keyword to indicate save for last step in period. This keyword may be used in conjunction with other keywords to print or save results for multiple time steps.

**frequency**—save at the specified time step frequency. This keyword may be used in conjunction with other keywords to print or save results for multiple time steps.

**steps**—save for each step specified in STEPS. This keyword may be used in conjunction with other keywords to print or save results for multiple time steps.

# Output Control (OC)选项

Example Input File

```
BEGIN OPTIONS
  HEAD FILEOUT AdvGW_tidal.hds
  BUDGET FILEOUT AdvGW_tidal.cbc
  HEAD PRINT_FORMAT COLUMNS 100 WIDTH 15 DIGITS 4 GENERAL
END OPTIONS

BEGIN PERIOD 1
  PRINT HEAD FIRST
  PRINT HEAD LAST
  PRINT BUDGET LAST
  SAVE HEAD ALL
  SAVE BUDGET ALL
END PERIOD

# No output for stress periods 2 through 24
BEGIN PERIOD 2
END PERIOD

BEGIN PERIOD 25
  PRINT HEAD STEPS 6 12 23
  SAVE BUDGET FIRST
  SAVE BUDGET LAST
  SAVE BUDGET FREQUENCY 5
END PERIOD
```

# Observation (OBS) Utility for a GWF Model

GWF模型观测值包括：模拟的地下水头(head)，计算的节点上的drawdown以及两个连接节点之间的流动(flow-ja-face)。各GWF模型观测类型需要的数据列于表6。对于flow-ja-face观测类型，负值和正值分别代表对ID定义的cellid的损失和收益。

**Table 6.** Available GWF model observation types.

| Model | Observation type | ID | ID2 | Description |
|---|---|---|---|---|
| GWF | head | cellid | – | Head at a specified cell. |
| GWF | drawdown | cellid | – | Drawdown at a specified cell calculated as difference between starting head and simulated head for the time step. |
| GWF | flow-ja-face | cellid | cellid | Flow between two adjacent cells. |

# Observation (OBS) Utility for a GWF Model

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DIGITS <digits>]
  [PRINT_INPUT]
END OPTIONS
```

```
BEGIN CONTIN
  <obsname>
  <obsname>
  ...
END CONTINUO
```

**Block: OPTIONS**

`digits`—Keyword and an integer digits specifier used for conversion of simulated values to text on output. If not specified, the default is the maximum number of digits stored in the program (as written with the G0 Fortran specifier). When simulated values are written to a comma-separated value text file specified in a CONTINUOUS block below, the digits specifier controls the number of significant digits with which simulated values are written to the output file. The digits specifier has no effect on the number of significant digits with which the simulation time is written for continuous observations. If DIGITS is specified as zero, then observations are written with the default setting, which is the maximum number of digits.

`PRINT_INPUT`—keyword to indicate that the list of observation information will be written to the listing file immediately after it is read.

# Observation (OBS) Utility for a GWF Model

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DIGITS <digits>]
  [PRINT_INPUT]
END OPTIONS
```

```
BEGIN CONTINUOUS FILEOUT <obs_output_file_name> [BINARY]
  <obsname> <obstype> <id> [<id2>]
  <obsname> <obstype> <id> [<id2>]
  ...
END CONTINUOUS
```

**Block: CONTINUOUS**

FILEOUT—keyword to specify that an output filename is expected next.

obs_output_file_name—Name of a file to which simulated values corresponding to observations in the block are to be written. The file name can be an absolute or relative path name. A unique output file must be specified for each CONTINUOUS block. If the "BINARY" option is used, output is written in binary form. By convention, text output files have the extension "csv" (for "Comma-Separated Values") and binary output files have the extension "bsv" (for "Binary Simulated Values").

BINARY—an optional keyword used to indicate that the output file should be written in binary (unformatted) form.

obsname—string of 1 to 40 nonblank characters used to identify the observation. The identifier need not be unique; however, identification and post-processing of observations in the output files are facilitated if each observation is given a unique name.

obstype—a string of characters used to identify the observation type.

id—Text identifying cell where observation is located. For packages other than NPF, if boundary names are defined in the corresponding package input file, ID can be a boundary name. Otherwise ID is a cellid. If the model discretization is type DIS, cellid is three integers (layer, row, column). If the discretization is DISV, cellid is two integers (layer, cell number). If the discretization is DISU, cellid is one integer (node number).

id2—Text identifying cell adjacent to cell identified by ID. The form of ID2 is as described for ID. ID2 is used for intercell-flow observations of a GWF model, for three observation types of the LAK Package, for two observation types of the MAW Package, and one observation type of the UZF Package.

# Horizontal Flow Barrier (HFB) 软件包

从名称文件的HFB6类型的文件读取HFB软件包的输入。

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [PRINT_INPUT]
END OPTIONS

BEGIN DIMENSIONS
  MAXHFB <maxhfb>
END DIMENSIONS
```

*FOR ANY STRESS PE*

```
BEGIN PERIOD <iper>
  <cellid1(ncelldim)> <c
  <cellid1(ncelldim)> <c
  ...
END PERIOD
```

Explanation of Variables

**Block: OPTIONS**

PRINT_INPUT—keyword to indicate that the list of horizontal flow barriers will be written to the listing file immediately after it is read.

**Block: DIMENSIONS**

maxhfb—integer value specifying the maximum number of horizontal flow barriers that will be entered in this input file. The value of MAXHFB is used to allocate memory for the horizontal flow barriers.

# Horizontal Flow Barrier (HFB) 软件包

## Structure of Blocks

*FOR EACH SIMULATI*

```
BEGIN OPTIONS
  [PRINT_INPUT]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXHFB <maxhfb>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid1(ncelldim)> <cellid2
  <cellid1(ncelldim)> <cellid2
  ...
END PERIOD
```

**Block: PERIOD**

`iper`—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

`cellid1`—identifier for the first cell. For a structured grid that uses the DIS input file, CELLID1 is the layer, row, and column numbers of the cell. For a grid that uses the DISV input file, CELLID1 is the layer number and CELL2D number for the two cells. If the model uses the unstructured discretization (DISU) input file, then CELLID1 is the node numbers for the cell. The barrier is located between cells designated as CELLID1 and CELLID2. For models that use the DIS and DISV grid types, the layer number for CELLID1 and CELLID2 must be the same. For all grid types, cells must be horizontally adjacent or the program will terminate with an error.

`cellid2`—identifier for the second cell. See CELLID1 for description of how to specify.

`hydchr`—is the hydraulic characteristic of the horizontal-flow barrier. The hydraulic characteristic is the barrier hydraulic conductivity divided by the width of the horizontal-flow barrier. If the hydraulic characteristic is negative, then the absolute value of HYDCHR acts as a multiplier to the conductance between the two model cells specified as containing the barrier. For example, if the value for HYDCHR was specified as -1.5, the conductance calculated for the two cells would be multiplied by 1.5.

# Horizontal Flow Barrier (HFB) 软件包

## Example Input File

```
BEGIN OPTIONS
  PRINT_INPUT
END OPTIONS

BEGIN DIMENSIONS
  MAXHFB 1
END DIMENSIONS

BEGIN PERIOD 1
  #L1 R1 C1 L2 R2 C2 HYDCHR
    1  1  4  1  1  5 0.1
END PERIOD 1
```

# Storage软件包

从名称文件的STO6类型读取STO软件包的输入。如果模型不考虑STO软件，则不计算储水变化，模型将是恒定态。一个GWF模型仅定义一个STO软件。

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [SAVE_FLOWS]
  [STORAGECOEFFICIENT]
  [SS_CONFINED_ONLY]
  [TVS6 FILEIN <tvs_filename>]
END OPTIONS
```

```
BEGIN GRIDDATA
  ICONVERT [LAYERED]
    <iconvert(nodes)> -- READARRAY
  SS [LAYERED]
    <ss(nodes)> -- READARRAY
  SY [LAYERED]
    <sy(nodes)> -- READARRAY
END GRIDDATA
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  [STEADY-STATE]
  [TRANSIENT]
END PERIOD
```

**Block: OPTIONS**

SAVE_FLOWS—keyword to indicate that cell-by-cell flow terms will be written to the file specified with "BUDGET SAVE FILE" in Output Control.

STORAGECOEFFICIENT—keyword to indicate that the SS array is read as storage coefficient rather than specific storage.

SS_CONFINED_ONLY—keyword to indicate that specific storage is only calculated when a cell is under confined conditions (head greater than or equal to the top of the cell). This option is identical to the approach used to calculate storage changes under confined conditions in MODFLOW-2005.

TVS6—keyword to specify that record corresponds to a time-varying storage (TVS) file. The behavior of TVS and a description of the input file is provided separately.

FILEIN—keyword to specify that an input filename is expected next.

tvs_filename—defines a time-varying storage (TVS) input file. Records in the TVS file can be used to change specific storage and specific yield properties at specified times or stress periods.

# Storage软件包

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [SAVE_FLOWS]
  [STORAGECOEFFICIENT]
  [SS_CONFINED_ONLY]
  [TVS6 FILEIN <tvs_filename>]
END OPTIONS
```

```
BEGIN GRIDDATA
  ICONVERT [LAYERED]
    <iconvert(nodes)> -- READARRAY
  SS [LAYERED]
    <ss(nodes)> -- READARRAY
  SY [LAYERED]
    <sy(nodes)> -- READARRAY
END GRIDDATA
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  [STEADY-STATE]
  [TRANSIENT]
END PERIOD
```

**Block: GRIDDATA**

`iconvert`—is a flag for each cell that specifies whether or not a cell is convertible for the storage calculation. 0 indicates confined storage is used. >0 indicates confined storage is used when head is above cell top and a mixed formulation of unconfined and confined storage is used when head is below cell top.

`ss`—is specific storage (or the storage coefficient if STORAGECOEFFICIENT is specified as an option). Specific storage values must be greater than or equal to 0. If the CSUB Package is included in the GWF model, specific storage must be zero for every cell.

`sy`—is specific yield. Specific yield values must be greater than or equal to 0. Specific yield does not have to be specified if there are no convertible cells (ICONVERT=0 in every cell).

**Block: PERIOD**

`iper`—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

`STEADY-STATE`—keyword to indicate that stress period IPER is steady-state. Steady-state conditions will apply until the TRANSIENT keyword is specified in a subsequent BEGIN PERIOD block. If the CSUB Package is included in the GWF model, only the first and last stress period can be steady-state.

`TRANSIENT`—keyword to indicate that stress period IPER is transient. Transient conditions will apply until the STEADY-STATE keyword is specified in a subsequent BEGIN PERIOD block.

# Storage软件包

## Example Input File

```
BEGIN OPTIONS
  SAVE_FLOWS
END OPTIONS

BEGIN GRIDDATA
  #cell storage conversion 0:confined, 1:convertible
  ICONVERT
    constant 1
  #specific storage (for all model cells)
  SS
    constant 1.e-5
  #specific yield (specified by layer because of LAYERED keyword)
  SY LAYERED
    constant 0.2
    constant 0.15
    constant 0.15
END GRIDDATA

BEGIN PERIOD 1
  STEADY-STATE
END PERIOD
```

```
BEGIN PERIOD 2
  TRANSIENT
END PERIOD

#stress period 3 will be transient because
#a BEGIN PERIOD block is not provided.

BEGIN PERIOD 4
  STEADY-STATE
END PERIOD
```

# 随时间变化的存储(TVS)软件包

在STO软件包的OPTIONS块的TVS6记录定义，读入TVS软件的输入。

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DISABLE_STORAGE_CHANGE_INTEGRATION]
  [PRINT_INPUT]
  [TS6 FILEIN <ts6_filename>]
END OPTIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <tvssetting>
  <cellid(ncelldim)> <tvssetting>
  ...
END PERIOD
```

**Block: OPTIONS**

DISABLE_STORAGE_CHANGE_INTEGRATION—keyword that deactivates inclusion of storage derivative terms in the STO package matrix formulation. In the absence of this keyword (the default), the groundwater storage formulation will be modified to correctly adjust heads based on transient variations in stored water volumes arising from changes to SS and SY properties.

PRINT_INPUT—keyword to indicate that information for each change to a storage property in a cell will be written to the model listing file.

TS6—keyword to specify that record corresponds to a time-series file.

FILEIN—keyword to specify that an input filename is expected next.

ts6_filename—defines a time-series file defining time series that can be used to assign time-varying values. See the "Time-Variable Input" section for instructions on using the time-series capability.

