UDEC

Universal Distinct Element Code
Example Applications

©2011
Itasca Consulting Group Inc.
Mill Place
111 Third Avenue South, Suite 450
Minneapolis, Minnesota 55401 USA

Phone: (1) 612-371-4711
Fax: (1) 612-371-4717
E-Mail: software@itascag.com
Web: www.itascag.com
**PRECIS**

This volume contains documentation on a series of example problems that have been solved using *UDEC*. These example applications demonstrate the various classes of problems to which *UDEC* may be applied.*

Table 1 presents a summary of the example applications. The table also identifies the specific *UDEC* feature that is examined in each problem.

In addition, verification problems and example applications for specific features are provided in *Constitutive Models*, in *Special Features* and in *Theory and Background*: continuously yielding joint model in *Section 2 in Constitutive Models*, structural elements in *Section 1 in Special Features*, fluid flow analysis in *Section 2 in Special Features*, thermal analysis in *Section 3 in Special Features*, dynamic analysis in *Section 4 in Special Features*, and energy calculations in *Section 3 in Theory and Background*.

The problems in this volume represent a brief sampling of potential applications for *UDEC*. We plan to update this volume on a regular basis and will send new examples as they are prepared. We also invite users to submit their own examples for inclusion, or inform us of any type of problem they would like to see in this volume.

* All problems in this volume were run on a Core 2 i7 computer running Windows 7.
### Example Applications

#### Table 1  Summary of example applications

<table>
<thead>
<tr>
<th>DESCRIPTION</th>
<th>PAGE</th>
<th>INPUT FILE</th>
<th>BLOCK MODEL</th>
<th>JOINT MODEL</th>
<th>STRUCTURAL ELEMENTS</th>
<th>ANALYSES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic-Induced Groundfall</td>
<td>1.1</td>
<td>SEISMIC.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Open Stopping Using Vertical Retreat</td>
<td>2.1</td>
<td>STOPE.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Tunnel Support Loading</td>
<td>3.1</td>
<td>TUNNEL.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Gravity Dam: Fluid Flow and Dynamic Loading</td>
<td>4.1</td>
<td>DAM.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Cement Grouting Simulation</td>
<td>5.1</td>
<td>GROUT.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Thermomechanical Analysis of a Waste Emplacement Drift</td>
<td>6.1</td>
<td>DRIFT.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Inflow into a Tunnel</td>
<td>7.1</td>
<td>INFLOW.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Flow through a Jointed Rock Slope</td>
<td>8.1</td>
<td>SLOPEFLO.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Flow from a Borehole in a Biaxial Stress Field</td>
<td>9.1</td>
<td>BH.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Influence of the Placement of Backfill in a Deep Longwall Excavation</td>
<td>10.1</td>
<td>LONGWALL.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Shalecrete and Cable Support</td>
<td>11.1</td>
<td>SUPPORT.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Blocks Bouncing down Slope</td>
<td>12.1</td>
<td>BOUNCE.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Step-Path Failure of Rock Slopes</td>
<td>13.1</td>
<td>STEPPATH.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Hydraulic Fracturing Simulation</td>
<td>14.1</td>
<td>HF_CASE1.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td></td>
<td></td>
<td>HF_CASE2.DAT</td>
<td>X</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>
# TABLE OF CONTENTS

1 **Seismic-Induced Groundfall**
   1.1 Problem Statement ...................................................... 1-1
   1.2 *UDEC* Analysis ........................................................ 1-2
   1.3 Listing of Data File .................................................... 1-9

2 **Open Stoping Using Vertical Retreat**
   2.1 Problem Statement ...................................................... 2-1
   2.2 *UDEC* Analysis ........................................................ 2-3
   2.3 Listing of Data File .................................................... 2-5

3 **Tunnel Support Loading**
   3.1 Problem Statement ...................................................... 3-1
   3.2 *UDEC* Analysis ........................................................ 3-2
   3.3 Listing of Data File .................................................... 3-7

4 **Gravity Dam: Fluid Flow and Dynamic Loading**
   4.1 Problem Statement ...................................................... 4-1
   4.2 *UDEC* Analysis ........................................................ 4-1
   4.3 Listing of Data File .................................................... 4-11

5 **Cement Grouting Simulation**
   5.1 Problem Statement ...................................................... 5-1
   5.2 *UDEC* Analysis ........................................................ 5-2
   5.3 References ............................................................. 5-3
   5.4 Listing of Data File .................................................... 5-8

6 **Thermomechanical Analysis of a Waste Emplacement Drift**
   6.1 Problem Statement ...................................................... 6-1
   6.2 *UDEC* Analysis ........................................................ 6-3
   6.3 Results ................................................................. 6-5
   6.4 References ............................................................. 6-8
   6.5 Listing of Data File .................................................... 6-8

---

*UDEC Version 5.0*
## Inflow into a Tunnel

7.1 Problem Statement ........................................... 7 - 1
7.2 UDEC Analysis ............................................... 7 - 2
7.3 Listing of Data File ........................................... 7 - 9

## Flow through a Jointed Rock Slope

8.1 Problem Statement ........................................... 8 - 1
8.2 UDEC Analysis ............................................... 8 - 2
8.3 Listing of Data File ........................................... 8 - 6

## Flow from a Borehole in a Biaxial Stress Field

9.1 Problem Statement ........................................... 9 - 1
9.2 UDEC Analysis ............................................... 9 - 2
9.3 Listing of Data File ........................................... 9 - 5

## Influence of the Placement of Backfill in a Deep Longwall Excavation

10.1 Problem Statement ........................................ 10 - 1
10.2 UDEC Analysis ............................................. 10 - 4
10.3 Listing of Data File ....................................... 10 - 9

