Enhancements to the TOUGH2 Simulator Implemented in iTOUGH2

Stefan Finsterle

Earth Sciences Division
Lawrence Berkeley National Laboratory
University of California
Berkeley, CA 94720

March 2015

This work was supported by the U.S. Dept. of Energy under Contract No. DE-AC02-05CH11231.
DISCLAIMER

This document was prepared as an account of work sponsored by the United States Government. While this document is believed to contain correct information, neither the United States Government nor any agency thereof, nor The Regents of the University of California, nor any of their employees, makes any warranty, express or implied, or assumes any legal responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by its trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof, or The Regents of the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, or The Regents of the University of California.

Ernest Orlando Lawrence Berkeley National Laboratory
is an equal opportunity employer.
# TABLE OF CONTENTS

1 Introduction .................................................................................................................. 1

2 More Options (MOMOP) ............................................................................................ 3

3 Printout Options .......................................................................................................... 6
   3.1 Printout-Control through KDATA ...................................................................... 6
   3.2 Excluding Domains from Global Material Balance Printout .......................... 6

4 Secondary Mesh (ELEM2, CONN2) ........................................................................... 7

5 Permeability Assignments .......................................................................................... 8
   5.1 Element-by-Element Permeabilities in block ELEME .................................. 8
   5.2 Element-by-Element Permeabilities from external files using block MAPPI .... 8
   5.3 Permeability and Porosity Regions ................................................................... 12
   5.4 Geostatistics ....................................................................................................... 14

6 Boundary Conditions and Sink/Source Terms ......................................................... 15
   6.1 Time-Dependent Dirichlet Boundary Conditions ........................................ 15
   6.2 Free-Drainage Boundary Condition ................................................................ 15
   6.3 User-Specified Boundary Conditions ............................................................. 15
   6.4 Tabular Input of Time-Dependent Rates ....................................................... 17
   6.5 Honoring Generation Times ............................................................................ 18
   6.6 Material-Related Sinks/Sources ...................................................................... 18
   6.7 Sink/Source Regions ......................................................................................... 19

7 Relative Permeability and Capillary Pressure Functions ........................................ 23
   7.1 Modified Brooks-Corey Model ................................................................. 23
   7.2 Modified van Genuchten Model .................................................................... 25

8 Active Fracture Model ............................................................................................... 28
   8.1 Active Fracture Concept ................................................................................ 28
   8.2 Reduction of Fracture-Matrix Interface Area ................................................ 29

9 Coupled Overland – Subsurface Flow ................................................................. 30

10 Semi-Analytical Radial Heat Exchange ............................................................... 34
LIST OF FIGURES

Figure 1. Primary and secondary mesh generation ................................................................. 7
Figure 2. TOUGH2 input file for mapping permeability modifiers from external file .... 11
Figure 3. Data file containing log-10 permeability modifiers on indexed regular mesh ... 11
Figure 4. TOUGH2 input file for assigning ellipsoidal permeability region ................. 14
Figure 5. TOUGH2 input file for assigning time-dependent Dirichlet boundary conditions ................................................................................................................................. 15
Figure 6. Subroutine USERBC for specifying time-dependent boundary conditions ...... 17
Figure 7. TOUGH2 input format for specifying time-dependent generation rates in tabular format, and using conversion factors for time and rate ........................................ 18
Figure 8. TOUGH2 input format for specifying material-related, volume-specific generation rates ......................................................................................................................... 19
Figure 9. TOUGH2 input file for testing ellipsoidal sink/source regions ..................... 22
Figure 10. Excerpt of TOUGH2 output file showing allocation of generation rates within ellipsoidal source region ........................................................................................................ 22
Figure 11. Modified Brooks-Corey relative permeability and capillary pressure curves . 24
Figure 12. Modified van Genuchten relative permeability and capillary pressure curves. 26
Figure 13. TOUGH2 input file for testing coupled surface water – subsurface flow .... 33
Figure 14. TOUGH2 input file for testing radial heat exchange option ......................... 36
LIST OF TABLES

Table 1. Documentation of iTOUGH2 ................................................................. 1
Table 2. Documentation of TOUGH2 and Related Modules .................................. 2
Table 3. More Options .......................................................................................... 3
Table 4. Amount of Printout as a Function of Variable KDATA .............................. 6
Table 5. Geometrical Parameters Defining Region ................................................. 14
Table 6. Geometrical Parameters Defining Region ................................................. 21
Table 7. Input Parameters for Modified Brooks-Corey Model ............................... 24
Table 8. Input Parameters for Modified van Genuchten Model .............................. 27
Table 9. Option for Reducing Fracture-Matrix Interface Area ............................... 29
1 Introduction

iTOUGH2 is a program for parameter estimation, sensitivity analysis, and uncertainty propagation analysis. It is based on the TOUGH2 simulator for non-isothermal multiphase, multicomponent flow and transport in fractured and porous media. The manuals describing iTOUGH2’s capabilities are listed in Table 1.

The core of iTOUGH2 contains slightly modified versions of TOUGH2 modules as documented in Pruess et al. [2012] and other reports (see Table 2 below).

Most code modifications are editorial and do not affect the simulation results. Thus, standard TOUGH2 input files can be used in iTOUGH2, and consistent results are obtained if iTOUGH2 is run in forward mode. However, a number of modifications have been made to the version of TOUGH2 that is implemented in iTOUGH2. They enhance the functionality, flexibility, and ease-of-use of the forward simulator. Some of these enhancements are described in this report; more significant developments are documented in separate reports. Table 2 lists manuals describing general aspects of TOUGH2 modules or specific features of the forward simulator. (Note that not all capabilities mentioned in Table 2 are publicly released.)

The key to a successful application of iTOUGH2 is (i) a good understanding of multiphase flow processes, (ii) the ability to conceptualize the given flow and transport problem and to develop a corresponding TOUGH2 model, (iii) detailed knowledge about the data used for calibration, (iv) an understanding of parameter estimation theory and the correct interpretation of inverse modeling results, (v) proficiency in using iTOUGH2 options. This report I smainly concerned with issues (i) and (ii).

Table 1. Documentation of iTOUGH2

<table>
<thead>
<tr>
<th>Module / Feature / Capability</th>
<th>Reference</th>
<th>Source Files</th>
</tr>
</thead>
<tbody>
<tr>
<td>iTOUGH2 user’s guide; theoretical background</td>
<td>Finsterle, 2015a</td>
<td>-</td>
</tr>
<tr>
<td>iTOUGH2 command reference</td>
<td>Finsterle, 2015b</td>
<td>-</td>
</tr>
<tr>
<td>iTOUGH2 sample problems</td>
<td>Finsterle, 2015c</td>
<td>-</td>
</tr>
<tr>
<td>Parallel execution of TOUGH2 runs</td>
<td>Finsterle, 1998</td>
<td>it2pvm.f</td>
</tr>
<tr>
<td>Geostatistical simulations</td>
<td>Finsterle and Kowalsky, 2007</td>
<td>it2gslib.f</td>
</tr>
<tr>
<td>Link to external models using PEST protocol</td>
<td>Finsterle, 2010</td>
<td>it2pest.f mio.f90</td>
</tr>
<tr>
<td>Global sensitivity and data-worth analysis</td>
<td>Wainwright and Finsterle, 2015</td>
<td>it2sa.f</td>
</tr>
</tbody>
</table>
# Table 2. Documentation of TOUGH2 and Related Modules

<table>
<thead>
<tr>
<th>Module / Feature / Capability</th>
<th>Reference</th>
<th>Source Files</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main TOUGH2 reference; general description of code capabilities, mathematical model, numerical scheme, input formats, and sample problems</td>
<td><em>Pruess et al.</em>, 2012</td>
<td>eos1.f eos2.f eos3.f eos4.f eos5.f eos7.f eos7r.f eos8.f eos9.f ewasg.f</td>
</tr>
<tr>
<td>Three-phase (gas, aqueous, NAPL), three-component (water, air, VOC) module T2VOC</td>
<td><em>Falta et al.</em>, 1995</td>
<td>eost2voc.f</td>
</tr>
<tr>
<td>Three-phase (gas, aqueous, NAPL), multi-component (water, NCGs, VOCs) module TMVOC</td>
<td><em>Pruess and Battistelli</em>, 2002</td>
<td>eostmvoc.f</td>
</tr>
<tr>
<td>Three-phase (CO₂-rich phase, aqueous, solid), three-component (water, CO₂, NaCl) module ECO2N</td>
<td><em>Pruess</em>, 2005</td>
<td>eco2n.f</td>
</tr>
<tr>
<td>Four-phase (gaseous and liquid CO₂, aqueous, solid), three-component (water, CO₂, NaCl) module ECO2M</td>
<td><em>Pruess</em>, 2011</td>
<td>eco2m.f</td>
</tr>
<tr>
<td>Two-phase (gas, aqueous), five-component (water, brine, CO₂ or N₂, tracer, CH₄) module EOS7C</td>
<td><em>Oldenburg et al.</em>, 2004</td>
<td>eos7c.f</td>
</tr>
<tr>
<td>Two-phase (steam, liquid), two-component (water₁, water₂) supercritical module EOS1sc</td>
<td><em>Magnúsdóttir and Finsterle</em>, 2015</td>
<td>eos1sc.f</td>
</tr>
<tr>
<td>Two-phase (gas, liquid), two-component (non-Newtonian fluid, air) module EOS3nn</td>
<td><em>Wu et al.</em>, 2002</td>
<td>eos3nn.f</td>
</tr>
<tr>
<td>Two-phase (gas, liquid), three-component (water, air, miscible, solidifying gel) module EOS11</td>
<td><em>Finsterle et al.</em>, 1994</td>
<td>eos11.f</td>
</tr>
<tr>
<td>Non-Darcy flow based on Forchheimer equation</td>
<td><em>Finsterle and Witherspoon</em>, 2001</td>
<td>-</td>
</tr>
<tr>
<td>Iterative linear equation solvers</td>
<td><em>Moridis and Pruess</em>, 1997</td>
<td>t2cg22.f t2solv.f</td>
</tr>
<tr>
<td>Hysteretic relative permeability and capillary pressure functions</td>
<td><em>Doughty</em>, 2013</td>
<td>it2hyster.f</td>
</tr>
<tr>
<td>Wellbore simulator</td>
<td><em>Guðmundsdóttir et al.</em>, 2015</td>
<td>flowell.f</td>
</tr>
<tr>
<td>Geostatistical simulations</td>
<td><em>Finsterle and Kowalsky</em>, 2007</td>
<td>it2gslib.f</td>
</tr>
</tbody>
</table>
2 More Options (MOMOP)

Table 3 describes additional options invoked by integer flags on a line following keyword MOMOP.