# 随时间变化的存储(TVS)软件包

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DISABLE_STORAGE_CHANGE_INTEGRATION]
  [PRINT_INPUT]
  [TS6 FILEIN <ts6_filename>]
END OPTIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <tvssetting>
  <cellid(ncelldim)> <tvssetting>
  ...
END PERIOD
```

**Block: PERIOD**

**iper**—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

**cellid**—is the cell identifier, and depends on the type of grid that is used for the simulation. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column. For a grid that uses the DISV input file, CELLID is the layer and CELL2D number. If the model uses the unstructured discretization (DISU) input file, CELLID is the node number for the cell.

**tvssetting**—line of information that is parsed into a property name keyword and values. Property name keywords that can be used to start the TVSSETTING string include: SS and SY.

```
SS <ss>
SY <sy>
```

**ss**—is the new value to be assigned as the cell's specific storage (or storage coefficient if the STORAGECOEFFICIENT STO package option is specified) from the start of the specified stress period, as per SS in the STO package. Specific storage values must be greater than or equal to 0. If the OPTIONS block includes a TS6 entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

**sy**—is the new value to be assigned as the cell's specific yield from the start of the specified stress period, as per SY in the STO package. Specific yield values must be greater than or equal to 0. If the OPTIONS block includes a TS6 entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

# 随时间变化的存储(TVS)软件包

输入文件举例：

```
BEGIN OPTIONS
  TS6 FILEIN tvs_cells.ts
  # Note: Time-series file tvs_cells.ts defines time series cells_sy
END OPTIONS

# Cell 45 will have its SS value changed to 1e-6 in the first time step of
# stress period 2, and changed once more to 1e-7 in the first time step of
# stress period 4.
#
# Cells 188 and 291 will have their respective SY values changed according
# to the time series cells_sy specified in the file tvs_cells.ts. Note that
# these values may continue to change beyond stress period 2, depending on
# the duration of the time series cells_sy.
#
# No changes are made in stress period 1 due to an absence of a block
# for that period; cells maintain the initial property values specified in
# the STO package for the entirety of that period.

BEGIN PERIOD 2
  45  SS  1e-6
  188 SY  cells_sy
  291 SY  cells_sy
END PERIOD

BEGIN PERIOD 4
  45  SS  1e-7
END PERIOD

# After the last specified change (or after the last specified time record,
# when a time series is used), each affected cell will retain its latest
# changed value for the remainder of the simulation.
```

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

从名称文件的CSUB6类型文件读取CSUB软件包的输入。CSUB软件包的技术细节参考Hughes et al., 2022b。如果模型中不考虑CSUB软件，则不计算由于密实导致的储水变化。一个GWF模型仅定义一个CSUB软件包。当GWF模型中使用CSUB软件时，仅第一个和最后一个stress period可以在STO软件包中的STEADY-STATE定义。对各单元，在STO软件包中必须也要单位储水量SS定义为零。

Hughes, J.D., Leake, S.A., Galloway, D.L., and White, J.T., 2022b, Documentation for the Skeletal Storage, Compaction, and Subsidence (CSUB) Package of MODFLOW 6: U.S. Geological Survey Techniques and Methods, book 6, chap. A62, 57 p.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BOUNDNAMES]
  [PRINT_INPUT]
  [SAVE_FLOWS]
  [GAMMAW <gammaw>]
  [BETA <beta>]
  [HEAD_BASED]
  [INITIAL_PRECONSOLIDATION_HEAD]
  [NDELAYCELLS <ndelaycells>]
  [COMPRESSION_INDICES]
  [UPDATE_MATERIAL_PROPERTIES]
  [CELL_FRACTION]
  [SPECIFIED_INITIAL_INTERBED_STATE]
  [SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS]
  [SPECIFIED_INITIAL_DELAY_HEAD]
  [EFFECTIVE_STRESS_LAG]
  [STRAIN_CSV_INTERBED FILEOUT <interbedstrain_filenam
  [STRAIN_CSV_COARSE FILEOUT <coarsestrain_filename>]
  [COMPACTION FILEOUT <compaction_filename>]
  [COMPACTION_ELASTIC FILEOUT <elastic_compaction_file
  [COMPACTION_INELASTIC FILEOUT <inelastic_compaction_
  [COMPACTION_INTERBED FILEOUT <interbed_compaction_fil
  [COMPACTION_COARSE FILEOUT <coarse_compaction_filenam
  [ZDISPLACEMENT FILEOUT <zdisplacement_filename>]
  [PACKAGE_CONVERGENCE FILEOUT <package_convergence_filename>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NINTERBEDS <ninterbeds>
  [MAXSIG0 <maxsig0>]
END DIMENSIONS
```

```
BEGIN GRIDDATA
  CG_SKE_CR
    <cg_ske_cr(nodes)> -- READARRAY
  CG_THETA
    <cg_theta(nodes)> -- READARRAY
  [SGM
    <sgm(nodes)> -- READARRAY]
  [SGS
    <sgs(nodes)> -- READARRAY]
END GRIDDATA
```

```
BEGIN PACKAGEDATA
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <theta> <kv> <h0> [<boundname>]
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <theta> <kv> <h0> [<boundname>]
  ...
END PACKAGEDATA
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <sig0>
  <cellid(ncelldim)> <sig0>
  ...
END PERIOD
```

注：这是一个挺复杂的软件包。

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

在PERIOD块中的所有驱动力软件信息，将继续应用于接下来的驱动周期，直到模拟结束，或直到遇到其他PERIOD块。当遇到新的PERIOD块时，使用新的PERIOD块中的驱动力代替之前的块中的所有驱动力。注意：该行为与一些软件包（MAW, SFR, LAK, UZF）不同。为了关闭一个驱动周期的所有驱动力，一个PERIOD块必须不能定义有入口(be specified with no entry?)。如果一个PERIOD块不在第一个驱动周期定义，则直到文件的第一个PERIOD块的iper值之前都不会施加驱动力。

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BOUNDNAMES]
  [PRINT_INPUT]
  [SAVE_FLOWS]
  [GAMMAW <gammaw>]
  [BETA <beta>]
  [HEAD_BASED]
  [INITIAL_PRECONSOLIDATION_HEAD]
  [NDELAYCELLS <ndelaycells>]
  [COMPRESSION_INDICES]
  [UPDATE_MATERIAL_PROPERTIES]
  [CELL_FRACTION]
  [SPECIFIED_INITIAL_INTERBED_STATE]
  [SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS]
  [SPECIFIED_INITIAL_DELAY_HEAD]
  [EFFECTIVE_STRESS_LAG]
  [STRAIN_CSV_INTERBED FILEOUT <interbedstrain_filename>]
  [STRAIN_CSV_COARSE FILEOUT <coarsestrain_filename>]
  [COMPACTION FILEOUT <compaction_filename>]
  [COMPACTION_ELASTIC FILEOUT <elastic_compaction_filename>]
  [COMPACTION_INELASTIC FILEOUT <inelastic_compaction_filename>]
  [COMPACTION_INTERBED FILEOUT <interbed_compaction_filename>]
  [COMPACTION_COARSE FILEOUT <coarse_compaction_filename>]
  [ZDISPLACEMENT FILEOUT <zdisplacement_filename>]
  [PACKAGE_CONVERGENCE FILEOUT <package_convergence_filename>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

**Block: OPTIONS**

BOUNDNAMES—keyword to indicate that boundary names may be provided with the list of CSUB cells.

PRINT_INPUT—keyword to indicate that the list of CSUB information will be written to the listing file immediately after it is read.

SAVE_FLOWS—keyword to indicate that cell-by-cell flow terms will be written to the file specified with "BUDGET SAVE FILE" in Output Control.

gammaw—unit weight of water. For freshwater, GAMMAW is 9806.65 Newtons/cubic meters or 62.48 lb/cubic foot in SI and English units, respectively. By default, GAMMAW is 9806.65 Newtons/cubic meters.

beta—compressibility of water. Typical values of BETA are 4.6512e-10 1/Pa or 2.2270e-8 lb/square foot in SI and English units, respectively. By default, BETA is 4.6512e-10 1/Pa.

HEAD_BASED—keyword to indicate the head-based formulation will be used to simulate coarse-grained aquifer materials and no-delay and delay interbeds. Specifying HEAD_BASED also specifies the INITIAL_PRECONSOLIDATION_HEAD option.

INITIAL_PRECONSOLIDATION_HEAD—keyword to indicate that preconsolidation heads will be specified for no-delay and delay interbeds in the PACKAGEDATA block. If the SPECIFIED_INITIAL_INTERBED_STATE option is specified in the OPTIONS block, user-specified preconsolidation heads in the PACKAGEDATA block are absolute values. Otherwise, user-specified preconsolidation heads in the PACKAGEDATA block are relative to steady-state or initial heads.

ndelaycells—number of nodes used to discretize delay interbeds. If not specified, then a default value of 19 is assigned.

COMPRESSION_INDICES—keyword to indicate that the recompression (CR) and compression (CC) indices are specified instead of the elastic specific storage (SSE) and inelastic specific storage (SSV) coefficients. If not specified, then elastic specific storage (SSE) and inelastic specific storage (SSV) coefficients must be specified.

UPDATE_MATERIAL_PROPERTIES—keyword to indicate that the thickness and void ratio of coarse-grained and interbed sediments (delay and no-delay) will vary during the simulation. If not specified, the thickness and void ratio of coarse-grained and interbed sediments will not vary during the simulation.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BOUNDNAMES]
  [PRINT_INPUT]
  [SAVE_FLOWS]
  [GAMMAW <gammaw>]
  [BETA <beta>]
  [HEAD_BASED]
  [INITIAL_PRECONSOLIDATION_HEAD]
  [NDELAYCELLS <ndelaycells>]
  [COMPRESSION_INDICES]
  [UPDATE_MATERIAL_PROPERTIES]
  [CELL_FRACTION]
  [SPECIFIED_INITIAL_INTERBED_STATE]
  [SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS]
  [SPECIFIED_INITIAL_DELAY_HEAD]
  [EFFECTIVE_STRESS_LAG]
  [STRAIN_CSV_INTERBED FILEOUT <interbedstrain_filename>]
  [STRAIN_CSV_COARSE FILEOUT <coarsestrain_filename>]
  [COMPACTION FILEOUT <compaction_filename>]
  [COMPACTION_ELASTIC FILEOUT <elastic_compaction_filename>]
  [COMPACTION_INELASTIC FILEOUT <inelastic_compaction_filename>]
  [COMPACTION_INTERBED FILEOUT <interbed_compaction_filename>]
  [COMPACTION_COARSE FILEOUT <coarse_compaction_filename>]
  [ZDISPLACEMENT FILEOUT <zdisplacement_filename>]
  [PACKAGE_CONVERGENCE FILEOUT <package_convergence_filename>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

## 接上张PPT的说明：

CELL_FRACTION—keyword to indicate that the thickness of interbeds will be specified in terms of the fraction of cell thickness. If not specified, interbed thickness must be specified.

SPECIFIED_INITIAL_INTERBED_STATE—keyword to indicate that absolute preconsolidation stresses (heads) and delay bed heads will be specified for interbeds defined in the PACKAGEDATA block. The SPECIFIED_INITIAL_INTERBED_STATE option is equivalent to specifying the SPECIFIED_INITIAL_PRECONSOLITATION_STRESS and SPECIFIED_INITIAL_DELAY_HEAD. If SPECIFIED_INITIAL_INTERBED_STATE is not specified then preconsolidation stress (head) and delay bed head values specified in the PACKAGEDATA block are relative to simulated values of the first stress period if steady-state or initial stresses and GWF heads if the first stress period is transient.

SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS—keyword to indicate that absolute preconsolidation stresses (heads) will be specified for interbeds defined in the PACKAGEDATA block. If SPECIFIED_INITIAL_PRECONSOLITATION_STRESS and SPECIFIED_INITIAL_INTERBED_STATE are not specified then preconsolidation stress (head) values specified in the PACKAGEDATA block are relative to simulated values if the first stress period is steady-state or initial stresses (heads) if the first stress period is transient.

SPECIFIED_INITIAL_DELAY_HEAD—keyword to indicate that absolute initial delay bed head will be specified for interbeds defined in the PACKAGEDATA block. If SPECIFIED_INITIAL_DELAY_HEAD and SPECIFIED_INITIAL_INTERBED_STATE are not specified then delay bed head values specified in the PACKAGEDATA block are relative to simulated values if the first stress period is steady-state or initial GWF heads if the first stress period is transient.

EFFECTIVE_STRESS_LAG—keyword to indicate the effective stress from the previous time step will be used to calculate specific storage values. This option can 1) help with convergence in models with thin cells and water table elevations close to land surface; 2) is identical to the approach used in the SUBWT package for MODFLOW-2005; and 3) is only used if the effective-stress formulation is being used. By default, current effective stress values are used to calculate specific storage values.

STRAIN_CSV_INTERBED—keyword to specify the record that corresponds to final interbed strain output.

FILEOUT—keyword to specify that an output filename is expected next.

interbedstrain_filename—name of the comma-separated-values output file to write final interbed strain information.

STRAIN_CSV_COARSE—keyword to specify the record that corresponds to final coarse-grained material strain output.

coarsestrain_filename—name of the comma-separated-values output file to write final coarse-grained material strain information.

COMPACTION—keyword to specify that record corresponds to the compaction.

compaction_filename—name of the binary output file to write compaction information.

COMPACTION_ELASTIC—keyword to specify that record corresponds to the elastic interbed compaction binary file.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [BOUNDNAMES]
  [PRINT_INPUT]
  [SAVE_FLOWS]
  [GAMMAW <gammaw>]
  [BETA <beta>]
  [HEAD_BASED]
  [INITIAL_PRECONSOLIDATION_HEAD]
  [NDELAYCELLS <ndelaycells>]
  [COMPRESSION_INDICES]
  [UPDATE_MATERIAL_PROPERTIES]
  [CELL_FRACTION]
  [SPECIFIED_INITIAL_INTERBED_STATE]
  [SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS]
  [SPECIFIED_INITIAL_DELAY_HEAD]
  [EFFECTIVE_STRESS_LAG]
  [STRAIN_CSV_INTERBED FILEOUT <interbedstrain_filename>]
  [STRAIN_CSV_COARSE FILEOUT <coarsestrain_filename>]
  [COMPACTION FILEOUT <compaction_filename>]
  [COMPACTION_ELASTIC FILEOUT <elastic_compaction_filename>]
  [COMPACTION_INELASTIC FILEOUT <inelastic_compaction_filename>]
  [COMPACTION_INTERBED FILEOUT <interbed_compaction_filename>]
  [COMPACTION_COARSE FILEOUT <coarse_compaction_filename>]
  [ZDISPLACEMENT FILEOUT <zdisplacement_filename>]
  [PACKAGE_CONVERGENCE FILEOUT <package_convergence_filename>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

## 接上张PPT的说明：

**elastic_compaction_filename**—name of the binary output file to write elastic interbed compaction information.

**COMPACTION_INELASTIC**—keyword to specify that record corresponds to the inelastic interbed compaction binary file.

**inelastic_compaction_filename**—name of the binary output file to write inelastic interbed compaction information.

**COMPACTION_INTERBED**—keyword to specify that record corresponds to the interbed compaction binary file.

**interbed_compaction_filename**—name of the binary output file to write interbed compaction information.

**COMPACTION_COARSE**—keyword to specify that record corresponds to the elastic coarse-grained material compaction binary file.

**coarse_compaction_filename**—name of the binary output file to write elastic coarse-grained material compaction information.

**ZDISPLACEMENT**—keyword to specify that record corresponds to the z-displacement binary file.

**zdisplacement_filename**—name of the binary output file to write z-displacement information.

**PACKAGE_CONVERGENCE**—keyword to specify that record corresponds to the package convergence comma spaced values file.

**package_convergence_filename**—name of the comma spaced values output file to write package convergence information.

**TS6**—keyword to specify that record corresponds to a time-series file.

**FILEIN**—keyword to specify that an input filename is expected next.

**ts6_filename**—defines a time-series file defining time series that can be used to assign time-varying values. See the "Time-Variable Input" section for instructions on using the time-series capability.

**OBS6**—keyword to specify that record corresponds to an observations file.

**obs6_filename**—name of input file to define observations for the CSUB package. See the "Observation utility" section for instructions for preparing observation input files. Tables 31 and 32 lists observation type(s) supported by the CSUB package.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

```
BEGIN DIMENSIONS
  NINTERBEDS <ninterbeds>
  [MAXSIGO <maxsig0>]
END DIMENSIONS
```

```
BEGIN GRIDDATA
  CG_SKE_CR
    <cg_ske_cr(nodes)> -- READARRAY
  CG_THETA
    <cg_theta(nodes)> -- READARRAY
  [SGM
    <sgm(nodes)> -- READARRAY]
  [SGS
    <sgs(nodes)> -- READARRAY]
END GRIDDATA
```

**Block: DIMENSIONS**

**ninterbeds**—is the number of CSUB interbed systems. More than 1 CSUB interbed systems can be assigned to a GWF cell; however, only 1 GWF cell can be assigned to a single CSUB interbed system.

**maxsig0**—is the maximum number of cells that can have a specified stress offset. More than 1 stress offset can be assigned to a GWF cell. By default, MAXSIG0 is 0.

**Block: GRIDDATA**

**cg_ske_cr**—is the initial elastic coarse-grained material specific storage or recompression index. The recompression index is specified if COMPRESSION_INDICES is specified in the OPTIONS block. Specified or calculated elastic coarse-grained material specific storage values are not adjusted from initial values if HEAD_BASED is specified in the OPTIONS block.

**cg_theta**—is the initial porosity of coarse-grained materials.

**sgm**—is the specific gravity of moist or unsaturated sediments. If not specified, then a default value of 1.7 is assigned.

**sgs**—is the specific gravity of saturated sediments. If not specified, then a default value of 2.0 is assigned.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

```
BEGIN PACKAGEDATA
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <the
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <the
  ...
END PACKAGEDATA
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <sig0>
  <cellid(ncelldim)> <sig0>
  ...
END PERIOD
```

**Block: PACKAGEDATA**

icsubno—integer value that defines the CSUB interbed number associated with the specified PACKAGEDATA data on the line. CSUBNO must be greater than zero and less than or equal to NINTERBEDS. CSUB information must be specified for every CSUB cell or the program will terminate with an error. The program will also terminate with an error if information for a CSUB interbed number is specified more than once.

cellid—is the cell identifier, and depends on the type of grid that is used for the simulation. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column. For a grid that uses the DISV input file, CELLID is the layer and CELL2D number. If the model uses the unstructured discretization (DISU) input file, CELLID is the node number for the cell.

cdelay—character string that defines the subsidence delay type for the interbed. Possible subsidence package CDELAY strings include: NODELAY–character keyword to indicate that delay will not be simulated in the interbed. DELAY–character keyword to indicate that delay will be simulated in the interbed.

pcs0—is the initial offset from the calculated initial effective stress or initial preconsolidation stress in the interbed, in units of height of a column of water. PCS0 is the initial preconsolidation stress if SPECIFIED_INITIAL_INTERBED_STATE or SPECIFIED_INITIAL_PRECONSOLIDATION_STRESS are specified in the OPTIONS block. If HEAD_BASED is specified in the OPTIONS block, PCS0 is the initial offset from the calculated initial head or initial preconsolidation head in the CSUB interbed and the initial preconsolidation stress is calculated from the calculated initial effective stress or calculated initial geostatic stress, respectively.

thick_frac—is the interbed thickness or cell fraction of the interbed. Interbed thickness is specified as a fraction of the cell thickness if CELL_FRACTION is specified in the OPTIONS block.

rnb—is the interbed material factor equivalent number of interbeds in the interbed system represented by the interbed. RNB must be greater than or equal to 1 if CDELAY is DELAY. Otherwise, RNB can be any value.

ssv_cc—is the initial inelastic specific storage or compression index of the interbed. The compression index is specified if COMPRESSION_INDICES is specified in the OPTIONS block. Specified or calculated interbed inelastic specific storage values are not adjusted from initial values if HEAD_BASED is specified in the OPTIONS block.

sse_cr—is the initial elastic coarse-grained material specific storage or recompression index of the interbed. The recompression index is specified if COMPRESSION_INDICES is specified in the OPTIONS block. Specified or calculated interbed elastic specific storage values are not adjusted from initial values if HEAD_BASED is specified in the OPTIONS block.

theta—is the initial porosity of the interbed.

kv—is the vertical hydraulic conductivity of the delay interbed. KV must be greater than 0 if CDELAY is DELAY. Otherwise, KV can be any value.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

```
BEGIN PACKAGEDATA
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <theta> <kv> <h0> [<boundname>]
  <icsubno> <cellid(ncelldim)> <cdelay> <pcs0> <thick_frac> <rnb> <ssv_cc> <sse_cr> <theta> <kv> <h0> [<boundname>]
  ...
END PACKAGEDATA
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <sig0>
  <cellid(ncelldim)> <sig0>
  ...
END PERIOD
```

接上张**PPT**的说明：

h0—is the initial offset from the head in cell cellid or the initial head in the delay interbed. H0 is the initial head in the delay bed if SPECIFIED_INITIAL_INTERBED_STATE or SPECIFIED_INITIAL_DELAY_HEAD are specified in the OPTIONS block. H0 can be any value if CDELAY is NODELAY.

boundname—name of the CSUB cell. BOUNDNAME is an ASCII character variable that can contain as many as 40 characters. If BOUNDNAME contains spaces in it, then the entire name must be enclosed within single quotes.

**Block: PERIOD**

iper—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

cellid—is the cell identifier, and depends on the type of grid that is used for the simulation. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column. For a grid that uses the DISV input file, CELLID is the layer and CELL2D number. If the model uses the unstructured discretization (DISU) input file, CELLID is the node number for the cell.

sig0—is the stress offset for the cell. SIG0 is added to the calculated geostatic stress for the cell. SIG0 is specified only if MAXSIG0 is specified to be greater than 0 in the DIMENSIONS block. If the Options block includes a TIMESERIESFILE entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

# Skeletal Storage, Compaction, Subsidence (CSUB)软件包

## Example Input File

```
BEGIN OPTIONS
  COMPRESSION_INDICES
  SPECIFIED_INITIAL_INTERBED_STATE
  BOUNDNAMES
  SAVE_FLOWS
END OPTIONS

BEGIN DIMENSIONS
  NINTERBEDS 4
  MAXSIGO 1
END DIMENSIONS

BEGIN GRIDDATA
  # compression indices of coarse grained aquifer material
  cg_ske_cr LAYERED
    CONSTANT      0.01
    CONSTANT      0.01
    CONSTANT      0.01
    CONSTANT      0.01
  # porosity of coarse grained aquifer materials
  cg_theta LAYERED
    CONSTANT      0.45
    CONSTANT      0.45
    CONSTANT      0.45
    CONSTANT      0.45
  # specific gravity of saturated sediment
  SGS LAYERED
      CONSTANT 2.0
      CONSTANT 2.0
      CONSTANT 2.0
      CONSTANT 2.0
  # specific gravity of moist sediment
  SGM LAYERED
      CONSTANT 1.7
      CONSTANT 1.7
      CONSTANT 1.7
      CONSTANT 1.7
END GRIDDATA

BEGIN PACKAGEDATA
# icsubsno cellid cdelay  pcs0  thick_frac rnb ssv_cc sse_cr theta  kv  h0 boundname
         1 1 1 6  delay   15.0      0.450 1.0  0.25   0.01   0.45 0.1 15. nsystm0
         2 1 1 7  nodelay 15.0      0.450 1.0  0.25   0.01   0.45 0.0 0.0 nsystm1
         3 1 1 8  nodelay 15.0      0.450 1.0  0.25   0.01   0.45 0.0 0.0 nsystm1
         4 1 1 9  delay   15.0      0.450 1.0  0.25   0.01   0.45 0.1 15. nsystm2
END PACKAGEDATA

BEGIN PERIOD 1
# stress offset for stress period 1
  1 1 6   1700.00000000
END PERIOD
```

# 浮力(BUY)软件

变密度的地下水流模拟，。。。

# 恒定水头(CHD)软件

在名称文件中从CHD6类型文件读取CHD软件包的输入。单个GWF模型可以有任意个CHD软件。但是，当CHD软件要设置一个GWF单元为恒定水头单元时，此时该单元在CHD软件包中已经分配为恒定水位单元，会发生错误。

之前版本的MODFLOW，不能将恒定水头单位转换为激活的单元。一旦一个单元分配为恒定水头单元，直到模拟结束前都是恒定水头单元。在MODFLOW6中，在接下来的驱动周期中，一个单元不是恒定水头单元，则之前的恒定水头单元可以转变为激活的单元。

在MODFLOW6中，在任何驱动周期内，任何恒定水头单元只能设定为一个水头值。必须使用时间序列功能来插值到单个时间步长上。

# 恒定水头(CHD)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <head> [<aux(naux)>] [<boundname>]
  <cellid(ncelldim)> <head> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

## Explanation of Variables

**Block: OPTIONS**

**auxiliary**—defines an array of one or more auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided on this line; however, lists of information provided in subsequent blocks must have a column of data for each auxiliary variable name defined here. The number of auxiliary variables detected on this line determines the value for naux. Comments cannot be provided anywhere on this line as they will be interpreted as auxiliary variable names. Auxiliary variables may not be used by the package, but they will be available for use by other parts of the program. The program will terminate with an error if auxiliary variables are specified on more than one line in the options block.