Table 3. More Options

<table>
<thead>
<tr>
<th>MOP2</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0, 1</td>
<td>Minimum number of Newton-Raphson iterations</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>Perform at least two iterations; primary variables are always updated.</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>Allow convergence in a single Newton-Raphson iteration for negative simulation times, but require at least two for positive times; useful for steady-state followed by transient simulations.</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>Allow convergence in a single Newton-Raphson iteration for positive simulation times, but require at least two for negative times.</td>
</tr>
<tr>
<td>2</td>
<td>5–9</td>
<td>Length of element names (default: 5 characters)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Format of blocks ELEME, CONNE, INCON, and GENER change depending on Element-name length as follows:</td>
</tr>
</tbody>
</table>

```
ELEME
5  (A3,I2,I5,I5,A2,A3,6E10.4)
6  (A3,I3,I5,I4,A2,A3,6E10.4)
7  (A3,I4,I4,I4,A2,A3,6E10.4)
8  (A3,I5,I4,I3,A2,A3,6E10.4)
9  (A3,I6,I3,I3,A2,A3,6E10.4)

CONNE
5  (2(A3,I2),I5,2I5,I5,4E10.4)
6  (2(A3,I3),I5,2I4,I5,4E10.4)
7  (2(A3,I4),I5,2I3,I5,4E10.4)
8  (2(A3,I5),I3,2I3,I5,4E10.4)
9  (2(A3,I6),I3,2I2,I5,4E10.4)

INCON
5  (A3,I2,I5,I5,E15.8,4E12.4)
6  (A3,I3,I5,I4,E15.8,4E12.4)
7  (A3,I4,I4,I4,E15.8,4E12.4)
8  (A3,I5,I4,I3,E15.8,4E12.4)
9  (A3,I6,I3,I3,E15.8,4E12.4)

GENER
5  (A3,I2,A3,I2,I5,2I5,I5,5X,A4,A1,3E10.4)
6  (A3,I3,A3,I2,I6,2I4,I5,5X,A4,A1,3E10.4)
7  (A3,I4,A3,I2,I5,2I4,I5,5X,A4,A1,3E10.4)
8  (A3,I5,A3,I2,I4,2I4,I5,5X,A4,A1,3E10.4)
9  (A3,I6,A3,I2,I5,2I3,I5,5X,A4,A1,3E10.4)
```
<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
</table>
| 3 | Honoring generation times  
0  | Generation times ignored  
>0 | Time steps adjusted to match generation times |
| 4 | Vapor pressure reduction  
0  | No vapor pressure reduction at low liquid saturation  
>0 | Reduces vapor pressure for \( S_i < 0.02 \) to prevent liquid disappearance by evaporation (only certain EOS modules) |
| 5 | Active Fracture Model (see Sections A7 and A8)  
0  | Active Fracture Model applied to liquid phase only  
>0 | Active Fracture Model applied to all phases |
| 6 | Leverett scaling of capillary pressure  
0  | No Leverett scaling  
>0 | Rescale capillary pressure: \( P_c = P_{c,ref} \sqrt{k_{ref}/k} \) if element-specific permeabilities are specified (see Section A4) |
| 7 | Zero nodal distance  
0  | Take absolute permeability from other element  
>0 | Take absolute and relative permeability from other element |
| 8 | Version of sink/source subroutine  
0  | Take TOUGH2 version of subroutine QU  
>0 | Take TOUGH2V2 version of subroutine QU |
| 9 | Time stepping after time-step reduction to honor printout time  
0  | Continue with time step used before forced time-step reduction  
>0 | Continue with time step imposed by forced time-step reduction |
| 10 | Writing SAVE file  
0  | Write SAVE file only at the end of a forward run  
>0 | Write SAVE file after each printout time |
| 11 | Water properties  
0  | International Formulation Committee (1967)  
1 | IAPWS-IF97  
2 | EOS1sc only: IAPWS-IF97 for \( T < 800^\circ C \)  
 | IAPWS-95 for \( T \geq 800^\circ C \) |
| 12 | Enthalpy of liquid water  
0  | Potential energy not included in enthalpy of liquid water  
>0 | Potential energy included in enthalpy of liquid water |
| 13 | Adjustment of Newton-Raphson increment weighting  
0  | No adjustment  
>0 | Reduce \( \text{WNR} \) by \( MOP2 (13) \) percent if Newton-Raphson iterations oscillate and time step is reduced because \( \text{ITER=NOITE} \) |
| 14 | Air-entry pressure in Brooks-Corey capillary pressure curve (ICP=10)  
0  | Spherical model for interpolation between \( P_c = P_e \) at \( S_i = 0.99 \) and \( P_c = 0.0 \) at \( S_i = 1.0 \)  
1 | Step change in \( P_c \) from \( P_e \) to 0.0 at \( S_i = 0.9999 \)  
2 | Linear interpolation between \( P_c = P_e \) at \( S_i = 0.99 \) and \( P_c = 0.0 \) at \( S_i = 1.0 \)  
3 | Spherical interpolation between \( P_c = P_e \) at \( S_i = 0.99 \) and \( P_c = 0.0 \) at \( S_i = 1.0 \) |
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
</table>
| 15 | 0 | Porosity used for calculation of rock energy content
Use porosity of block ROCKS; this assumes that the porosities provided in block INCON were the result of a pore compressibility/expansivity calculation; the “original” porosity from block ROCKS is used to compensate for equivalent rock-grain density changes.

>0 | Use porosity from block INCON; this assumes that these porosities were not the result from a pore compressibility/expansivity calculation; changes in rock-grain density due to pore compressibility/expansivity are not compensated. |
|   |   |   |
| 16 |   | Porosity-permeability relationships for heterogeneous media
0 | No deterministic correlation |
1 | Material-specific empirical correlations (see subroutine PER2POR) |
|   |   |   |
| 17 |   | Invokes wellbore simulator FloWell
0 | Do not run FloWell simulator |
1 | Run FloWell for each calibration and printout time |
2 | Run FloWell for each time step |
|   |   |   |
| 18 |   | Porosity update by ROCMECH
0 | Do not update porosity |
1 | Apply porosity correction |
|   |   |   |
| 19 | 0 | Treatment of residuals
Keep all residuals

>0 | Do not update primary variable if scaled residual is smaller that $10^{-MOP2(19)-10}$ |
|   |   |   |
| 20 |   | Reading anisotropic permeability modifiers in block ELEME
(Also see block MAPPI (Section 5.2) for reading and mapping permeabilities stored on external files)
0 | Read isotropic permeability modifiers from columns 41–50 |
1 | Read anisotropic permeability modifiers from columns 81–110 |
2 | Read anisotropic permeability modifiers for ISOT=1 from columns 41–50 and for ISOT=2 and 3 from columns 91–110 |
3 Printout Options

3.1 Printout-Control through KDATA

In standard TOUGH2, printout of simulation results is controlled by variables KDATA and MCYPR in block PARAM.1, as well as by the additional printout times TIS given in block TIMES. In iTough2, the calibration times are also stored in array TIS which means that at each calibration time the amount of printout specified by variable KDATA is written to the output file. This may make the TOUGH2 output file extremely long, and requires unnecessary CPU time for disk writing, since the TOUGH2 output file is overwritten each time a new TOUGH2 simulation is initiated by iTough2.

If a negative number is specified for variable KDATA, the amount of printout is reduced (see Table 4), saving both disk space and CPU time. Recall that printout at the calibration points is always written to the iTough2 plot file (see command >>> FORMAT). Also, if a TOUGH2 run is terminated due to a convergence failure or using command kit, the full output is automatically generated for the last time step.

Full output is always provided for the times specified in the TOUGH2 block TIMES; the amount of printout is given by the absolute value of KDATA. The times provided in TOUGH2 block TIMES can be in arbitrary order; they are sorted internally.

Table 4. Amount of Printout as a Function of Variable KDATA

<table>
<thead>
<tr>
<th></th>
<th>-3</th>
<th>-2</th>
<th>-1</th>
<th>Printout of...</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>volume and mass balance</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>+</td>
<td>generation rates</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td></td>
<td></td>
<td>+</td>
<td></td>
<td>most important variables</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td></td>
<td>+</td>
<td></td>
<td></td>
<td>fluxes and velocities</td>
<td>-</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td></td>
<td></td>
<td>-</td>
<td></td>
<td>primary variables</td>
<td>-</td>
<td>-</td>
<td>+</td>
</tr>
</tbody>
</table>

3.2 Excluding Domains from Global Material Balance Printout

Domains with an element volume larger than $10^{20}$ m$^3$ or a negative rock grain specific heat SPHT or with SPHT greater than $10^4$ J/kg °C are excluded from global material balance calculations. The absolute value of SPHT is used in the heat balance equation. Material balances for individual rock types are available through iTough2 commands >> TOTAL MASS and >>> VOLUME.
Secondary Mesh (ELEM2, CONN2)

This enhancement was introduced especially for the estimation of a skin radius (see command >> SKIN) or a MINC parameter (see command >> MINC), both requiring that a new TOUGH2 mesh is generated automatically each iteration without user interference. In order to achieve this, a primary mesh must be generated using the MESHMAKER utility. Elements and connections of this primary mesh can then be overwritten by a secondary mesh provided through blocks ELEM2 and CONN2, which have the same format as blocks ELEM and CONNE, respectively. The names of the elements and connections to be modified by the secondary mesh must be identical with the corresponding ones of the primary mesh. If a secondary mesh is specified, both blocks, ELEM2 and CONN2, must be given; either of the two keywords may be followed by an empty line to indicate that no modification is made.