**auxmultname**—name of auxiliary variable to be used as multiplier of CHD head value.

**BOUNDNAMES**—keyword to indicate that boundary names may be provided with the list of constant-head cells.

**PRINT_INPUT**—keyword to indicate that the list of constant-head information will be written to the listing file immediately after it is read.

**PRINT_FLOWS**—keyword to indicate that the list of constant-head flow rates will be printed to the listing file for every stress period time step in which "BUDGET PRINT" is specified in Output Control. If there is no Output Control option and "PRINT_FLOWS" is specified, then flow rates are printed for the last time step of each stress period.

**SAVE_FLOWS**—keyword to indicate that constant-head flow terms will be written to the file specified with "BUDGET FILEOUT" in Output Control.

**TS6**—keyword to specify that record corresponds to a time-series file.

**FILEIN**—keyword to specify that an input filename is expected next.

**ts6_filename**—defines a time-series file defining time series that can be used to assign time-varying values. See the "Time-Variable Input" section for instructions on using the time-series capability.

**OBS6**—keyword to specify that record corresponds to an observations file.

**obs6_filename**—name of input file to define observations for the constant-head package. See the "Observation utility" section for instructions for preparing observation input files. Tables 31 and 32 lists observation type(s) supported by the constant-head package.

# 恒定水头(CHD)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <head> [<aux(naux)>] [<boundn
  <cellid(ncelldim)> <head> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

**Block: DIMENSIONS**

maxbound—integer value specifying the maximum number of constant-head cells that will be specified for use during any stress period.

**Block: PERIOD**

iper—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

cellid—is the cell identifier, and depends on the type of grid that is used for the simulation. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column. For a grid that uses the DISV input file, CELLID is the layer and CELL2D number. If the model uses the unstructured discretization (DISU) input file, CELLID is the node number for the cell.

head—is the head at the boundary. If the Options block includes a TIMESERIESFILE entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

aux—represents the values of the auxiliary variables for each constant head. The values of auxiliary variables must be present for each constant head. The values must be specified in the order of the auxiliary variables specified in the OPTIONS block. If the package supports time series and the Options block includes a TIMESERIESFILE entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

boundname—name of the constant head boundary cell. BOUNDNAME is an ASCII character variable that can contain as many as 40 characters. If BOUNDNAME contains spaces in it, then the entire name must be enclosed within single quotes.

# 恒定水头(CHD)软件

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
  AUXILIARY temperature
  BOUNDNAMES
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
END OPTIONS

#The DIMENSIONS block is required
BEGIN DIMENSIONS
  MAXBOUND 2
END DIMENSIONS

#The following block of constant-head cells will be activated
#for stress period 1. This block will remain active throughout
#the simulation.

BEGIN PERIOD 1
#l r c head  temperature boundname
 1 1 2 100.  20.5        chd_1_2
 1 1 3 100.  20.4        chd_1_3
END PERIOD 1
```

# 井(WEL)软件

在名称文件中的WEL6类型文件读取井(WEL)软件包的输入。单个GWF模型可以定义任意数目的WEL软件。

Structure of Blocks

*FOR EACH SIMULATIC*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(na
  [AUXMULTNAME <auxmultnam
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [AUTO_FLOW_REDUCE <auto_flow_reduce>]
  [AUTO_FLOW_REDUCE_CSV FILEOUT <afrcsvfile>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS

BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname>]
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

All of the stress package information in the PERIOD block will continue to apply for subsequent stress periods until the end of the simulation, or until another PERIOD block is encountered. When a new PERIOD block is encountered, all of the stresses from the previous block are replaced with the stresses in the new PERIOD block. Note that this behavior is different from the advanced packages (MAW, SFR, LAK, and UZF). To turn off all of the stresses for a stress period, a PERIOD block must be specified with no entries. If a PERIOD block is not specified for the first stress period, then no stresses will be applied until the `iper` value of the first PERIOD block in the file.

# 井(WEL)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [AUTO_FLOW_REDUCE <auto_flow_reduce>]
  [AUTO_FLOW_REDUCE_CSV FILEOUT <afrcsvfile>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname>]
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

Explanation of Variables

**Block: OPTIONS**

**auxiliary**—defines an array of one or more auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided on this line; however, lists of information provided in subsequent blocks must have a column of data for each auxiliary variable name defined here. The number of auxiliary variables detected on this line determines the value for naux. Comments cannot be provided anywhere on this line as they will be interpreted as auxiliary variable names. Auxiliary variables may not be used by the package, but they will be available for use by other parts of the program. The program will terminate with an error if auxiliary variables are specified on more than one line in the options block.

**auxmultname**—name of auxiliary variable to be used as multiplier of well flow rate.

**BOUNDNAMES**—keyword to indicate that boundary names may be provided with the list of well cells.

**PRINT_INPUT**—keyword to indicate that the list of well information will be written to the listing file immediately after it is read.

**PRINT_FLOWS**—keyword to indicate that the list of well flow rates will be printed to the listing file for every stress period time step in which "BUDGET PRINT" is specified in Output Control. If there is no Output Control option and "PRINT_FLOWS" is specified, then flow rates are printed for the last time step of each stress period.

**SAVE_FLOWS**—keyword to indicate that well flow terms will be written to the file specified with "BUDGET FILEOUT" in Output Control.

**auto_flow_reduce**—keyword and real value that defines the fraction of the cell thickness used as an interval for smoothly adjusting negative pumping rates to 0 in cells with head values less than or equal to the bottom of the cell. Negative pumping rates are adjusted to 0 or a smaller negative value when the head in the cell is equal to or less than the calculated interval above the cell bottom. AUTO_FLOW_REDUCE is set to 0.1 if the specified value is less than or equal to zero. By default, negative pumping rates are not reduced during a simulation.

**AUTO_FLOW_REDUCE_CSV**—keyword to specify that record corresponds to the AUTO_FLOW_REDUCE output option in which a new record is written for each well and for each time step in which the user-requested extraction rate is reduced by the program.

**FILEOUT**—keyword to specify that an output filename is expected next.

**afrcsvfile**—name of the comma-separated value (CSV) output file to write information about well extraction rates that have been reduced by the program. Entries are only written if the extraction rates are reduced.

**TS6**—keyword to specify that record corresponds to a time-series file.

**FILEIN**—keyword to specify that an input filename is expected next.

**ts6_filename**—defines a time-series file defining time series that can be used to assign time-varying values. See the "Time-Variable Input" section for instructions on using the time-series capability.

**OBS6**—keyword to specify that record corresponds to an observations file.

**obs6_filename**—name of input file to define observations for the Well package. See the "Observation utility" section for instructions for preparing observation input files. Tables 31 and 32 lists observation type(s) supported by the Well package.

**MOVER**—keyword to indicate that this instance of the Well Package can be used with the Water Mover (MVR) Package. When the MOVER option is specified, additional memory is allocated within the package to store the available, provided, and received water.

# 井(WEL)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [AUTO_FLOW_REDUCE <auto_flow_reduce>]
  [AUTO_FLOW_REDUCE_CSV FILEOUT <afrcsvfile>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname
  <cellid(ncelldim)> <q> [<aux(naux)>] [<boundname
  ...
END PERIOD
```

**Block: DIMENSIONS**

maxbound—integer value specifying the maximum number of wells cells that will be specified for use during any stress period.

**Block: PERIOD**

iper—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

cellid—is the cell identifier, and depends on the type of grid that is used for the simulation. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column. For a grid that uses the DISV input file, CELLID is the layer and CELL2D number. If the model uses the unstructured discretization (DISU) input file, CELLID is the node number for the cell.

q—is the volumetric well rate. A positive value indicates recharge (injection) and a negative value indicates discharge (extraction). If the Options block includes a TIMESERIESFILE entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

aux—represents the values of the auxiliary variables for each well. The values of auxiliary variables must be present for each well. The values must be specified in the order of the auxiliary variables specified in the OPTIONS block. If the package supports time series and the Options block includes a TIMESERIESFILE entry (see the "Time-Variable Input" section), values can be obtained from a time series by entering the time-series name in place of a numeric value.

boundname—name of the well cell. BOUNDNAME is an ASCII character variable that can contain as many as 40 characters. If BOUNDNAME contains spaces in it, then the entire name must be enclosed within single quotes.

# 井(WEL)软件

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
  AUXILIARY depth screen_length
  BOUNDNAMES
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
END OPTIONS

#The DIMENSIONS block is required
BEGIN DIMENSIONS
  MAXBOUND 5
END DIMENSIONS

#The following block of wells will be activated for stress periods
#2 and 3. No wells are present in stress period 1 due to an
#absence of a block for that period.
BEGIN PERIOD 2
    #layer row col      Q    depth screen_length boundname

        #wells 1 and 2
       7  102   17 -19000   275.9         17.6      CW_1
       9  192   44 -13000   280.0         24.0      CW_2

        #wells 3 through 5
       9  109   67 -24000   295.1         12.1      CW_3
      10   43   17 -12000   301.3          9.6      CW_4
      11   12   17 -17000   315.0         18.6      CW_5
END PERIOD

#Turn off all wells for stress period 4
BEGIN PERIOD 4
  #An empty block indicates that there are no wells.
END PERIOD

#For stress period 5, turn on wells 1 and 4,
#and add three wells that are grouped in a well field
BEGIN PERIOD 5
    #layer row col      Q    depth screen_length boundname
        7  102   17 -19000   275.9         17.6      CW_1
       10   43   17 -12000   301.3          9.6      CW_4

    #wells in well field
        5   27   50 -11000   190.0         20.0 well_field
        5   27   51 -10000   185.0         20.0 well_field
        5   28   50 -12000   187.3         15.0 well_field
END PERIOD

#Use a list of wells in ASCII file wells_sp6.txt for stress period 6.
#Use these wells until the end of the simulation.
BEGIN PERIOD 6
  OPEN/CLOSE wells_sp6.txt
END PERIOD
```

# 井(WEL)软件

## Available observation types

Well Package observations include the simulated well rates (`wel`), the well discharge that is available for the MVR package (`to-mvr`), and the reduction in the specified `q` when the `AUTO_FLOW_REDUCE` option is enabled. The data required for each WEL Package observation type is defined in table 10. The sum of `wel` and `to-mvr` is equal to the simulated well discharge rate, which may be less than the specified `q` if the `AUTO_FLOW_REDUCE` option is enabled. The `DNODATA` value is returned if the `wel-reduction` observation is specified but the `AUTO_FLOW_REDUCE` option is not enabled. Negative and positive values for an observation represent a loss from and gain to the GWF model, respectively.

**Table 10.** Available WEL Package observation types.

| Stress Package | Observation type | ID | ID2 | Description |
|---|---|---|---|---|
| WEL | wel | cellid or boundname | – | Flow between the groundwater system and a well boundary or a group of well boundaries. |
| WEL | to-mvr | cellid or boundname | – | Well boundary discharge that is available for the MVR package for a well boundary or a group of well boundaries. |
| WEL | wel-reduction | cellid or boundname | – | Reduction in the specified well boundary discharge calculated when the `AUTO_FLOW_REDUCE` option is specified. |

# 井(WEL)软件

## Example Observation Input File

```
BEGIN OPTIONS
  DIGITS 7
  PRINT_INPUT
END OPTIONS

BEGIN CONTINUOUS FILEOUT my_model.wel.obs.csv
# obsname          obstype ID
  wel-7-102-17     WEL     7  102  17
  wel-7-102-17     WEL     CW_1
  well-field       WEL     well_field
END CONTINUOUS
```

# 排水(DRN)软件

DRAIN (DRN)软件用来模拟由于农业灌溉、泉水和其他特征引起的，从含水层中以一定的含水层中水头差或一些固定水头或高度有关的速率比例抽取水，只要含水层水头高于排水高度。但是，当含水层水头低于抽取高程，排水对含水层没有影响。比例常数称为排水传导度。

# 排水(DRN)软件

从名称文件中的DRN6类型的文件读取DRN软件的输入。一个GWF模型可以定义任意数目的DRN软件。

PERIOD块中的所有驱动软件信息，将继续施加到接下来的驱动周期，直到模拟结束，或直到遇到其他的PERIOD块。

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [AUXDEPTHNAME <auxdepthname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS
```

**Block: OPTIONS**

auxiliary—defines an array of one or more auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided on this line; however, lists of information provided in subsequent blocks must have a column of data for each auxiliary variable name defined here. The number of auxiliary variables detected on this line determines the value for naux. Comments cannot be provided anywhere on this line as they will be interpreted as auxiliary variable names. Auxiliary variables may not be used by the package, but they will be available for use by other parts of the program. The program will terminate with an error if auxiliary variables are specified on more than one line in the options block.

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

**Block: DIMENSIONS**

maxbound—integer value specifying the maximum number of drains cells that will be specified for use during any stress period.