Figure 1 illustrates an application. A radial mesh is generated using MESHMAKER for simulating a pump test. In block ELEM2, the volume of the first grid block is changed to represent the actual interval volume. In block CONN2, the nodal distance from the first element to the interface is reduced to a very small number as usually done for connections to boundary elements. These last two modifications, which are usually made by editing the mesh file, are now automatically performed whenever a new mesh is generated during an iTOUGH2 run for the estimation of the skin radius.

```
MESHMAKER------*-----2------*-----3------*-----4------*-----5------*-----6
RZ2D
RADI2
   2  1
  0.000E+00  0.100E+00
LOGAR
   20  2  0.300E+00  0.100E-01
LOGAR
   80  3  1.000E+01
LAYER
   1
  0.100E+01

ELEM2------*-----2------*-----3------*-----4------*-----5------*-----6
A1  1  1 .1250E+00

CONN2------*-----2------*-----3------*-----4------*-----5------*-----6
A1  A1  2  1  1.000E-10  .5332E-02  .6283E+00
```

**Figure 1.** Primary and secondary mesh generation.
5 Permeability Assignments

5.1 Element-by-Element Permeabilities in block ELEME

Heterogeneity is introduced into TOUGH2 models by assigning a certain rock type, for which material properties are defined in block ROCKS, to each grid block. The maximum number of rock types one can specify is given by variable \textit{MAXROC} (see file \textit{maxsize.inc}). For high-resolution simulations of permeability heterogeneity, it is inconvenient to specify hundreds of rock types, one for each grid block. TOUGH2 has been extended so that grid block permeabilities or permeability modifiers can be directly specified in block ELEME, columns 41-50. If a positive value is given, it is interpreted as absolute permeability; if a negative value is provided, it is interpreted as a permeability modifier, i.e., a factor with which the absolute permeability specified in block ROCKS is multiplied. If columns 41–50 are blank for the first element, the element-by-element permabilities are ignored. Alternatively, the same information can be provided through block INCON, columns 31–40. Four additional parameters specific to a grid block can be provided in block INCON, columns 41–80.

The anisotropy ratio as given by ratios of \textit{PER(1)}, \textit{PER(2)}, and \textit{PER(3)} in block ROCKS is preserved, i.e., the single permeability modifier is applied to all three permeabilities. To read three permeabilities or permeability modifiers, i.e., one for each of the three directions, \textit{MOP2(20)} has to be set according to Table 3. (Permeabilities and permeability modifiers can also be read from an extranal file and mapped onto the TOUGH2 mesh; see block \textit{MAPPI} described in Section 5.2). Note that the permeability or permeability modifiers are stored in array \textit{USERX(i,N)}, \textit{i}=1…3. To write element-by-element parameters to file \textit{SAVE}, set \textit{MOP(13)} to 1.

Using a permeability modifier instead of permeability itself has the advantage that the mean of the permeability field can easily be changed (or estimated) by adjusting parameter \textit{PER(ISOT)} in block ROCKS.

5.2 Element-by-Element Permeabilities from external files using block MAPPI

Instead of specifying element-by-element permeabilities or permeability modifiers in block ELEME as described in Section 5.1, heterogeneous and anisotropic, potentiall transformed permeabilities or permeability modifiers may be generated externally and provided on a file, which is read in and then mapped onto the TOUGH2 mesh. Various file formats as well as mapping procedures are supported; the related information has to be provided in block \textit{MAPPI}, which contains the following records (read in free format):
Record MAPPI.1

FILENAME : Filename containing (potentially transformed) permeabilities in one of the supported formats (see MAPFMT); make sure the file is available in the local directory

Record MAPPI.2

MAFMT : File format

0: GSLIB format as internally generated by iTOUGH2 [Finsterle and Kowalsky, 2007]

1: VALUE(i), i = 1, ..., MAPNVAL
   Values need to be provided in same order as elements in block ELEME

2: X Y Z VALUE(i), i = 1, ..., MAPNVAL
   Coordinates followed by values; the coordinates may or may not be identical to those in block ELEME

3: I J K VALUE(i), i = 1, ..., MAPNVAL
   Grid indices followed by values; assumes values are provided on a regular grid (see record MAPPI.6); the TOUGH2 grid does not need to be regular

4: I J K NFRACT COND(i), i = 1, ..., MAPNVAL
   Format used by discrete fracture network to continuum grid mapper of Parashar and Reeves [2011]; assumes values are provided on a regular grid with uniform grid spacing (see record MAPPI.6); the TOUGH2 grid does not need to be regular

MAPHEADER : Number of header lines to be skipped before data reading

MAPNVAL : Number of values to be read (1 for single permeability or modifier applied to all three permeabilities; 3 for individual permeabilities or modifiers)

MAPPOINTS : Number of data lines (points in space) where permeability information is provided

Record MAPPI.3

IFIELD1 : Identifies variable

0: porosity, PHI(N)

1–3: permeabilities or permeability modifiers, USERX(i,N)

4: capillary-strength parameter (see Section 7) USERX(4,N)

5: user specified, USERX(5,N)
IFIELD2 : Interpret as value or modifier
0: value
1: modifier

IFIELD3 : Variable transformation
0: no transformation
1: value represents decadic logarithm
2: value represents natural logarithm

Record MAPPI.4

NMAPROCK : Number of material types to be given (see IMAPROCK).
Mapping can be restricted to certain zones within the TOUGH2 model, where a zone is defined by all elements that belong to one or more material domains. This option may be useful to exclude the properties of special elements (e.g., those representing boundary elements, wells, engineered structures, etc.) If NMAPROCK is zero, the heterogeneous values will be mapped onto the entire TOUGH2 grid.

Record MAPPI.5

MAPPING : Type of mapping and averaging procedure
1: nearest neighbor; the property value from the point nearest to the TOUGH2 element center point will be taken
2: arithmetic mean; the arithmetic mean of all property values within a user-specified radius from the TOUGH2 element center point will be taken
3: geometric mean; the geometric mean of all property values within a certain radius from the TOUGH2 element center point will be taken
4: harmonic mean; the harmonic mean of all property values within a certain radius from the TOUGH2 element center point will be taken

SEARCHR : Search radius; For MAPPING = 2, 3, or 4, a search radius has to be specified. Specifying a search radius for option 1 may speed up the mapping, as the nearest point is considered identified as soon as the distance is less than the search radius. Three options exist to specify the search radius r, where SR is an input parameter to be provided by the user, V is the volume of the
given TOUGH2 element, and $D = 1, 2, \text{or } 3$ is the dimension of the model:

$$
\begin{align*}
    r &= \begin{cases} 
        -SR \cdot V^{1/D} & \text{for } SR < 0 \\
        V^{1/D} & \text{for } SR = 0 \\
        SR & \text{for } SR > 0 
    \end{cases}
\end{align*}
$$

Record MAPPI.6 (only if MAPFM = 3 or 4)

$X0, Y0, Z0 : \text{ Origin of coordinate system used in the external file (the TOUGH2 mesh may be shifted by specifying SHIFTX, SHIFTY, and SHIFTZ in columns 51–60, 61–70, and 71–80, respectively, on the line containing keyword ELEME).}$

$DX, DY, DZ : \text{ Grid spacings}$

During the mapping process, each element of the TOUGH2 grid is visited. First, it is checked whether the element belongs to a material type that defines the mapping zone. If so, the coordinates of the element are compared to the coordinates of grid points specified in the external file, and if the distance between the two points is less than the search radius, the property values are retained, averaged, and eventually assigned to the element.

Figure 2 shows an example of block MAPPI in a TOUGH2 input file; Figure 3 shows an excerpt of the data file $Log10PermModifier.dat$.

![Figure 2](image)

**Figure 2.** TOUGH2 input file for mapping permeability modifiers from external file.

![Figure 3](image)

**Figure 3.** Data file containing log-10 permeability modifiers on indexed regular mesh.
5.3 Permeability and Porosity Regions

Instead of providing permeabilities and porosities for individual elements or material domains through blocks ELEME or INCON, the option described in this section allows one to specify permeabilities over a certain geometric region. The location and size of this region is parameterized and can thus be subjected to parameter estimation by inverse modeling (e.g., for sensitivity analysis on fault location). For the system response to be a smooth function of the location and size of the permeability region, the region must comprise multiple elements, with a smooth function describing the adjustment of the permeability to its background value as a function of distance from the center of the region. This option can only be used if element coordinates are provided in columns 51–80 in block ELEME.

The permeabilities and porosities will be changed over all elements within a user-specified region. This region is defined by the location of its center and the extent. The shape can be either an ellipsoid, a rectangular box or cube, or a cylinder. The ellipsoid and box are aligned with the coordinate axes; the cylinder may be arbitrarily oriented.

The box region includes all elements $i$ with coordinates $X_i$, $Y_i$, and $Z_i$ that lie within a rectangular box, i.e.,

$$
\begin{align*}
    d_x &= |X_i - X_c| \leq L_X \\
    d_y &= |Y_i - Y_c| \leq L_Y \\
    d_z &= |Z_i - Z_c| \leq L_Z
\end{align*}
$$

The ellipsoidal region includes all elements $i$ with coordinates $X_i$, $Y_i$, and $Z_i$ that satisfy the equation

$$
d^2 = \left(\frac{X_i - X_c}{L_X}\right)^2 + \left(\frac{Y_i - Y_c}{L_Y}\right)^2 + \left(\frac{Z_i - Z_c}{L_Z}\right)^2 < 1
$$

Here, $X_c$, $Y_c$, and $Z_c$ are the center coordinates of the box or ellipsoid, and $L_X$, $L_Y$, and $L_Z$ are the three half-lengths of the box or the three semi-axes of the ellipsoid. The region is aligned with the coordinate axes.

A cylindrical region is defined by the starting and end coordinates of its axis, and a radius. All elements within this arbitrarily oriented cylinder are selected.

The permeabilities and porosities of the elements within the region will be calculated as a simple weighted average of the respective background values $k_0$ and $\phi_0$ (i.e., the values in the ROCKS block assigned to the element in block ELEME) and those specified for the particular region, i.e., $k_{\text{reg}}$ and $\phi_{\text{reg}}$: 
where \( \omega_i \) is one of the following “influence functions”:

\[
\begin{align*}
\omega &= 1 \\
\omega(d) &= 1 - d \\
\omega(d) &= \left(1 - \left(\frac{3}{2}d - \frac{1}{2}d^2\right)\right) \\
\omega(d) &= a^{-d} \\
\omega(d) &= \min(1, (1 - d) / a)
\end{align*}
\]

Here, \( d \) is a normalized distance from the center of the region (for boxes and ellipsoids) or from the axis of the cylinder. In Eqs. (3d) and (3e), \( a \) is a user-specified parameter, provided at the end of the region-definition parameters (see Table 5). The effect of the influence function is that the farther away the element is from the region’s center, the lower is the weight assigned to the region-specific permeability and porosity, i.e., the properties near the edges of the region approach those of the background material. Note that if Eq. (3a) is chosen, each element within the region will have the same property value. However, changing the geometry of the region will not lead to a smooth, differentiable change of properties; consequently, this parameter value cannot be used identifying the permeability structure using a gradient based algorithm.

The option is invoked by selecting “XYZ” as the first three characters of the material name (see variable MAT in block ROCKS. 1). If a region is requested, the following parameters are read in free format, starting on the next line:

**IPERMGEOM**: Defines geometry of the region
1 = Box
2 = Ellipsoid
3 = Cylinder
4 = Cube

**IPERMINFF**: Defines influence function. If negative, the complementary region is selected (i.e., all elements outside the defined geometry).
0 = Constant (Eq. 3a)
1 = Linear (Eq. 3b)
2 = Spherical (Eq. 3c)
3 = Exponential (Eq. 3d)
4 = Constant-linear (Eq. 3e)

**XREGION(i)** (see Table 5)

**XREGINFF** Parameter \( a \) (only if \(|IPERMINFF| = 3 \text{ or } 4|\)
Table 5. Geometrical Parameters Defining Region

<table>
<thead>
<tr>
<th>IREGGEOM</th>
<th>XREGION(i)</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (box)</td>
<td></td>
<td>Xmin</td>
<td>Ymin</td>
<td>Zmin</td>
<td>Xmax</td>
<td>Ymax</td>
<td>Zmax</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
<tr>
<td>2 (ellipsoid)</td>
<td></td>
<td>Xc</td>
<td>Yc</td>
<td>Zc</td>
<td>Lx</td>
<td>Ly</td>
<td>Lz</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
<tr>
<td>3 (cylinder)</td>
<td></td>
<td>Xs</td>
<td>Ys</td>
<td>Zs</td>
<td>Ze</td>
<td>Ye</td>
<td>Ze</td>
<td>radius</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>4 (cube)</td>
<td></td>
<td>Xc</td>
<td>Yc</td>
<td>Zc</td>
<td>Lx</td>
<td>Ly</td>
<td>Lz</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
</tbody>
</table>

If the region is not aligned with the coordinate axes, set IREGGEOM negative and provide three correction angles (azimuth, dip, and plunge). Figure 4 shows an input file that uses a rotated ellipsoidal region to represent an inclined fault. The parameters XREGION(i) can be varied through the iTOUGH2 command >> REGION and thus be estimated using inverse modeling. For example, the location and extent of the fault can be varied.

Figure 4. TOUGH2 input file for assigning ellipsoidal permeability region.

5.4 Geostatistics

Spatially correlated permeability fields can be generated internally using methods of the geostatistical library GSLIB [Deutsch and Journel, 1992]. A special user’s guide [Finsterle and Kowalsky, 2007] describes the approach and options.
6 Boundary Conditions and Sink/Source Terms

6.1 Time-Dependent Dirichlet Boundary Conditions

Time-dependent Dirichlet boundary conditions can be read from the input file following keyword TIMBC, or from the file given on the line following keyword “TIMBC F”.

FILENAME : Filename with boundary condition data (only if “TIMBC F”); make sure the file is available in the local directory
NTPTAB  : Number of elements with time-dependent boundary conditions
Repeat the following entries NTPTAB times
NBCP, NBCPV  : Number of times and identification number of primary variable
BCELM  : Name of boundary element (start in Column 1)
TIMEBCV, PGBCEL : Time and value of primary variable NBCPV at boundary element BCELM; repeat this entry NBCP times

All values are read in free format. Boundary values will be linearly interpolated between table entries. An example is given in Figure 5.

<table>
<thead>
<tr>
<th>TIMBC---</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2</td>
<td>3</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TOP99</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0</td>
<td>1.0E5</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>86400.0</td>
<td>1.1E5</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1.0E50</td>
<td>1.1E5</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>T 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0</td>
<td>10.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1.0E7</td>
<td>10.9</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 5. TOUGH2 input file for assigning time-dependent Dirichlet boundary conditions.

6.2 Free-Drainage Boundary Condition

A free drainage boundary condition for liquid flow can be implemented, in which gravity is the only driving force, i.e., (capillary) pressure pressure gradients are ignored across an interface to the boundary gridblock. This type of boundary condition comes into effect at each connection in which one of the gridblocks belongs to rock type DRAIN.

6.3 User-Specified Boundary Conditions

In TOUGH2, Neumann boundary conditions are specified by introducing sinks and sources in block GENER. Dirichlet boundary conditions can be implemented by assigning very large volumes to grid blocks adjacent to the boundary so that the thermodynamic
conditions in those elements do not change from fluid and heat exchange with finite-size grid blocks in the model domain.

Prescribed, but time-varying boundary conditions can be implemented by specifying appropriate (large) sinks and sources in grid blocks having a very large volume or by using keyword TIMBC as described in Section 6.1. Moreover, for simple step changes, iTOUGH2 offers an alternative option (see command >> RESTART TIME). Another possibility is described in this section. The user can provide values of the primary variables for selected elements as a function of time. The function has to be programmed into subroutine USERBC, which can be found in file it2user.f. In subroutine USERBC, the user has the possibility to provide the value of one or more primary variables for selected elements as a function of time. For example, these values can be calculated internally or read from a file. The header of subroutine USERBC is shown in Figure 6. The element number \( N \) or element name \( CELEM \) can be used to identify the boundary grid block. The user is supposed to return a value for one or several of the primary variables through array \( X \). In the example given below, a table of time versus pressure data is read from file \( atm\_pres.dat \) and assigned to element 'ATM 0' using the linear interpolation function INTERP1. Note that either the full path to file \( atm\_pres.dat \) must be given, or the file must be copied to the temporary directory using option -fi filename.

Subroutine USERBC is called only if \( MOP(22) \) is either 1 or 2. If \( MOP(22) \) is 2, the EOS module is called after completion of a time step to ensure that the user-specified thermodynamic conditions are updated. If \( MOP(22) \) is 1, subroutine USERBC is called, but no additional call to subroutine EOS is made. This allows one to make time-dependent changes to TOUGH2 variables that are not primary variables, and that do not require a recalculation of the thermodynamic state.
Figure 6. Subroutine USERBC for specifying time-dependent boundary conditions.

6.4 Tabular Input of Time-Dependent Rates

In addition to the standard input format, time-dependent generation rates (i.e., if \( L\)TAB > 1 in block GENER.1) can be provided as a free-format table with time in the first column, injection or production rate in the second column, and (if \( IT\)AB is not left blank) specific enthalpy in the third column. The number of table rows is given by \( L\)TAB. The tabular format is chosen by providing the character “T” or “D” in Column 7 after keyword GENER. Moreover, time and rate conversion factors can be given in Columns 11–20 and 21–30. If character “D” is specified in Column 7, time can be given in (any) date format; it will be converted to seconds (relative to the first date given). These conversion factors only apply to sinks/source with time-dependent generation rates (i.e., constant rates given in Columns 41–50 of block GENER.1 are not affected). Figure 7 shows an example, in which time-dependent water injection rates (in m³/hour) are given as a four-entry table, with time given...
in hours. The options discussed in this section are only available if sinks/sources are given directly in the TOUGH2 input deck. The external file \textit{GENER} has to be provided in the standard format.

\begin{verbatim}
MOMOP====1====2====3====4====5====6====7
1
----7---GTFACT---GRFACT====4====5====6====7
GENER T 3600.0 2.7778E-7 ===4====5====6====7
A11 LINJ 1 0 0 0 4 WATE
0.00 1.0
6.00 1.0
6.01 0.0
1.0e50 0.0
\end{verbatim}

\textbf{Figure 7.} TOUGH2 input format for specifying time-dependent generation rates in tabular format, and using conversion factors for time and rate.