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <elev> <cond> [<aux(naux)>] [<boun
  <cellid(ncelldim)> <elev> <cond> [<aux(naux)>] [<boun
  ...
END PERIOD
```

**Block: PERIOD**

iper—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

# 排水(DRN)软件

## Example Input File

```
#The OPTIONS block is optional
BEGIN OPTIONS
  BOUNDNAMES
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
END OPTIONS

#The DIMENSIONS block is required
BEGIN DIMENSIONS
  MAXBOUND 5
END DIMENSIONS

#The following block of drains will be activated for for the entire stress period
BEGIN PERIOD 1
  #node elevation conductance boundname
       73       10.2           1000.       my_drn
       76       10.2           1000.       my_drn
       79       10.2           1000.       my_drn
       80       10.2           1000.       my_drn
       81       10.2           1000.       my_drn
END PERIOD
```

## Example Observation Input File

```
BEGIN OPTIONS
  DIGITS 8
  PRINT_INPUT
END OPTIONS

BEGIN CONTINUOUS FILEOUT my_model.drn01.csv
# obsname     obstype  ID
  drn_73      DRN      73
  drn_79      DRN      79
END CONTINUOUS

BEGIN CONTINUOUS FILEOUT my_model.drn02.csv
# obsname    obstype  ID
  drn_80     DRN      80
  drn_all    DRN      my_drn
END CONTINUOUS
```

# 河流(RIV)软件

RIV软件包不模拟河流内的地表水流动，仅考虑河流-含水层渗流。如果河流内的流动是重要的，则使用Streamflow Routine软件（SFR）。因此，河段编号的顺序对RIV软件的计算结果没有影响，河道渗流对每个河段是独立计算的。

从名称文件中的RIV6类型的文件读取RIV软件的输入。一个GWF模型可以定义任意数目的RIV 软件。

SFR 软件包有3个局限性（参考原理手册）。某些情况下，需要与地表水水动力模型耦合（如求解浅水方程）。

# 河流(RIV)软件

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS
```

**Block: OPTIONS**

`auxiliary`—defines an array of one or more auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided on this line; however, lists of information provided in subsequent blocks must have a column of data for each auxiliary variable name defined here. The number of auxiliary variables detected on this line determines the value for naux. Comments cannot be provided anywhere on this line as they will be interpreted as auxiliary variable names. Auxiliary variables may not be used by the package, but they will be available for use by other parts of the program. The program will terminate with an error if auxiliary variables are specified on more than one line in the options block.

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

**Block: DIMENSIONS**

`maxbound`—integer value specifying the maximum number of rivers cells that will be specified for use during any stress period.

**Block: PERIOD**

`iper`—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <stage> <cond> <rbot> [<aux(naux)>] [<boundname>]
  <cellid(ncelldim)> <stage> <cond> <rbot> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

# 河流(RIV)软件

## Example Input File

```
BEGIN OPTIONS
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
  BOUNDNAMES
  TS6 FILEIN river_stages.ts
END OPTIONS

begin dimensions
  MAXBOUND 20
end dimensions

BEGIN PERIOD 1
# layer  row   col   stage          cond    rbot  BoundName
      1    3     1   river_stage_1  1001.   35.9
      1    4     2   river_stage_1  1002.   35.8
      1    5     3   river_stage_1  1003.   35.7
      1    5     4   river_stage_1  1004.   35.6
      1    6     5   river_stage_1  1005.   35.5
      1    6     6   river_stage_1  1006.   35.4  riv1_c6
      1    6     7   river_stage_1  1007.   35.3  riv1_c7
      1    5     8   river_stage_1  1008.   35.2
      1    5     9   river_stage_1  1009.   35.1
      1    5    10   river_stage_1  1010.   35.0
      1   10     1   river_stage_2  1001.   36.9  riv2_upper
      1    9     2   river_stage_2  1002.   36.8  riv2_upper
      1    8     3   river_stage_2  1003.   36.7  riv2_upper
      1    7     4   river_stage_2  1004.   36.6
      1    7     5   river_stage_2  1005.   36.5
      1    6     6   river_stage_2  1006.   36.4  riv2_c6
      1    6     7   river_stage_2  1007.   36.3  riv2_c7
      1    7     8   river_stage_2  1008.   36.2
      1    7     9   river_stage_2  1009.   36.1
      1    7    10   river_stage_2  1010.   36.0
END PERIOD
```

## Available observation types

River Package observations include the simulated river flow rates (`riv`) and the river discharge that is available for the MVR package (`to-mvr`). The data required for each RIV Package observation type is defined in table 12. The sum of `riv` and `to-mvr` is equal to the simulated river flow rate. Negative and positive values for an observation represent a loss from and gain to the GWF model, respectively.

## Example Observation Input File

```
BEGIN OPTIONS
  DIGITS 7
  PRINT_INPUT
END OPTIONS

BEGIN CONTINUOUS FILEOUT my_model.riv.csv
# obsname       type  ID
  rv1-5-4       RIV   1    5    4
  rv1-6-5       RIV   1    6    5
  rv1-c7        RIV   riv1_c7     # flow at boundary "riv1_c7"
  rv2-7-4       RIV   1    7    4
  rv2-8-5       RIV   1    7    5
  rv2-9-6       RIV   1    6    6
END CONTINUOUS

BEGIN CONTINUOUS FILEOUT my_model.riv.flows.csv
# obsname       type  ID
  rv1-3-1       RIV   1    3    1
  rv1-4-2       RIV   1    4    2
  rv1-5-3       RIV   1    5    3
  rv1-c6        RIV   riv1_c6
  rv2-upper     RIV   riv2_upper
END CONTINUOUS
```

# 统一的水头边界(GHB)软件

从名称文件中的GHB6类型的文件读取GHB软件的输入。一个GWF模型可以定义任意数目的GHB软件。

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [AUXMULTNAME <auxmultname>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
END OPTIONS
```

```
BEGIN DIMENSIONS
  MAXBOUND <maxbound>
END DIMENSIONS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <cellid(ncelldim)> <bhead> <cond> [<aux(naux)>] [<boundname>]
  <cellid(ncelldim)> <bhead> <cond> [<aux(naux)>] [<boundname>]
  ...
END PERIOD
```

**Block: OPTIONS**

`auxiliary`—defines an array of one or more auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided on this line; however, lists of information provided in subsequent blocks must have a column of data for each auxiliary variable name defined here. The number of auxiliary variables detected on this line determines the value for naux. Comments cannot be provided anywhere on this line as they will be interpreted as auxiliary variable names. Auxiliary variables may not be used by the package, but they will be available for use by other parts of the program. The program will terminate with an error if auxiliary variables are specified on more than one line in the options block.

**Block: DIMENSIONS**

`maxbound`—integer value specifying the maximum number of general-head boundary cells that will be specified for use during any stress period.

**Block: PERIOD**

`iper`—integer value specifying the starting stress period number for which the data specified in the PERIOD block apply. IPER must be less than or equal to NPER in the TDIS Package and greater than zero. The IPER value assigned to a stress period block must be greater than the IPER value assigned for the previous PERIOD block. The information specified in the PERIOD block will continue to apply for all subsequent stress periods, unless the program encounters another PERIOD block.

# 统一的水头边界(GHB)软件

## Example Input File

```
BEGIN OPTIONS
  PRINT_INPUT (echo input to listing file)
  PRINT_FLOWS (print the flows to the listing
  TS6 FILEIN tides.ts
  BOUNDNAMES
END OPTIONS

# Dimensions block
BEGIN DIMENSIONS
  MAXBOUND 15
END DIMENSIONS

# Stress period block(s)
BEGIN PERIOD 1
#Lay Row Col  Bhead Cond   boundname
  2   1   10   tides   15.0   Estuary-L2
  2   2   10   tides   15.0   Estuary-L2
  2   3   10   tides   15.0   Estuary-L2
  2   4   10   tides   15.0   Estuary-L2
  2   5   10   tides   15.0   Estuary-L2
  2   6   10   tides   15.0   Estuary-L2
  2   7   10   tides   15.0   Estuary-L2
  2   8   10   tides   15.0   Estuary-L2
  2   9   10   tides   15.0   Estuary-L2
  2  10   10   tides   15.0   Estuary-L2
  2  11   10   tides   15.0   Estuary-L2
  2  12   10   tides   15.0   Estuary-L2
  2  13   10   tides   15.0   Estuary-L2
  2  14   10   tides   15.0   Estuary-L2
  2  15   10   tides   15.0   Estuary-L2
END PERIOD
```

## Available observation types

General-Head Boundary Package observations include the simulated general-head boundary flow rates (`ghb`) and the general-head boundary discharge that is available for the MVR package (`to-mvr`). The data required for each GHB Package observation type is defined in table 13. The sum of `ghb` and `to-mvr` is equal to the simulated general-head boundary flow rate. Negative and positive values for an observation represent a loss from and gain to the GWF model, respectively.

## Example Observation Input File

```
BEGIN OPTIONS
  DIGITS 7
  PRINT_INPUT
END OPTIONS

BEGIN CONTINUOUS FILEOUT my_model.ghb.obs.csv
# obsname     obstype  ID
  ghb-2-6-10  GHB      2   6   10
  ghb-2-7-10  GHB      2   7   10
END CONTINUOUS

BEGIN CONTINUOUS FILEOUT my_model.ghb.flows.csv
# obsname       obstype  ID
  Estuary2      GHB      Estuary-L2
END CONTINUOUS
```

# 补水(RCH)软件

# 蒸散发(EVT)软件—基于列表的输入

# 蒸散发(EVT)软件—基于数组的输入

# 河道径流调蓄(SFR)软件

从名称文件中的SFR6类型的文件读取SFR软件的输入。一个GWF模型可以定义任意数目的SFR软件。但是，单独的软件包中的河段之间不能计算水流，除了使用MVR软件计算各软件包之间的水流传输。河段可定义为宽矩形横断面或使用测站编号-高度点的不规则横断面。不规则横断面在Streamflow Routing Package Cross-Section Table Input File一节讨论。

# 河道径流调蓄(SFR)软件

必须显式地定义河段连接关系，程序可更方便地验证河网的连接关系。假设的河网的河段连接如图2。

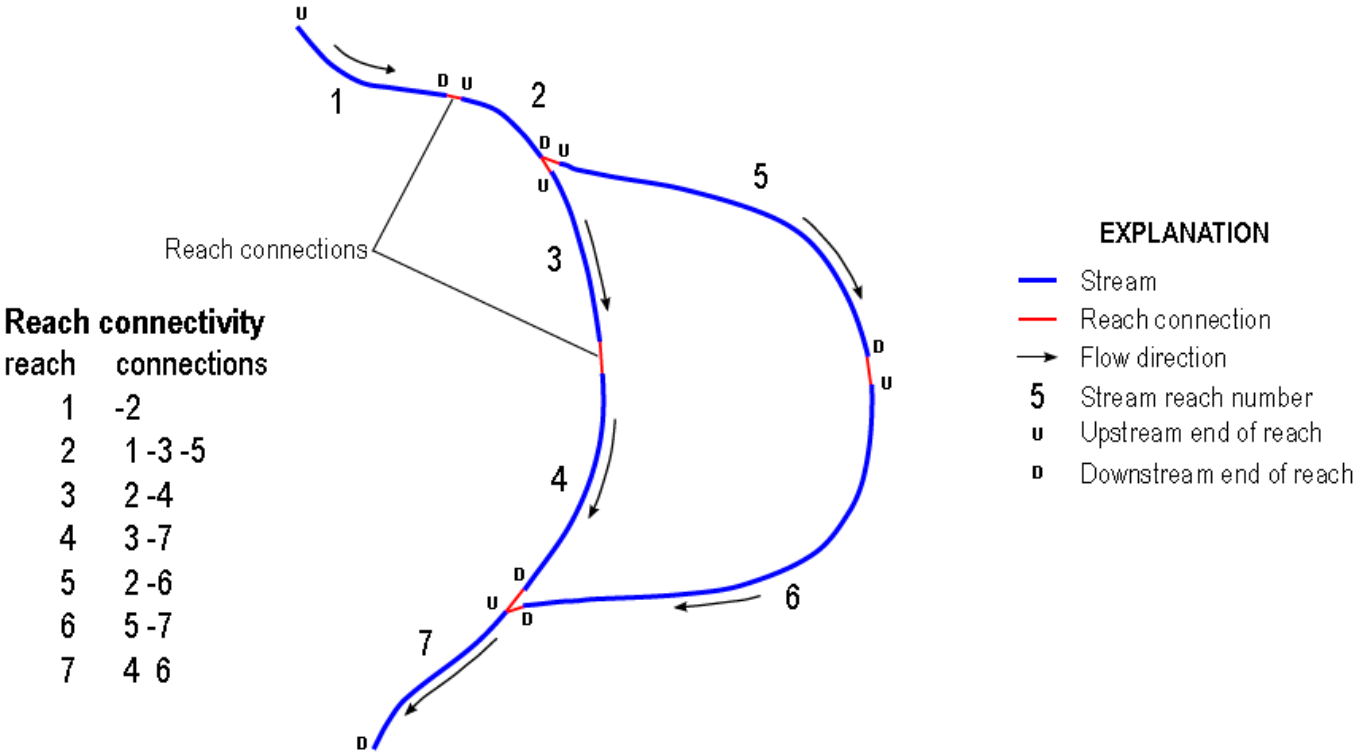


**Figure 2.** Simple stream network having seven reaches with a junction having two reaches, a confluence of two reaches, and the resulting reach connectivity. Downstream connections for a reach must include the reach as an upstream connection for all downstream connections to the reach. Downstream connections for a reach are denoted with a negative reach number.

# 河道径流调蓄(SFR)软件

这个软件包的输入文件很复杂，需要Python脚本程序来创建：SFRMaker

## Example Input File