\section{Honoring Generation Times}

By setting \textit{MOP2(3)} = 1 in block \textit{MOMOP} (see Figure 7), each time specified for any sink/source with time-dependent generation rates will be honored in the simulation, i.e., time stepping will automatically be adjusted to coincide with the time when the generation rate changes.

\section{Material-Related Sinks/Sources}

Instead of providing sinks/sources for individual elements, the option described in this section allows one to specify sinks/sources for all elements that belong to a certain material domain. The injection or production rates specified this way are volume-specific, i.e., they will have units of kg/s per m$^3$. A mass rate in kg/s will then be internally calculated based on the volume of the element that belongs to the identified material type. The option is invoked by the sink/source code name “\textit{MATi}” (see variables \textit{SL} and \textit{NS} in block \textit{GENER.1}), where the integer \textit{i} is the sequence number of the material domain as entered in block \textit{ROCKS}. An example is shown in Figure 8.

For example, infiltration can be conveniently specified if a one-meter thick land surface layer assigned to a unique material type is provided. Using material-related volume-specific mass flow rates, the infiltration in kg/s (≈ mm/s) can be provided in a single \textit{GENER}-block entry; it will be internally converted to mass flow rates that are proportional to the surface areas of all infiltration elements.

The model-related sinks/sources option is also convenient to specify volumetric generation rates, e.g., for hydrogen generation due to corrosion, gas from biodegradation in landfills, or radionuclide and gas generation rates in rock masses.
6.7 Sink/Source Regions

Instead of providing sinks/sources for individual elements, the option described in this section allows one to specify a sink/source over a certain geometric region. The location and size of this region is parameterized and can thus be subjected to parameter estimation by inverse modeling (e.g., for contaminant source identification or optimization of well locations). For the sink/source response to be a smooth function of its location and size, the sink/source region must comprise multiple elements, with a smooth function describing the decline of the rate with distance from the center of the region. This option can only be used if element coordinates are provided in columns 51–80 in block ELEME.

The injection or production rate will be distributed over all elements within a user-specified region. This region is defined by the location of its center and the extent. The shape can be either an ellipsoid, a rectangular box or cube, or a cylinder. The ellipsoid and rectangle are aligned with the coordinate axes; the cylinder may be arbitrarily oriented.

The box region includes all elements \( i \) with coordinates \( X_i, Y_i, \) and \( Z_i \) that lie within a rectangular box, i.e.,

\[
\begin{align*}
    d_X &= |X_i - X_c| \leq L_X \\
    d_Y &= |Y_i - Y_c| \leq L_Y \\
    d_Z &= |Z_i - Z_c| \leq L_Z
\end{align*}
\]

(4a)

The ellipsoidal region includes all elements \( i \) with coordinates \( X_i, Y_i, \) and \( Z_i \) that satisfy the equation

\[
    d^2 = \left( \frac{X_i - X_c}{L_X} \right)^2 + \left( \frac{Y_i - Y_c}{L_Y} \right)^2 + \left( \frac{Z_i - Z_c}{L_Z} \right)^2 < 1
\]

(4b)

Here, \( X_c, Y_c, \) and \( Z_c \) are the center coordinates of the box or ellipsoid, and \( L_X, L_Y, \) and \( L_Z \) are the three half-lengths of the box or the three semi-axes of the ellipsoid. The region is aligned with the coordinate axes.

A cylindrical region is defined by the starting and end coordinates of its axis, and a radius. All elements within this arbitrarily oriented cylinder are selected.
The total generation rate $Q$ of the sink/source is distributed among these elements as follows:

$$q_i = Q \frac{\omega_i}{\sum \omega_i} \quad (5)$$

where $\omega_i$ is one of the following “influence functions”:

$$\omega = 1 \quad (6a)$$
$$\omega(d) = V \cdot (1 - d) \quad (6b)$$
$$\omega(d) = V \cdot \left(1 - \left(\frac{3}{2} d - \frac{1}{2} d^3\right)\right) \quad (6c)$$
$$\omega(d) = V \cdot a^{-d} \quad (6d)$$
$$\omega(d) = \min(1, (1 - d) / a) \quad (6e)$$

Here, $V$ is the element volume, and $d$ is a normalized distance from the center of the region (for boxes and ellipsoids) or from the axis of the cylinder. In Eqs. (6d) and (6e), $a$ is a user-specified parameter, to be provided after the region-definition parameters (see Table 6). The effect of the influence function is that the farther away the element is from the region’s center, the smaller is the rate assigned to this element. The rate is also weighted by the element’s volume. Note that if Eq. (6a) is chosen, each element within the region will have the same volume-weighted sink/source strength. However, changing the geometry of the region will not lead to a smooth, differentiable reallocation of the generation rates; consequently, this parameter value cannot be used for contaminant source identification or well location optimization using a gradient-based algorithm.

The option is invoked by the sink/source code name “XYZ..” (see variable SL in block GENER.1; variable NS is an arbitrary two-digit integer). If a region is requested, the following parameters are read in free format, starting on the line following GENER.1:

**IPERMGEOM:** Defines geometry of the region
1 = Box
2 = Ellipsoid
3 = Cylinder
4 = Cube

**IREGINFF:** Defines influence function. If negative, the complementary region is selected (i.e., all sinks/sources outside the defined geometry).
0 = Constant \quad (Eq. 6a)
1 = Linear \quad (Eq. 6b)
2 = Spherical \quad (Eq. 6c)
3 = Exponential \quad (Eq. 6d)
4 = Constant-linear \quad (Eq. 6d)
**XREGION(i)** (see Table 6)

**XREGINFF** Parameter a (only if |IREGINFF| = 3 or 4)

### Table 6. Geometrical Parameters Defining Region

<table>
<thead>
<tr>
<th>IREGGEOEM</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (box)</td>
<td>$X_{min}$</td>
<td>$Y_{min}$</td>
<td>$Z_{min}$</td>
<td>$X_{max}$</td>
<td>$Y_{max}$</td>
<td>$Z_{max}$</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
<tr>
<td>2 (ellipsoid)</td>
<td>$X_c$</td>
<td>$Y_c$</td>
<td>$Z_c$</td>
<td>$L_x$</td>
<td>$L_y$</td>
<td>$L_z$</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
<tr>
<td>3 (cylinder)</td>
<td>$X_S$</td>
<td>$Y_S$</td>
<td>$Z_S$</td>
<td>$Z_E$</td>
<td>$Y_E$</td>
<td>$Z_E$</td>
<td>radius</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>4 (cube)</td>
<td>$X_c$</td>
<td>$Y_c$</td>
<td>$Z_c$</td>
<td>$L_x$</td>
<td>$L_y$</td>
<td>$L_z$</td>
<td>azimuth</td>
<td>dip</td>
<td>plunge</td>
</tr>
</tbody>
</table>

If the region is not aligned with the coordinate axes, set IREGGEOEM negative and provide 3 correction angles (azimuth, dip, and plunge). Figure 9 shows an input file that uses an elliptical source region. Note that the vertical semi-axis (variable $XREGION(6)$) of the ellipsoid is very small, so only elements at an elevation of -4.5 m (variable $XREGION(1)$) will be selected. The parameters $XREGION(i)$, $i=1,...,7$ can be varied through the iTOUGH2 command $>>$ REGION and thus be estimated using inverse modeling.

Since the flow generation rates are calculated internally, set MOP(4) = 2 to see the allocation of the total flow rate of 1 kg/s to the eight elements within the ellipsoid (see Figure 10).
Test ellipsoidal source region option
ROCKS----1----2----3----4----5----6----7
SAND1 0.5 1.000E-12 1.000E-12 1.000E-12
MESHM----1----2----3----4----5----6----7
XYZ
NX 10 1.0
NY 10 1.0
NZ 10 1.0
PARAM----1-MOP: 123456789012345678901234----5----6----7
2 1 1 2
1.0E-5
START----1----2----3----4----5----6----7
INCON----1----2----3----4----5----6----7
GENER----1----2----3----4----5----6----7
1XYZ 1 1 WATER 1.0
2 1 5.0 4.7 -4.5 2.5 1.1 0.1
ENDCY----1----2----3----4----5----6----7

**Figure 9.** TOUGH2 input file for testing ellipsoidal sink/source regions.

QQQQQQQQQQ SUBROUTINE QU QQQQQQQQQQ --- [KCyc, ITER] = [ 1, 1]
ELEMENT A55 4 SOURCE XYZ 1 --- FLOW RATE = 0.132698E+00
ELEMENT A56 4 SOURCE XYZ 1 --- FLOW RATE = 0.203361E-01
ELEMENT A55 5 SOURCE XYZ 1 --- FLOW RATE = 0.259560E+00
ELEMENT A56 5 SOURCE XYZ 1 --- FLOW RATE = 0.874065E-01
ELEMENT A55 6 SOURCE XYZ 1 --- FLOW RATE = 0.259560E+00
ELEMENT A56 6 SOURCE XYZ 1 --- FLOW RATE = 0.874065E-01
ELEMENT A55 7 SOURCE XYZ 1 --- FLOW RATE = 0.132698E+00
ELEMENT A56 7 SOURCE XYZ 1 --- FLOW RATE = 0.203361E-01

**Figure 10.** Excerpt of TOUGH2 output file showing allocation of generation rates within ellipsoidal source region.
7 Relative Permeability and Capillary Pressure Functions

Subroutines RELP and PCAP provide the relative permeability and capillary pressure functions, respectively (see Pruess [1987]). They are frequently modified to accommodate particular needs. The user should therefore carefully check the functional form and required parameters before selecting a certain curve through variable IRP and ICP, respectively (see also command >>> CHARACTERISTIC).

The functions provided in this version include the three-phase curves used by the T2VOC module [Falta et al., 1995]. Additional functions are provided as described in the following subsections.