```
BEGIN OPTIONS
  UNIT_CONVERSION 1.486
  BOUNDNAMES
  PRINT_STAGE
  PRINT_FLOWS
  STAGE FILEOUT sfr-1.stage.bin
  BUDGET FILEOUT sfr-1.cbc
END OPTIONS

#dimension block is required
BEGIN DIMENSIONS
  NREACHES 37
END DIMENSIONS

BEGIN PACKAGEDATA
#rno k  i  j  rlen rwid      rgrd      rtp   rbth     rhk   man  ncon ustrf ndv  boundna
   1 1  1  1  4500. 12    8.67E-04 1093.048  3.0 0.00003 0.03    1   1.0    0    reach
   2 1  2  2  7000. 12    8.67E-04 1088.059  3.0 0.00003 0.03    2   1.0    0    reach
   3 1  3  3  6000. 12    8.67E-04 1082.419  3.0 0.00003 0.03    2   1.0    0    reach
   4 1  3  4  5550. 12    8.67E-04 1077.408  3.0 0.00003 0.03    3   1.0    1    reach4
   5 1  4  5  6500. 12    9.43E-04 1071.934  3.0 0.00003 0.03    2   1.0    0
```

```
BEGIN CONNECTIONDATA
#rno ic1 ic2 ic3
   1  -2
   2   1  -3
   3   2  -4
   4   3  -5 -10
```

```
END CONNECTIONDATA

BEGIN DIVERSIONS
#  rno idv iconr cprior
     4   1    10 UPTO
END DIVERSIONS

BEGIN PERIOD 1
# rno sfrsetting
    1 inflow 25.
   16 inflow 10.
   28 inflow 150.
```

# 径流路径软件的断面输入文件

不规则横断面的测站-高程数据使用xfraction和高度数据定义（图3），使用河段的底部高程（RTP）转换为使用指定河段宽度（REID）和高程的测站位置。

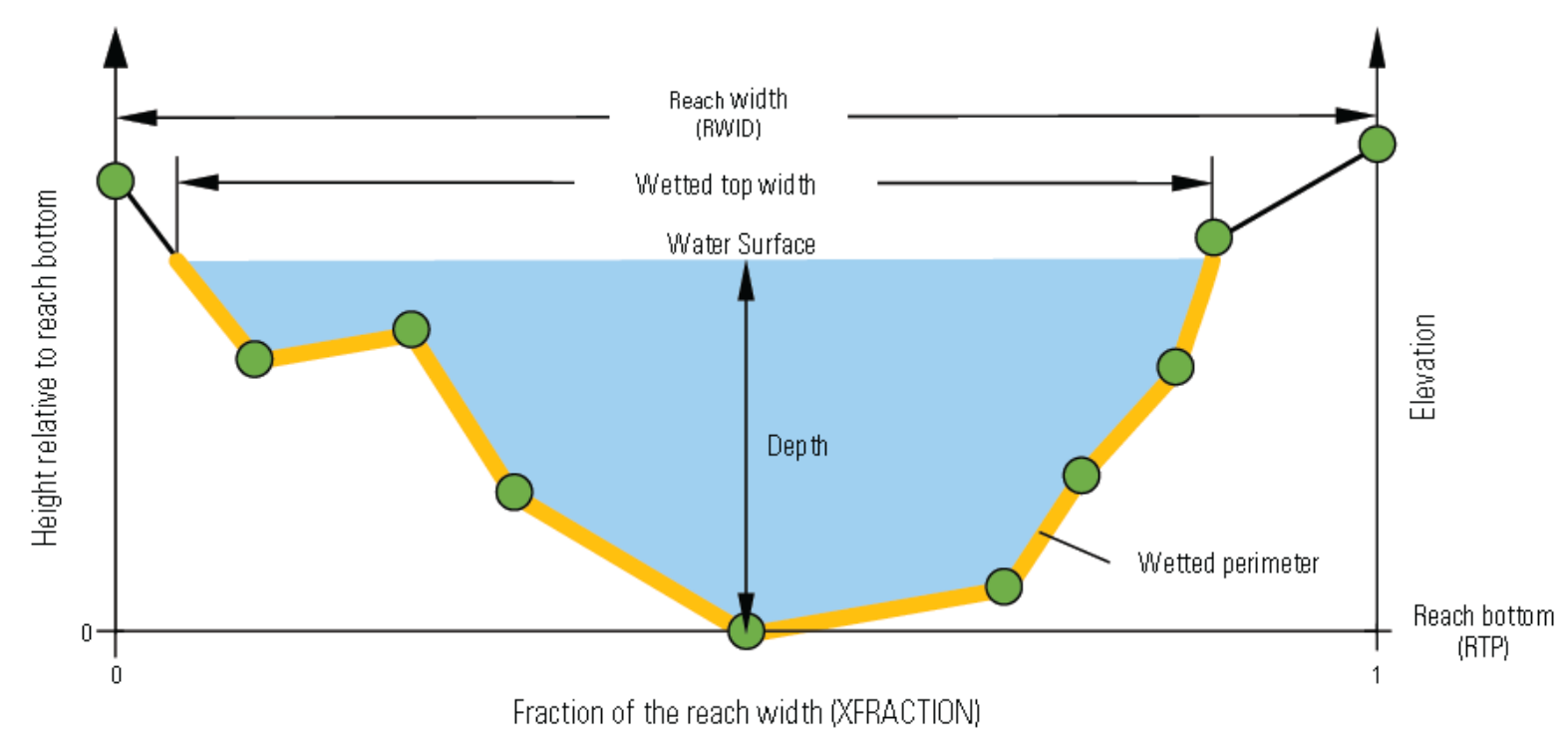


**Figure 3.** Irregular cross section used to compute depth, wetted top width, wetted perimeter, and wetted cross-sectional area for a stream reach for the case where the maximum XFRACTION is one.

# 径流路径软件的断面输入文件

## Structure of Blocks

```
BEGIN DIMENSIONS
  NROW <nrow>
  NCOL <ncol>
END DIMENSIONS
```

```
BEGIN TABLE
  <xfraction> <height> [<manfraction>]
  <xfraction> <height> [<manfraction>]
  ...
END TABLE
```

## Example Input File

```
begin dimensions
  nrow 11
  ncol  3
end dimensions

begin table
# xfraction  height   manfraction
        0.0     1.0          10.0
        0.1     1.0          10.0
        0.2     1.0           1.0
        0.3     0.0           1.0
        0.4     0.0           1.0
        0.5     0.0           1.0
        0.6     0.0           1.0
        0.7     0.0           1.0
        0.8     1.0          10.0
        0.9     1.0          10.0
        1.0     1.0         999.0 #any value can be used for manfraction
end table
```

# 蒸散发(EVT)软件—基于数组的输入

# 湖泊(LAK)软件

在名称文件中读取LAK6类型的文件，LAK软件的输入。一个GWF模型可以定义任意数目的LAK软件。

Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_STAGE]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [STAGE FILEOUT <stagefile>]
  [BUDGET FILEOUT <budgetfile>]
  [BUDGETCSV FILEOUT <budgetcsvfile>]
  [PACKAGE_CONVERGENCE FILEOUT <package_convergence_filename>]
  [TS6 FILEIN <ts6_filename>]
  [OBS6 FILEIN <obs6_filename>]
  [MOVER]
  [SURFDEP <surfdep>]
  [TIME_CONVERSION <time_conversion>]
  [LENGTH_CONVERSION <length_conversion>]
END OPTIONS
```

```
BEGIN DIMENSIONS
  NLAKES <nlakes>
  NOUTLETS <noutlets>
  NTABLES <ntables>
END DIMENSIONS
```

```
BEGIN PACKAGEDATA
  <lakeno> <strt> <nlakeconn> [<aux(naux)>] [<boundname>]
  <lakeno> <strt> <nlakeconn> [<aux(naux)>] [<boundname>]
  ...
END PACKAGEDATA
```

*IF nlakeconn IS GREATER THAN ZERO FOR ANY LAKE*

```
BEGIN CONNECTIONDATA
  <lakeno> <iconn> <cellid(ncelldim)> <claktype> <bedleak> <belev> <telev> <connlen> <connwidth>
  <lakeno> <iconn> <cellid(ncelldim)> <claktype> <bedleak> <belev> <telev> <connlen> <connwidth>
  ...
END CONNECTIONDATA
```

*IF ntables IS GREATER THAN ZERO*

```
BEGIN TABLES
  <lakeno> TAB6 FILEIN <tab6_filename>
  <lakeno> TAB6 FILEIN <tab6_filename>
  ...
END TABLES
```

*IF noutlets IS GREATER THAN ZERO FOR ANY LAKE*

```
BEGIN OUTLETS
  <outletno> <lakein> <lakeout> <couttype> <invert> <width> <rough> <slope>
  <outletno> <lakein> <lakeout> <couttype> <invert> <width> <rough> <slope>
  ...
END OUTLETS
```

*FOR ANY STRESS PERIOD*

```
BEGIN PERIOD <iper>
  <number> <laksetting>
  <number> <laksetting>
  ...
END PERIOD
```

# 湖泊(LAK)软件

```
BEGIN OPTIONS
  PRINT_INPUT
  PRINT_STAGE
  PRINT_FLOWS
  STAGE FILEOUT lak-1.stage.bin
  BUDGET FILEOUT lak-1.cbc
END OPTIONS

BEGIN DIMENSIONS
  NLAKES 1
  NOUTLETS 1
END DIMENSIONS

BEGIN PACKAGEDATA
# lakeno    strt lakeconn boundname
      1  110.00      57  LAKE_1
END PACKAGEDATA

BEGIN CONNECTIONDATA
# lakeno iconn layer row column    ctype bedleak belev telev dx width
      1     1     1   7      6  HORIZONTAL     0.1     0     0 500   500
      1     2     1   8      6  HORIZONTAL     0.1     0     0 500   500
      1     3     1   9      6  HORIZONTAL     0.1     0     0 500   500
      1     4     1  10      6  HORIZONTAL     0.1     0     0 500   500
      1     5     1  11      6  HORIZONTAL     0.1     0     0 500   500
      1     6     1   6      7  HORIZONTAL     0.1     0     0 500   500
      1     7     2   7      7    VERTICAL     0.1     0     0   0     0
      1     8     2   8      7    VERTICAL     0.1     0     0   0     0
      1     9     2   8      7  HORIZONTAL     0.1     0     0 250   500
      1    10     2   9      7    VERTICAL     0.1     0     0   0     0
      1    11     2   9      7  HORIZONTAL     0.1     0     0 250   500
      1    12     2  10      7    VERTICAL     0.1     0     0   0     0
      1    13     2  10      7  HORIZONTAL     0.1     0     0 250   500
      1    14     2  11      7    VERTICAL     0.1     0     0   0     0
      1    15     1  12      7  HORIZONTAL     0.1     0     0 500   500
      1    16     1   6      8  HORIZONTAL     0.1     0     0 500   500
      1    17     2   7      8    VERTICAL     0.1     0     0   0     0
      1    18     2   7      8  HORIZONTAL     0.1     0     0 250   500
      1    19     3   8      8    VERTICAL     0.1     0     0   0     0
      1    20     3   9      8    VERTICAL     0.1     0     0   0     0
      1    21     3  10      8    VERTICAL     0.1     0     0   0     0
      1    22     2  11      8    VERTICAL     0.1     0     0   0     0
      1    23     2  11      8  HORIZONTAL     0.1     0     0 250   500
      1    24     1  12      8  HORIZONTAL     0.1     0     0 500   500
      1    25     1   6      9  HORIZONTAL     0.1     0     0 500   500
      1    26     2   7      9    VERTICAL     0.1     0     0   0     0
      1    27     2   7      9  HORIZONTAL     0.1     0     0 250   500
      1    28     3   8      9    VERTICAL     0.1     0     0   0     0
      1    29     3   9      9    VERTICAL     0.1     0     0   0     0
      1    30     3  10      9    VERTICAL     0.1     0     0   0     0
      1    31     2  11      9    VERTICAL     0.1     0     0   0     0
      1    32     2  11      9  HORIZONTAL     0.1     0     0 250   500
      1    33     1  12      9  HORIZONTAL     0.1     0     0 500   500
      1    34     1   6     10  HORIZONTAL     0.1     0     0 500   500
      1    35     2   7     10    VERTICAL     0.1     0     0   0     0
      1    36     2   7     10  HORIZONTAL     0.1     0     0 250   500
      1    37     3   8     10    VERTICAL     0.1     0     0   0     0
      1    38     3   9     10    VERTICAL     0.1     0     0   0     0
      1    39     3  10     10    VERTICAL     0.1     0     0   0     0
      1    40     2  11     10    VERTICAL     0.1     0     0   0     0
      1    41     2  11     10  HORIZONTAL     0.1     0     0 250   500
      1    42     1  12     10  HORIZONTAL     0.1     0     0 500   500
      1    43     1   6     11  HORIZONTAL     0.1     0     0 500   500
      1    44     2   7     11    VERTICAL     0.1     0     0   0     0
      1    45     2   8     11    VERTICAL     0.1     0     0   0     0
      1    46     2   8     11  HORIZONTAL     0.1     0     0 250   500
      1    47     2   9     11    VERTICAL     0.1     0     0   0     0
      1    48     2   9     11  HORIZONTAL     0.1     0     0 250   500
      1    49     2  10     11    VERTICAL     0.1     0     0   0     0
      1    50     2  10     11  HORIZONTAL     0.1     0     0 250   500
      1    51     2  11     11    VERTICAL     0.1     0     0   0     0
      1    52     1  12     11  HORIZONTAL     0.1     0     0 500   500
      1    53     1   7     12  HORIZONTAL     0.1     0     0 500   500
      1    54     1   8     12  HORIZONTAL     0.1     0     0 500   500
      1    55     1   9     12  HORIZONTAL     0.1     0     0 500   500
      1    56     1  10     12  HORIZONTAL     0.1     0     0 500   500
      1    57     1  11     12  HORIZONTAL     0.1     0     0 500   500
END CONNECTIONDATA

BEGIN OUTLETS
# outletno lakein lakeout  outtype invert  width  rough  slope
        1      1       0 SPECIFIED      0      0      0      0
END OUTLETS

BEGIN PERIOD 1
  1 RAINFALL 0.0116
  1 EVAPORATION 0.0103
END PERIOD

BEGIN PERIOD 100
  1 STATUS CONSTANT
  1 STAGE 110.
END PERIOD
```

# Ghost-Node Correction (GNC)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [EXPLICIT]
END OPTIONS

BEGIN DIMENSIONS
  NUMGNC <numgnc>
  NUMALPHAJ <numalphaj>
END DIMENSIONS

BEGIN GNCDATA
  <cellidn> <cellidm> <cellidsj(n
  <cellidn> <cellidm> <cellidsj(n
  ...
END GNCDATA
```

**Block: OPTIONS**

PRINT_INPUT—keyword to indicate that the list of GNC information will be written to the listing file immediately after it is read.

PRINT_FLOWS—keyword to indicate that the list of GNC flow rates will be printed to the listing file for every stress period time step in which "BUDGET PRINT" is specified in Output Control. If there is no Output Control option and "PRINT_FLOWS" is specified, then flow rates are printed for the last time step of each stress period.

EXPLICIT—keyword to indicate that the ghost node correction is applied in an explicit manner on the right-hand side of the matrix. The explicit approach will likely require additional outer iterations. If the keyword is not specified, then the correction will be applied in an implicit manner on the left-hand side. The implicit approach will likely converge better, but may require additional memory. If the EXPLICIT keyword is not specified, then the BICGSTAB linear acceleration option should be specified within the LINEAR block of the Sparse Matrix Solver.

**Block: DIMENSIONS**

numgnc—is the number of GNC entries.

numalphaj—is the number of contributing factors.

**Block: GNCDATA**

# Ghost-Node Correction (GNC)软件

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [EXPLICIT]
END OPTIONS

BEGIN DIMENSIONS
  NUMGNC <numgnc>
  NUMALPHAJ <numalphaj>
END DIMENSIONS

BEGIN GNCDATA
  <cellidn> <cellidm> <cellidsj(numalphaj)> <alphasj(numalphaj)>
  <cellidn> <cellidm> <cellidsj(numalphaj)> <alphasj(numalphaj)>
  ...
END GNCDATA
```

**Block: GNCDATA**

cellidn—is the cellid of the cell, $n$, in which the ghost node is located. For a structured grid that uses the DIS input file, CELLIDN is the layer, row, and column numbers of the cell. For a grid that uses the DISV input file, CELLIDN is the layer number and CELL2D number for the two cells. If the model uses the unstructured discretization (DISU) input file, then CELLIDN is the node number for the cell.

cellidm—is the cellid of the connecting cell, $m$, to which flow occurs from the ghost node. For a structured grid that uses the DIS input file, CELLIDM is the layer, row, and column numbers of the cell. For a grid that uses the DISV input file, CELLIDM is the layer number and CELL2D number for the two cells. If the model uses the unstructured discretization (DISU) input file, then CELLIDM is the node number for the cell.

cellidsj—is the array of CELLIDS for the contributing j cells, which contribute to the interpolated head value at the ghost node. This item contains one CELLID for each of the contributing cells of the ghost node. Note that if the number of actual contributing cells needed by the user is less than NUMALPHAJ for any ghost node, then a dummy CELLID of zero(s) should be inserted with an associated contributing factor of zero. For a structured grid that uses the DIS input file, CELLID is the layer, row, and column numbers of the cell. For a grid that uses the DISV input file, CELLID is the layer number and cell2d number for the two cells. If the model uses the unstructured discretization (DISU) input file, then CELLID is the node number for the cell.

alphasj—is the contributing factors for each contributing node in CELLIDSJ. Note that if the number of actual contributing cells is less than NUMALPHAJ for any ghost node, then dummy CELLIDS should be inserted with an associated contributing factor of zero. The sum of ALPHASJ should be less than one. This is because one minus the sum of ALPHASJ is equal to the alpha term (alpha n in equation 4-61 of the GWF Model report) that is multiplied by the head in cell n.

# Ghost-Node Correction (GNC)软件

## Example Input File

```
BEGIN OPTIONS
  PRINT_INPUT
  PRINT_FLOWS
END OPTIONS

BEGIN DIMENSIONS
  NUMGNC 24
  NUMALPHAJ 1
END DIMENSIONS

BEGIN GNCDATA
  10 41 9 0.333333333333
  10 43 11 0.333333333333
  11 44 10 0.333333333333
  11 46 12 0.333333333333
  12 47 11 0.333333333333
  12 49 13 0.333333333333
  16 41 9 0.333333333333
  16 59 20 0.333333333333
  17 49 13 0.333333333333
  17 67 21 0.333333333333
  20 68 16 0.333333333333
  20 86 24 0.333333333333
  21 76 17 0.333333333333
  21 94 25 0.333333333333
  24 95 20 0.333333333333
  24 113 28 0.333333333333
  25 103 21 0.333333333333
  25 121 32 0.333333333333
  29 113 28 0.333333333333
  29 115 30 0.333333333333
  30 116 29 0.333333333333
  30 118 31 0.333333333333
  31 119 30 0.333333333333
  31 121 32 0.333333333333
END GNCDATA
```

# Ghost-Node Correction (GNC)软件

从名称文件中读取类型GNC6文件，GNC软件的输入。每个GWF模型仅能定义一个GNC软件。

GNC软件有2个可选项对方程系统增加修改项。

●隐式选项（默认），对方程的左手边和右手边增加修改项，使用默认选项时，稀疏矩阵求解器的LINEAR块中需要定义BICGSTAB加速项。

●当对GNC软件定义EXPLICIT选项时，对右手边增加修改项，可使用CG或BICGSTAB加速方法。

# 地下水流(GWF)模型的交换

在模拟名称文件中，从GWF6-GWF6类型文件读取GWF-GWF Exchange的输入。

使用XT3D功能改进某些类型单元连接的水流计算精度时，表征地下水流的各向异性，不能实施GWF-GWF交换。

# 地下水流(GWF)模型的交换

## Structure of Blocks

```
BEGIN OPTIONS
  [AUXILIARY <auxiliary(naux)>]
  [BOUNDNAMES]
  [PRINT_INPUT]
  [PRINT_FLOWS]
  [SAVE_FLOWS]
  [CELL_AVERAGING <cell_averaging>]
  [VARIABLECV [DEWATERED]]
  [NEWTON]
  [XT3D]
  [GNC6 FILEIN <gnc6_filename>]
  [MVR6 FILEIN <mvr6_filename>]
  [OBS6 FILEIN <obs6_filename>]
END OPTIONS
```

**Block: OPTIONS**

auxiliary—an array of auxiliary variable names. There is no limit on the number of auxiliary variables that can be provided. Most auxiliary variables will not be used by the GWF-GWF Exchange, but they will be available for use by other parts of the program. If an auxiliary variable with the name "ANGLDEGX" is found, then this information will be used as the angle (provided in degrees) between the connection face normal and the x axis, where a value of zero indicates that a normal vector points directly along the positive x axis. The connection face normal is a normal vector on the cell face shared between the cell in model 1 and the cell in model 2 pointing away from the model 1 cell. Additional information on "ANGLDEGX" is provided in the description of the DISU Package. If an auxiliary variable with the name "CDIST" is found, then this information will be used as the straight-line connection distance, including the vertical component, between the two cell centers. Both ANGLDEGX and CDIST are required if specific discharge is calculated for either of the groundwater models.

BOUNDNAMES—keyword to indicate that boundary names may be provided with the list of GWF Exchange cells.

```
BEGIN DIMENSIONS
  NEXG <nexg>
END DIMENSIONS
```

```
BEGIN EXCHANGEDATA
  <cellidm1> <cellidm2> <ihc> <cl1> <cl2> <hwva> [<aux(naux)>] [<boundname>]
  <cellidm1> <cellidm2> <ihc> <cl1> <cl2> <hwva> [<aux(naux)>] [<boundname>]
  ...
END EXCHANGEDATA
```

# 地下水流(GWF)模型的交换

## Example Input File

```
BEGIN OPTIONS
  PRINT_INPUT
  PRINT_FLOWS
  SAVE_FLOWS
  AUXILIARY testaux
  GNC6 FILEIN simulation.gnc
  MVR6 FILEIN simulation.mvr
END OPTIONS

BEGIN DIMENSIONS
  NEXG 36
END DIMENSIONS
```

```
  # nodem1 nodem2 ihc cl1   cl2    fahl  testaux
BEGIN EXCHANGEDATA
#
#       left side
        16      1    1  50. 16.67 33.33  100.99
        16     10    1  50. 16.67 33.33  100.99
        16     19    1  50. 16.67 33.33  100.99
        23     28    1  50. 16.67 33.33  100.99
        23     37    1  50. 16.67 33.33  100.99
        23     46    1  50. 16.67 33.33  100.99
        30     55    1  50. 16.67 33.33  100.99
        30     64    1  50. 16.67 33.33  100.99
        30     73    1  50. 16.67 33.33  100.99
#
#       right side
        20      9    1  50. 16.67 33.33  100.99
        20     18    1  50. 16.67 33.33  100.99
        20     27    1  50. 16.67 33.33  100.99
        27     36    1  50. 16.67 33.33  100.99
        27     45    1  50. 16.67 33.33  100.99
        27     54    1  50. 16.67 33.33  100.99
        34     63    1  50. 16.67 33.33  100.99
        34     72    1  50. 16.67 33.33  100.99
        34     81    1  50. 16.67 33.33  100.99
#
#       back
        10      1    1  50. 17.67 33.33  100.99
        10      2    1  50. 17.67 33.33  100.99
        10      3    1  50. 17.67 33.33  100.99
        11      4    1  50. 17.67 33.33  100.99
        11      5    1  50. 17.67 33.33  100.99
        11      6    1  50. 17.67 33.33  100.99
        12      7    1  50. 17.67 33.33  100.99
        12      8    1  50. 17.67 33.33  100.99
        12      9    1  50. 17.67 33.33  100.99
#
#       front
        38     73    1  50. 17.67 33.33  100.99
        38     74    1  50. 17.67 33.33  100.99
        38     75    1  50. 17.67 33.33  100.99
        39     76    1  50. 17.67 33.33  100.99
        39     77    1  50. 17.67 33.33  100.99
        39     78    1  50. 17.67 33.33  100.99
        40     79    1  50. 17.67 33.33  100.99
        40     80    1  50. 17.67 33.33  100.99
        40     81    1  50. 17.67 33.33  100.99
END EXCHANGEDATA
```