7.1 Modified Brooks-Corey Model

A modified version of the Brooks-Corey model [Luckner et al., 1989] has been implemented. In order to prevent the capillary pressure from decreasing towards negative infinity as the effective saturation approaches zero, a linear function is used for saturations $S_i$ below a certain value ($S_{lr} + \varepsilon$), where $\varepsilon$ is a small number. The slope of the linear extrapolation is identical with the slope of the capillary pressure curve at $S_i = S_{lr} + \varepsilon$. Alternatively, the capillary pressure is prevented from becoming more negative than $-p_{c,\text{max}}$.

The modified Brooks-Corey model is invoked by setting both IRP and ICP to 10. The model is described by the following set of equations (the input parameters are listed in Table 7):

$$S_{ec} = \frac{S_i - S_{bc}}{1 - S_{bc}}$$  \hspace{1cm} (8a)

$$S_{ek} = \frac{S_i - S_{lk}}{1 - S_{lk} - S_{gr}}$$  \hspace{1cm} (8b)

$$p_c = -p_e (S_{ec})^{-1/\lambda} \quad \text{for} \quad S_i \geq (S_{bc} + \varepsilon)$$  \hspace{1cm} (9a)

$$p_c = -p_e \left( \frac{\varepsilon}{1 - S_{bc}} \right)^{-1/\lambda} + p_e \frac{1}{\lambda} \left( \frac{\varepsilon}{1 - S_{bc}} \right)^{-1/\lambda} (S_i - S_{bc} - \varepsilon) \quad \text{for} \quad S_i < (S_{bc} + \varepsilon)$$  \hspace{1cm} (9b)

$$p_c \geq -p_{c,\text{max}}$$  \hspace{1cm} (10)
\[ k_{rl} = S_{ek}^{\frac{2+3\lambda}{3}} \]  
\[ k_{rg} = \left(1 - S_{ek}\right)^{3} \left(1 - S_{ek}^{\frac{2+3\lambda}{3}}\right) \]  
\[ k_{rg} = 1 - k_{rl} \]  
\( (11a) \)  
\( (11b) \)  
\( (11c) \)

**Table 7.** Input Parameters for Modified Brooks-Corey Model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IRP</td>
<td>10</td>
<td>select Brooks-Corey relative permeability model</td>
</tr>
<tr>
<td>RP(1)</td>
<td>( S_{lrk} )</td>
<td>residual liquid saturation for relative permeability functions</td>
</tr>
<tr>
<td>RP(2)</td>
<td>( S_{gr} )</td>
<td>residual gas saturation</td>
</tr>
<tr>
<td>RP(3)</td>
<td>(flag)</td>
<td>if zero, use (11b), otherwise (11c)</td>
</tr>
<tr>
<td>ICP</td>
<td>10</td>
<td>select Brooks-Corey capillary pressure model</td>
</tr>
<tr>
<td>CP(1)</td>
<td>( \lambda )</td>
<td>pore size distribution index</td>
</tr>
<tr>
<td>CP(2)</td>
<td>( p_e )</td>
<td>gas entry pressure [Pa]</td>
</tr>
<tr>
<td>CP(3)</td>
<td>( \varepsilon ) or ( p_{c,\text{max}} )</td>
<td>if ( CP(3) = 0 ) then ( p_{c,\text{max}} = 10^{50}, \varepsilon = -1 )</td>
</tr>
<tr>
<td>CP(6)</td>
<td>( S_{lrc} )</td>
<td>if zero, then ( S_{lrc} = S_{lrk} )</td>
</tr>
</tbody>
</table>

**Figure 11.** Modified Brooks-Corey relative permeability and capillary pressure curves.
7.2 Modified van Genuchten Model

A modified version of the van Genuchten model [Luckner et al., 1989] has been implemented. In order to prevent the capillary pressure from decreasing towards negative infinity as the effective saturation approaches zero, a linear function is used for saturations $S_i$ below a certain value $(S_{ic} + \varepsilon)$, where $\varepsilon$ is a small number. The slope of the linear extrapolation is identical with the slope of the capillary pressure curve at $S_i = S_{ic} + \varepsilon$. Alternatively, the capillary pressure is prevented from becoming more negative than $-p_{c,\text{max}}$.

The modified van Genuchten model is invoked by setting both IRP and ICP to 11. The model is described by the following set of equations (the input parameters are listed in Table 8):

$$S_{ec} = \frac{S_i - S_{ic}}{1 - S_{ic}}$$  \hspace{1cm} (12a)

$$S_{ekl} = \frac{S_i - S_{rk}}{1 - S_{rk}}$$  \hspace{1cm} (12b)

$$S_{ekg} = \frac{S_i}{1 - S_{gr}}$$  \hspace{1cm} (12c)

$$S_{ecx} = \frac{\varepsilon}{1 - S_{ic}}$$  \hspace{1cm} (12d)

$$p_c = -\frac{1}{\alpha} \left[ (S_{ec})^{(\gamma-1)/m} - 1 \right]^{1/n} \quad \text{for} \quad S_i \geq (S_{ic} + \varepsilon)$$  \hspace{1cm} (13a)

$$p_c = -\frac{1}{\alpha} \left[ S_{ecx}^{(\gamma-1)/m} - 1 \right]^{1/n} - \beta \cdot (S_i - S_{ic} - \varepsilon) \quad \text{for} \quad S_i < (S_{ic} + \varepsilon)$$  \hspace{1cm} (13b)

with

linear extension: \hspace{1cm} \beta = -\frac{(1-\gamma)}{amn} \cdot \frac{1}{(1-S_{ic})} \cdot \left( S_{ecx}^{(\gamma-1)/m} - 1 \right)^{1/n} \cdot \frac{(S_{ic} - S_{ic} - \varepsilon)}{S_{icx}}$$

$$p_c = -\frac{1}{\alpha} \left[ S_{ecx}^{(\gamma-1)/m} - 1 \right]^{1/n} \cdot 10^{\beta(S_i - S_{ic} - \varepsilon)} \quad \text{for} \quad S_i < (S_{ic} + \varepsilon)$$  \hspace{1cm} (13c)

with

log-linear extension: \hspace{1cm} \beta = -\log_{10}(\varepsilon) \cdot \left( \frac{1-m}{m} \cdot \frac{\varepsilon - 1}{\varepsilon} \cdot \frac{1}{S_{ecx}^{(\gamma-1)/m} - 1} \right)$$

$$p_c \geq -p_{c,\text{max}}$$  \hspace{1cm} (13d)
\[ k_{rl} = S_{ekl}^\gamma \cdot S_{ekl}^{(1-\gamma)\eta} \cdot \left[ 1 - \left( 1 - S_{ekl}^{(1-\gamma)/\eta} \right)^m \right]^2 \]  \hspace{1cm} (14a)

\[ k_{rg} = (1 - S_{ekg})^{\gamma} \cdot \left[ 1 - S_{ekg}^{1/\eta} \right]^{2m} \]  \hspace{1cm} (14b)

or

\[ k_{rg} = 1 - k_{rl} \]  \hspace{1cm} (14c)

**Figure 12.** Modified van Genuchten relative permeability and capillary pressure curves.
Table 8. Input Parameters for Modified van Genuchten Model

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Relative Permeability Function</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>IRP</strong></td>
<td>11</td>
<td>select van Genuchten relative permeability model</td>
</tr>
<tr>
<td><strong>RP(1)</strong></td>
<td>$S_{\text{lrk}}$</td>
<td>residual liquid saturation for relative permeability functions, if negative, $S_{\text{lrk}} = 0$ for calculating gas relative permeability, absolute value is used for calculating liquid relative permeability</td>
</tr>
<tr>
<td><strong>RP(2)</strong></td>
<td>$S_{\text{gr}}$</td>
<td>residual gas saturation, if negative, $S_{\text{gr}} = 0$ for calculating liquid relative permeability, absolute value is used for calculating gas relative permeability</td>
</tr>
<tr>
<td><strong>RP(3)</strong></td>
<td>(flag)</td>
<td>if zero, use (14b), if non-zero, use (14c)</td>
</tr>
<tr>
<td><strong>RP(4)</strong></td>
<td>$\eta$</td>
<td>exponent in (14a), default = $1/2$</td>
</tr>
<tr>
<td><strong>RP(5)</strong></td>
<td>$\varepsilon_k$</td>
<td>use linear function between $k_r(S_c = 1 - \varepsilon_k)$ and 1.0.</td>
</tr>
<tr>
<td><strong>RP(6)</strong></td>
<td>$a_{fm}$</td>
<td>Constant fracture-matrix interaction reduction factor, in combination with Active Fracture Model (see Section 8)</td>
</tr>
<tr>
<td><strong>RP(7)</strong></td>
<td>$\zeta$</td>
<td>exponent in (10b), default = $1/3$</td>
</tr>
<tr>
<td><strong>Capillary Pressure Function</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>ICP</strong></td>
<td>11</td>
<td>select van Genuchten capillary pressure model</td>
</tr>
<tr>
<td><strong>CP(1)</strong></td>
<td>$n$</td>
<td>parameter related to pore size distribution index (see also $CP(4)$)</td>
</tr>
<tr>
<td><strong>CP(2)</strong></td>
<td>$1/\alpha$</td>
<td>parameter related to gas entry pressure [Pa]</td>
</tr>
<tr>
<td> </td>
<td>$USERX(4,N) &gt; 0$</td>
<td>$1/\alpha_i = USERX(4,N)$</td>
</tr>
<tr>
<td> </td>
<td>$USERX(4,N) &lt; 0$</td>
<td>$1/\alpha_i = USERX(4,N) \cdot CP(2)$</td>
</tr>
<tr>
<td> </td>
<td>$CP(2) &lt; 0$: apply Leverett scaling rule: $1/\alpha_i = 1/\alpha_{\text{ref}} \cdot \sqrt{k_i / k_{\text{ref}}}$</td>
<td>where: $1/\alpha_{\text{ref}} =</td>
</tr>
<tr>
<td> </td>
<td>$k_{\text{ref}} = \text{PER(NMAT)}$</td>
<td></td>
</tr>
<tr>
<td> </td>
<td>$USERX(1,N) &gt; 0$</td>
<td>$k_i = USERX(1,N)$</td>
</tr>
<tr>
<td> </td>
<td>$USERX(1,N) &lt; 0$</td>
<td>$k_i = USERX(1,N) \cdot \text{PER(NMAT)}$</td>
</tr>
<tr>
<td><strong>CP(3)</strong></td>
<td>$\varepsilon$ or $p_{c,\text{max}}$</td>
<td>$CP(3) = 0$ : $p_{c,\text{max}} = 10^{30}$, $\varepsilon = -1$</td>
</tr>
<tr>
<td> </td>
<td>$0 &lt; CP(3) &lt; 1$</td>
<td>$\varepsilon = CP(3)$; use linear extension (13b)</td>
</tr>
<tr>
<td> </td>
<td>$CP(3) \geq 1$</td>
<td>$p_{c,\text{max}} = CP(3)$, $\varepsilon = -1$</td>
</tr>
<tr>
<td> </td>
<td>$-1 &lt; CP(3) &lt; 0$</td>
<td>$\varepsilon =</td>
</tr>
<tr>
<td><strong>CP(4)</strong></td>
<td>$m$</td>
<td>if zero then $m = 1 - 1/CP(1)$, else $m = CP(4)$ and $n = 1/(1-m)$</td>
</tr>
<tr>
<td><strong>CP(5)</strong></td>
<td>$T_{\text{ref}}$</td>
<td>if negative, $</td>
</tr>
<tr>
<td><strong>CP(6)</strong></td>
<td>$\gamma$</td>
<td>parameter of Active Fracture Model (see Section 8)</td>
</tr>
<tr>
<td><strong>CP(7)</strong></td>
<td>$S_{\text{lr}}$</td>
<td>if zero, then $S_{\text{lr}} = S_{\text{nrk}}$</td>
</tr>
</tbody>
</table>
8 Active Fracture Model