# 地下水物质输移(GWT)模型

地下水流中的溶质输移模拟，。。。。。。

# 迭代模型求解

迭代模型求解（IMS）在模拟名称文件中的SOLUTIONGROUP块中定义。模型求解将求解所有添加到其中的模型，与模拟名称文件中定义的一致，将包括Numerical Exchanges模块。

IMS需要定义非线性和线性的设置。

# 迭代模型求解

## Structure of Blocks

```
BEGIN OPTIONS
  [PRINT_OPTION <print_option>]
  [COMPLEXITY <complexity>]
  [CSV_OUTER_OUTPUT FILEOUT <outer_csvfile>]
  [CSV_INNER_OUTPUT FILEOUT <inner_csvfile>]
  [NO_PTC [<no_ptc_option>]]
  [ATS_OUTER_MAXIMUM_FRACTION <ats_outer_maximum_fraction>]
END OPTIONS
```

**Block: OPTIONS**

print_option—is a flag that controls printing of convergence information from the solver. NONE means print nothing. SUMMARY means print only the total number of iterations and nonlinear residual reduction summaries. ALL means print linear matrix solver convergence information to the solution listing file and model specific linear matrix solver convergence information to each model listing file in addition to SUMMARY information. NONE is default if PRINT_OPTION is not specified.

```
BEGIN NONLINEAR
  OUTER_DVCLOSE <outer_dvclose>
  OUTER_MAXIMUM <outer_maximum>
  [UNDER_RELAXATION <under_relaxation>]
  [UNDER_RELAXATION_GAMMA <under_relaxation_gamma>]
  [UNDER_RELAXATION_THETA <under_relaxation_theta>]
  [UNDER_RELAXATION_KAPPA <under_relaxation_kappa>]
  [UNDER_RELAXATION_MOMENTUM <under_relaxation_momentum>]
  [BACKTRACKING_NUMBER <backtracking_number>]
  [BACKTRACKING_TOLERANCE <backtracking_tolerance>]
  [BACKTRACKING_REDUCTION_FACTOR <backtracking_reduction_
  [BACKTRACKING_RESIDUAL_LIMIT <backtracking_residual_li
END NONLINEAR
```

**Block: NONLINEAR**

outer_dvclose—real value defining the dependent-variable (for example, head) change criterion for convergence of the outer (nonlinear) iterations, in units of the dependent-variable (for example, length for head). When the maximum absolute value of the dependent-variable change at all nodes during an iteration is less than or equal to OUTER_DVCLOSE, iteration stops. Commonly, OUTER_DVCLOSE equals 0.01. The keyword, OUTER_HCLOSE can be still be specified instead of OUTER_DVCLOSE for backward compatibility with previous versions of MODFLOW 6 but eventually OUTER_HCLOSE will be deprecated and specification of OUTER_HCLOSE will cause MODFLOW 6 to terminate with an error.

```
BEGIN LINEAR
  INNER_MAXIMUM <inner_maximum>
  INNER_DVCLOSE <inner_dvclose>
  INNER_RCLOSE <inner_rclose> [<rclose_option>]
  LINEAR_ACCELERATION <linear_acceleration>
  [RELAXATION_FACTOR <relaxation_factor>]
  [PRECONDITIONER_LEVELS <preconditioner_levels>]
  [PRECONDITIONER_DROP_TOLERANCE <preconditioner_drop_tolerance>]
  [NUMBER_ORTHOGONALIZATIONS <number_orthogonalizations>]
  [SCALING_METHOD <scaling_method>]
  [REORDERING_METHOD <reordering_method>]
END LINEAR
```

**Block: LINEAR**

inner_maximum—integer value defining the maximum number of inner (linear) iterations. The number typically depends on the characteristics of the matrix solution scheme being used. For nonlinear problems, INNER_MAXIMUM usually ranges from 60 to 600; a value of 100 will be sufficient for most linear problems.

# 迭代模型求解

## Example Input File

```
BEGIN OPTIONS
  PRINT_OPTION ALL
  COMPLEXITY MODERATE
END OPTIONS

BEGIN NONLINEAR
  OUTER_DVCLOSE 1.E-4
  OUTER_MAXIMUM 2000
  UNDER_RELAXATION DBD
  UNDER_RELAXATION_THETA 0.70
  UNDER_RELAXATION_KAPPA 0.100000E-03
  UNDER_RELAXATION_GAMMA 0.
  UNDER_RELAXATION_MOMENTUM 0.
  BACKTRACKING_NUMBER 20
  BACKTRACKING_TOLERANCE 2.
  BACKTRACKING_REDUCTION_FACTOR 0.6
  BACKTRACKING_RESIDUAL_LIMIT 5.000000E-04
END NONLINEAR
```

```
BEGIN LINEAR
   INNER_MAXIMUM 100
   INNER_DVCLOSE 1.0E-4
   INNER_RCLOSE 0.001
   LINEAR_ACCELERATION BICGSTAB
   RELAXATION_FACTOR 0.97
   SCALING_METHOD NONE
   REORDERING_METHOD NONE
END LINEAR
```

# 观测(OBS)工具

MODFLOW6支持Observation工具，不需要输入观察的值。
观察工具提供提取在模型运行期间产生的考察区域的数值解。
观测工具不计算残差值，即观测与模型计算值之差。
观测工具生成的输出方便后处理。
观察工具的输入是从一个或多个输入文件读取，各文件与某个模型或软件包相关。对于提取GWF模型的模拟值，从软件包的OPTIONS块的OBS6关键词的文件读取。
各OBS6文件可包含一个OPTIONS块，和一个或多个CONTINOUS块。
观测值在各时间步末尾输出，表示在时间步期间MODFLOW6使用的值。当OBS工具的输入涉及没有某驱动周期内定义的驱动软件边界时，给定NODATA值。
OBS工具生成的输出文件可以是文本或二进制格式。二进制文件后缀是.bsv

# 观测(OBS)工具

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DIGITS <digits>]
  [PRINT_INPUT]
END OPTIONS
```

```
BEGIN CONTINUOUS FILEOUT <obs_output_file_name> [BINARY]
  <obsname> <obstype> <id> [<id2>]
  <obsname> <obstype> <id> [<id2>]
  ...
END CONTINUOUS
```

**Explanation of Variables**

**Block: OPTIONS**

DIGITS—Keyword and an integer digits specifier used for conversion of simulated values to text on output. If not specified, the default is the maximum number of digits stored in the program (as written with the G0 Fortran specifier). When simulated values are written to a comma-separated value text file specified in a CONTINUOUS block below, the digits specifier controls the number of significant digits with which simulated values are written to the output file. The digits specifier has no effect on the number of significant digits with which the simulation time is written for continuous observations. If DIGITS is specified as zero, then observations are written with the default setting, which is the maximum number of digits.

PRINT_INPUT—keyword to indicate that the list of observation information will be written to the listing file immediately after it is read.

# 观测(OBS)工具

## Structure of Blocks

*FOR EACH SIMULATION*

```
BEGIN OPTIONS
  [DIGITS <digits>]
  [PRINT_INPUT]
END OPTIONS
```

```
BEGIN CONTINUOUS FILEOUT <obs_output_file_name> [BINARY]
  <obsname> <obstype> <id> [<id2>]
  <obsname> <obstype> <id> [<id2>]
  ...
END CONTINUOUS
```

**Block: CONTINUOUS**

FILEOUT—keyword to specify that an output filename is expected next.

obs_output_file_name—Name of a file to which simulated values corresponding to observations in the block are to be written. The file name can be an absolute or relative path name. A unique output file must be specified for each CONTINUOUS block. If the "BINARY" option is used, output is written in binary form. By convention, text output files have the extension "csv" (for "Comma-Separated Values") and binary output files have the extension "bsv" (for "Binary Simulated Values").

BINARY—an optional keyword used to indicate that the output file should be written in binary (unformatted) form.

obsname—string of 1 to 40 nonblank characters used to identify the observation. The identifier need not be unique; however, identification and post-processing of observations in the output files are facilitated if each observation is given a unique name.

obstype—a string of characters used to identify the observation type.

id—Text identifying cell where observation is located. For packages other than NPF, if boundary names are defined in the corresponding package input file, ID can be a boundary name. Otherwise ID is a cellid. If the model discretization is type DIS, cellid is three integers (layer, row, column). If the discretization is DISV, cellid is two integers (layer, cell number). If the discretization is DISU, cellid is one integer (node number).

id2—Text identifying cell adjacent to cell identified by ID. The form of ID2 is as described for ID. ID2 is used for intercell-flow observations of a GWF model, for three observation types of the LAK Package, for two observation types of the MAW Package, and one observation type of the UZF Package.

# 可提供的模型观测类型

GWF模型

GWT模型

# 随时间变化的输入(Time-variable Input)

之前版本的MODFLOW，大多数驱动力边界软件以驱动周期为单位读取输入，这些读取的值在驱动周期内是恒定的。在MODFLOW6中，很多驱动力值可以定义为更高维度的时间分辨率（逐时间步长或更短的时间步长），使用两种时间变化方法之一来定义。边界数据读取以列表单元形式，称为“时间序列”的时间变化输入。以2D数组形式的边界数据称为“时间序列数组”。

插值方法有3种：STEPWISE, LINEAR, LINEAREND

# 随时间变化的输入(Time-variable Input)

## 时间序列

时间序列文件，在软件包的OPTIONS块中的关键词TS6 FILEIN。
通常时间序列文件使用ts后缀。

**使用时间序列定义软件包输入的例子：**

Contents of file "well_pump_rates.ts":

```
BEGIN ATTRIBUTES
  NAMES well-A-series well-B-series well-C-series
  METHODS stepwise linear stepwise
END ATTRIBUTES

BEGIN TIMESERIES
  # time  well-A-series    well-B-series    well-C-series
  0.0          0.0              0.0              0.0
  1.0       -500.0              0.0           -400.0
  2.0       -500.0          -1000.0           -500.0
  5.0       -500.0          -1200.0           -200.0
  8.0       -500.0          -1100.0              0.0
END TIMESERIES
```

# 随时间变化的输入(Time-variable Input)

使用时间序列定义软件包输入的例子:

Contents of the Well Package input file:

```
BEGIN OPTIONS
  TS6 FILEIN well_pump_rates.ts
END OPTIONS

BEGIN DIMENSIONS
  MAXBOUND 4
END DIMENSIONS

BEGIN PERIOD 2
  #layer row col  Q (or time series)
      9  192  44  well-A-series
     10   43  17  well-B-series
     11   12  17  well-C-series
END PERIOD
```

```
BEGIN PERIOD 4
  #layer row col  Q (or time series)
      9  192  44  well-A-series
     10   43  17  well-B-series
     11   12  17  well-C-series
      2   27  36   -900.0
END PERIOD

BEGIN PERIOD 8
      2   27  36   -900.0
END PERIOD
```

# 随时间变化的输入(Time-variable Input)

## 时间-数组序列

Contents of Recharge package input file:

```
BEGIN OPTIONS
  READASARRAYS
  AUX RchMult
  TAS6 FILEIN rch_time_array_series.tas
  AUXMULTNAME RchMult
  PRINT_INPUT
END OPTIONS

BEGIN PERIOD 1
  IRCH
    CONSTANT 1
  RECHARGE TIMEARRAYSERIES RchArraySeries_1
  RchMult
    INTERNAL FACTOR 1.0
  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
  0.0  1.0  1.0  0.5  0.5  0.0  0.0  0.0  0.0  0.0
  0.0  1.0  1.0  1.0  1.0  0.5  0.0  0.0  0.0  0.0
  0.0  1.0  1.0  1.0  1.0  1.0  0.5  0.0  0.0  0.0
  0.0  0.2  1.0  1.0  1.0  1.0  1.0  0.5  0.2  0.0
  0.0  0.0  0.5  1.0  1.0  1.0  1.0  0.5  0.0  0.0
  0.0  0.0  0.0  0.2  0.2  0.2  0.2  0.0  0.0  0.0
  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0  0.0
END PERIOD
```

Contents of file "rch_time_array_series.tas":

```
BEGIN ATTRIBUTES
  NAME RchArraySeries_1
  METHOD LINEAR
END ATTRIBUTES

BEGIN TIME 0.0
  CONSTANT 0.0033
END TIME

BEGIN TIME 91.0
  CONSTANT 0.0035
END TIME

BEGIN TIME 183.0
  CONSTANT 0.0037
END TIME

BEGIN TIME 274.0
  CONSTANT 0.0039
END TIME

BEGIN TIME 365.0
  CONSTANT 0.0035
END TIME
```