8.1 Active Fracture Concept

There is evidence that only a portion of the connected fracture network conducts water under unsaturated conditions. The fractures contributing to liquid flow are referred to as “active fractures”. The Active Fracture Concept (AFC) was developed by Liu et al. [1998] to describe gravity-dominated, non-equilibrium, preferential liquid flow in fractures, which is expected to be similar to fingering in unsaturated porous media. AFC is based on the hypothesis that (1) the number of active fractures is small compared with the total number of connected fractures, (2) the number of active fractures within a grid block is large so that the continuum approach is valid, and (3) the fraction of active fractures, $f_a$, is related to water flux and equals one for a fully saturated system, and zero if the system is at residual saturation. The following power function of effective liquid saturation, $S_e$, fulfills these conditions:

$$f_a = S_e^\gamma$$

(15)

Here, $\gamma$ is a positive constant depending on properties of the fracture network, and $S_e$ is the effective liquid saturation given by

$$S_e = \frac{S_l - S_w}{1 - S_{lr}}$$

(16)

Capillary pressure and relative permeability functions are modified to account for the fact that the effective saturation in the active fractures, $S_{ea}$, is larger than the effective saturation of the total fracture continuum:

$$S_{ea} = \frac{S_e}{f_a} = S_e^{1-\gamma}$$

(17)

Using the van Genuchten model, capillary pressure and liquid relative permeability are given, respectively, by

$$p_e = -\frac{1}{\alpha_e} \left[ S_e^{(\gamma-1)/m} - 1 \right]^{1/n}$$

(18)

and

$$k_{rl} = S_e^{(1+\gamma)/2} \left\{ 1 - \left[ 1 - S_e^{(1-\gamma)/m} \right]^2 \right\}$$

(19)

The fracture-matrix interface area reduction factor (see Section A8) is given by

$$a_{fm} = S_e^{1+\gamma}$$

(20)

The AFC is invoked by selecting $\gamma > 0$, which is provided as an additional parameter of the standard van Genuchten model (ICP=7) through variable $CP$ ($6$, $NMAT$). Fracture-matrix interface area reduction is invoked by selecting $ISOT$ between -10 and -12 (see Table 9).
8.2 Reduction of Fracture-Matrix Interface Area

There is evidence that fracture-matrix interaction in the unsaturated zone is reduced as a result of fracture coatings as well as preferential flow in the fractures as invoked by flow instabilities (fingering) and small-scale heterogeneities. A number of options for reducing fracture-matrix interface area have been implemented for use in a dual-permeability flow simulation. Interface area reduction is applied to connections with a negative value for variable ISOT, which is provided in the CONNE block. Different modifiers are used depending on the value of ISOT and MOP(8) as summarized in Table 9.

Table 9. Option for Reducing Fracture-Matrix Interface Area

<table>
<thead>
<tr>
<th>ISOT</th>
<th>MOP(8)</th>
<th>Interface area reduction factor $a_{fm}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>positive</td>
<td>any</td>
<td>No interface area reduction, i.e., $a_{fm} = 1$</td>
</tr>
<tr>
<td>negative</td>
<td>1</td>
<td>$a_{fm} = RP(6, NMAT)$</td>
</tr>
<tr>
<td>-1, -2, -3</td>
<td>0</td>
<td>$a_{fm} = S_\beta$</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>$a_{fm} = S_\beta \cdot RP(6, NMAT)$</td>
</tr>
<tr>
<td>-4, -5, -6</td>
<td>0</td>
<td>$a_{fm} = k_{r\beta}$</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>$a_{fm} = k_{r\beta} \cdot RP(6, NMAT)$</td>
</tr>
<tr>
<td>-7, -8, -9</td>
<td>0</td>
<td>$a_{fm} = RP(6, NMAT)$</td>
</tr>
<tr>
<td>-10, -11, -12</td>
<td>0</td>
<td>$a_{fm} = S_e^{1+\gamma}$ (see Section 8.1)</td>
</tr>
</tbody>
</table>

- $a_{fm}$: Fracture-matrix interface area reduction factor.
- $S_\beta$: For flow of phase $\beta$, upstream saturation of phase $\beta$.
- $k_{r\beta}$: For flow of phase $\beta$, upstream relative permeability of phase $\beta$.
- $RP(6, NMAT)$: 6th parameter of rel. perm. function of upstream element; if zero (i.e., not specified), reset to one.
9 Coupled Overland – Subsurface Flow

iTOUGH2 provides the capability to fully couple overland flow (solving the non-inertial, diffusion wave form of the Saint-Venant equations) with subsurface flow using an approach similar to that proposed by Weill et al. [2009]. The momentum and continuity equations are given by:

\[ S_{fi} = -\nabla (z_i + h_s) \]  
(21)

\[ \frac{\partial h_i}{\partial t} + \nabla \cdot (h_i \bar{U}) = q_s \]  
(22)

where \( S_{fi} \) is the friction slope [-] in the direction \( i \), \( z_i \) is land surface elevation [L], \( h_s \) is the water depth on the surface, \( \bar{U} \) is the depth averaged flow velocity [LT^{-1}], and \( q_s \) is a source/sink term [LT^{-1}]. The Manning-Strickler formula is used for relating velocity to friction slopes:

\[ U_i = \frac{h_i^{2/3}}{n_{man}} \sqrt{S_{fi,i}} \]  
(23)

where \( n_{man} \) is the Manning roughness coefficient [L^{-1/3}T]. The diffusion-wave form of the Saint-Venant equations assumes slowly varying flow.

In order to couple the surface and subsurface flow equations, the approach developed by Weill et al. [2009] is followed. A surface layer of thickness \( e \) is expected to be present at the top of the numerical model. For liquid flow within the surface layer, Eqs. (21)–(23) are combined into a form that is similar to that describing flow in a porous medium:

\[ \frac{\partial h_i}{\partial t} - \nabla \cdot \left( K_s \nabla (z_i + h_s) \right) = q_s \]  
(24)

Here, the non-diagonal terms of the hydraulic conductivity tensor \( K_s \) are zero and the diagonal components are

\[ K_{s,xx} = \frac{h_i^{5/3}}{n_{man} \sqrt{\nabla_x(z_i + h_s)}} \]  
(25)

\[ K_{s,yy} = \frac{h_i^{5/3}}{n_{man} \sqrt{\nabla_y(z_i + h_s)}} \]  
(26)

\[ K_{s,zz} = k_{zz} \frac{k_{ii}}{\mu_i} \]  
(27)
The horizontal hydraulic conductivities describe surface water flow, while the vertical hydraulic conductivity describes resistance to liquid flow between the surface and subsurface layer, with \( k_{zz} \) equal to the vertical permeability of the subsurface layer. The liquid pressure in the surface layer is assumed hydrostatic. Because liquid and gas pressures are continuous across the surface/subsurface boundary, negative water depths occur when there is no runoff. The volumetric liquid content in the surface layer is defined as

\[
\theta_l = \begin{cases} 
0 & \text{for } h_s < 0 \\
\frac{h_s}{e} & \text{for } h_s \geq 0 
\end{cases} 
\]  

For vertical liquid flow, the liquid relative permeability is set to one, unless \( h_s/e < 10^{-5} \), when it is specified as zero. To capture the pressure head due to ponding in the surface layer, a positive capillary pressure is calculated as a function of \( h_s \).

For gas flow within the surface layer and between the surface and subsurface layers, the regular subsurface flow equations are used. If runoff occurs in the surface layer, i.e., \( \theta_l > 0 \) then \( k_{rg} = 0 \) for pressure gradients from the surface to the subsurface layers such that no gas flows between the surface and subsurface layers (note, however, that it is possible for pressurized gas to escape the subsurface and flow to the surface layer), and \( k_{rg} = 1 \) within the surface layer such that gas flows freely in the surface layer. If there is no runoff, \( \theta_l = 0 \), \( k_{rg} = 1 \) and the intrinsic permeability of the surface layer is assumed isotropic and equal to the vertical intrinsic permeability of the subsurface layer.

To implement coupled surface water – groundwater flow, the user has to set up a TOUGH2 model—similar to that shown in Figure 13—as follows:

1. Create a material in block ROCKS named SURWA. All elements representing surface water must be related to this material type.
2. For material SURWA, set porosity close to 1.0, set \( NAD=1 \), and provide two Manning’s coefficients referring to the first and second direction (i.e., for \( ISOT=1 \) and 2) and the surface layer thickness (consistent with its definition in block ELEME) in columns 51–60, 61–70, and 71–80, respectively. Do not specify material-dependent relative permeability and capillary pressure function using \( NAD=2 \); these functions are provided internally.
3. To specify a zero-water-depth-gradient boundary condition, create boundary elements (either inactive or of large volume), assign them to a material named SURZG, and connect them to the surface water elements at the desired locations.
4. Create a material in block ROCKS named ATMOS and assign properties suitable for representing atmospheric conditions.
5. Note that the absolute permeabilities in materials SURWA, SURGZ, and ATMOS will be used to calculate gas flow only; liquid flow will be determined by the Saint-Venant equation.
(6) Generate a mesh with an atmospheric element (or layer) and a surface-water layer (typically of thickness 1 m). Assign materials ATMOS and SURWA to these two layers.

(7) Connect the atmosphere to the surface-water layer, and the surface-water layer to the subsurface system. Connections between surface-water elements must be assigned to directions ISOT=1 and 2; the slope of the surface is provided through variable BETAX. Connections between surface-water and subsurface elements must be assigned to direction ISOT=3; the nodal distance from the surface-water element to the interface with the subsurface element is internally set to zero.

Surface-water flow is solved simultaneously and fully coupled with subsurface flow using the standard TOUGH2 implicit scheme. Note that time-step size may be governed by the relatively fast flow occurring in the surface-water layer.
Figure 13. TOUGH2 input file for testing coupled surface water–subsurface flow.
10 Semi-Analytical Radial Heat Exchange

Radial, conductive heat exchange between fluids in a discretized wellbore and the formation is calculated using a semi-analytical, time-convolution method. The time-dependent temperature evolution in the wellbore is calculated numerically using TOUGH2. At each time step, radial heat transfer with the formation is calculated by superposition of analytical solutions of heat flow that are dependent on the temperature differences between subsequent time steps.

Carslaw and Jaeger [1959, pp. 334–339] provided an approximate solution for heat conduction between a cylinder and surrounding media where the temperature of the cylinder is maintained constant. If the initial temperature difference between the two domains is $\Delta T = T_w - T_f$ (where $T_w$ and $T_f$ are the temperatures in the well and the formation, respectively), the heat flux $q$ from the wellbore to the formation can be calculated using as the product of a heat transfer function and the temperature using Equ. $(29)$ for small values of the dimensionless time $t_d = \alpha t / r_0^2$, where $\alpha$ is the thermal diffusivity, and Equ. $(30)$ for large values of $t_d$:

\[
q = f_1(t_d) \cdot \Delta T = \frac{k \Delta T}{r_0} \left\{ \frac{1}{2} - \frac{1}{4} \left( \frac{t_d}{\pi} \right)^{0.5} + \frac{1}{8} t_d - \cdots \right\} \quad (29)
\]

\[
q = f_2(t_d) \cdot \Delta T = \frac{2k \Delta T}{r_0} \left\{ \frac{1}{\ln(4t_d) - 2\gamma} - \frac{\gamma}{\left[ \ln(4t_d) - 2\gamma \right]^2} - \cdots \right\} \quad (30)
\]

Here, $k$ is thermal conductivity (W m$^{-1}$ K$^{-1}$), $r_0$ is the wellbore radius (m), and $\gamma$ is the Euler constant ($0.57722$).

The heat transfer functions $f_1$ and $f_2$ express the amount of heat flux with time due a unit temperature difference. As shown in Zhang et al. [2011], the heat transfer functions $f_1$ and $f_2$ are approximately the same at the dimensionless time $t_d = 2.8$. Therefore, $t_d = 2.8$ is considered the critical dimensionless time to switch from $f_1$ to $f_2$.

During fluid injection and production, and as a result of the heat exchange processes, temperature changes continuously over time at any point within the wellbore and at the wellbore-formation interface. Based on superposition, the radial heat flux across each wellbore element to the surrounding formation is a time-convolution result of varying temperature. The discretized form at each time step can be expressed by the following:

\[
q_{total} = \sum_{i=1}^{d-1} f(t_d - t_i) \cdot \Delta T(t_i) \quad (31)
\]

Here, $t_d$ represents the current time after $d$ time steps, and $t_i$ represents the time after $i$ time steps; the function $f$ is $f_1$ if $t_d - t_i \leq 2.8$, and $f_2$ if $t_d - t_i > 2.8$. The temperature difference
\( \Delta T(t_i) \) is the temperature in the well at time step \( i \), minus the formation temperature at the interface at the previous time step, i.e., \( \Delta T(t_i) = T_w(t_i) - T_f(t_{i-1}) \).

To implement the solution into the TOUGH2 simulator, we need to calculate \( q_{\text{total}} \) and its derivative at each time step, and incorporate these two terms into the heat balance equation and the corresponding linearized form, which is needed for the implicit solution of the fully coupled system of mass and heat flow equations in the well. This requires the algorithm to store the temperature history for each wellbore element. This may be problematic if the time history becomes very long, which increases the computational demand of the time-convolution approach, and potentially reaches the limit of the computer’s storage capacity. To mitigate this problem, a maximum number of time steps can be defined, beyond which the contributions from earlier temperature changes are lumped into a single term, which is calculated based on the time-weighted average of the individual temperature changes \( \Delta T(t_i) \) and associated times \( t_i \).

To make the algorithm flexible for handling various wellbore configurations and thermal conditions in the rock formation, the code gives the user an option to choose between uniform or depth-dependent formation properties, wellbore radii, and geothermal gradients.

The radial semi-analytical heat exchange model is currently incompatible with the option to perform semi-analytical linear heat exchange with confining beds (see \( MOP(15) = 1 \) in Pruess et al. [1999]). There are two options (selected by \( MOP(15) = 5 \) or 6, respectively) to invoke radial heat exchange:

**Option 1 (MOP (15)=5): Constant Well and Formation Properties**

Provide a material named QLOSS with the following parameters:

- **DROK**: Rock grain density \([\text{kg/m}^3]\) of formation near well
- **POR**: Well radius \([\text{m}]\)
- **PER(1)**: Reference elevation \([\text{m}]\); specify Z coordinate in block **ELEME**, Columns 71–80
- **PER(2)**: Reference temperature \([\text{°C}]\)
- **PER(3)**: Geothermal gradient \([\text{°C/m}]\)
- **CWET**: Heat conductivity near well \([\text{W/kg °C}]\) of formation near well
- **SPHT**: Rock grain specific heat \([\text{J/kg °C}]\) of formation near well

**Option 2 (MOP (15)=6): Variable Well and Formation Properties**

Provide an external file named radqloss.dat with information in the following format:

- First line: **NMATQLOSS**: number of elevations with geometric and thermal data
- Provide **NMATQLOSS** lines with the following data in free format:
  - Elevation \([\text{m}]\), well radius \([\text{m}]\), initial temperature \([\text{°C}]\), CWET, DROK, SPHT

Between elevations, properties are calculated using linear interpolation. Figure 14 shows an example input file using Option 1, for the simple heat injection into a single gridblock [Zhang et al., 2011].
Semi-analytical radial heat exchange
ROCKS-----1-----2-----3-----4-----5-----6-----7-----8
WELL0 2650.  0.01  1.E-09  1.E-09  1.E-09  2.10  1000.
QLOSS  2650.  0.05  -.5000  20.  0.00  2.10  1000.
ELEME-----1-----2-----3-----4-----5-----6-----7-----8
A1  1  10.7854E-020.3142E+00 0.2500E-01  -.5000E+00
CONNE-----1-----2-----3-----4-----5-----6-----7-----8
MULTI-----1-----2-----3-----4-----5-----6-----7-----8
PARAM-----1 MOP: 123456789*123456789*123456789
2 3 2 6
GENER-----1-----2-----3-----4-----5-----6-----7-----8
A1 1INJ 1  12  HEATA
0.0E7  1.0E7  2.0E7  3.0E7
4.0E7  5.0E7  6.0E7  7.0E7
8.0E7  9.0E7  1.0E8  5.0E7
8.e+1  2.e+1  8.e+1  2.e+1
8.e+1  2.e+1  8.e+1  2.e+1
8.e+5  2.e+5  8.e+5  2.e+5
8.e+5  2.e+5  8.e+5  2.e+5
8.e+5  2.e+5  8.e+5  2.e+5
ENDCY-----1-----2-----3-----4-----5-----6-----7-----8

Figure 14. TOUGH2 input file for testing radial heat exchange option.
ACKNOWLEDGMENT

This work was supported, in part, by the U.S. Dept. of Energy under Contract No. DE-AC02-05CH11231.

REFERENCES


Finsterle, S., iTOUGH2 Command Reference, Report LBNL-40401 (Updated reprint), Lawrence Berkeley National Laboratory, Berkeley, Calif., 2015b.

Finsterle, S., iTOUGH2 Sample Problems, Report LBNL-40402 (Updated reprint), Lawrence Berkeley National Laboratory, Berkeley, Calif., 2015c.