## Shotcrete and Cable Support

11.1 Problem Statement ...................................... 11 - 1
11.2 UDEC Analysis ........................................... 11 - 3
11.3 Listing of Data Files ................................... 11 - 6

## Blocks Bouncing down Slope

12.1 Problem Statement ....................................... 12 - 1
12.2 UDEC Analysis ........................................... 12 - 2
12.3 Listing of Data File ....................................... 12 - 4

## Step-Path Failure of Rock Slopes

13.1 Problem Statement ....................................... 13 - 1
13.2 UDEC Analysis ........................................... 13 - 2
13.3 Listing of Data File ....................................... 13 - 7
14 Hydraulic Fracturing Simulation

14.1 Problem Statement ........................................... 14-1
14.1.1 Data for the Simulations .................................... 14-1

14.2 *UDEC* Analyses .............................................. 14-2
14.2.1 Case 1 ...................................................... 14-2
   14.2.1.1 *UDEC* Model ....................................... 14-2
   14.2.1.2 Analytical Solution .................................. 14-3
   14.2.1.3 *UDEC* Results ..................................... 14-3
14.2.2 Case 2 ...................................................... 14-4
   14.2.2.1 *UDEC* Model ....................................... 14-4
   14.2.2.2 *UDEC* Results ..................................... 14-6

14.3 Summary ..................................................... 14-11
14.4 References .................................................. 14-11
14.5 Listing of Data Files .......................................... 14-12
# TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Properties of rock types</td>
<td>2 - 2</td>
</tr>
<tr>
<td>6.1</td>
<td>Thermal and mechanical properties used in thermomechanical analysis of a</td>
<td>6 - 2</td>
</tr>
<tr>
<td></td>
<td>waste emplacement drift</td>
<td></td>
</tr>
<tr>
<td>14.1</td>
<td>Data for the simulations</td>
<td>14 - 1</td>
</tr>
</tbody>
</table>
FIGURES

Figure 1.1  UDEC model for seismic-induced groundfall .......................... 1-1
Figure 1.2  Stress distribution around excavation at end of excavation stage .... 1-3
Figure 1.3  Stress distribution in roof of excavation after 0.02 sec. (applied stress = 1.25 × cos(2π 100t)) .......................................................... 1-3
Figure 1.4  y-displacement histories for two points on excavation boundary (applied stress = 1.25 × cos(2π 100t)) ........................................ 1-4
Figure 1.5  Plot of y-velocity at top of model (applied stress = 1.25 × cos(2π 100t)) ........................................ 1-5
Figure 1.6  Stress distribution in roof of excavation after 0.02 sec. (applied stress = 12.5 × cos(2π 100t)) ...................................................... 1-6
Figure 1.7  y-displacement histories for two points on excavation boundary (applied stress = 12.5 × cos(2π 100t)) ........................................ 1-6
Figure 1.8  Stress distribution around excavation after 0.25 sec. (applied stress = 12.5 × cos(2π 100t)) ...................................................... 1-7
Figure 1.9  Stress distribution around excavation after 0.50 sec. (applied stress = 12.5 × cos(2π 100t)) ...................................................... 1-7
Figure 1.10 Stress distribution around excavation after 0.75 sec. (applied stress = 12.5 × cos(2π 100t)) ...................................................... 1-8
Figure 1.11 Plot of y-velocity at top of model (applied stress = 12.5 × cos(2π 100t)) ........................................ 1-8
Figure 2.1  Initial geometry for blasthole open stoping operation .................. 2-1
Figure 2.2  Cohesion properties assigned to the UDEC model ...................... 2-3
Figure 2.3  Principal stresses at end of mining sequence ................................ 2-4
Figure 2.4  Changes in xx- and yy-stresses in crown pillar with mining stages History 2: xx-stress, History 3: yy-stress (note: compressive stresses are negative) . 2-4
Figure 3.1  Idealized geometry of service tunnel and main tunnel .................. 3-1
Figure 3.2  UDEC model zoning with service tunnel excavated .................... 3-3
Figure 3.3  Elastic displacements due to excavation of service tunnel ............ 3-3
Figure 3.4  Displacements after mining of main tunnel .............................. 3-5
Figure 3.5  Principal stress distribution after mining of main tunnel ............... 3-5
Figure 3.6  Thrust in liners after water level is raised ............................... 3-6
Figure 3.7  Moment in liners after water level is raised ............................... 3-6
Figure 4.1  UDEC model of gravity dam with principal stresses plotted at Stage 1 .......................................................... 4-1
Figure 4.2  Principal stress state and displacements at Stage 2 ..................... 4-3
Figure 4.3  x- and y-components of displacement at crest of dam during Stage 2 reservoir filling .............................................................. 4-3
Figure 4.4  Flow rates at Stage 2 ........................................................... 4-4
Figure 4.5  Shear and normal displacements along joint at x = −33.42, y = −30.37 . 4-4
Figure 4.6  Fluid pressure in domain at location x = −22.1, y = −26.3 ............. 4-5
Figure 4.7  x-velocity histories at dam crest and model base at Stage 3 .......... 4-7
Figure 4.8  x- and y-displacements at dam crest at Stage 3 ......................... 4-7

UDEC Version 5.0
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 4.9</td>
<td>Joint shear and normal displacement histories at joint beneath dam foundation at Stage 3</td>
<td>4-8</td>
</tr>
<tr>
<td>Figure 4.10</td>
<td>Displacements, principal stresses and joint shear displacement at Stage 3</td>
<td>4-8</td>
</tr>
<tr>
<td>Figure 4.11</td>
<td>$x$-velocity histories at dam crest and model base at Stage 3</td>
<td>4-9</td>
</tr>
<tr>
<td>Figure 4.12</td>
<td>$x$- and $y$-displacements at dam crest at Stage 3</td>
<td>4-9</td>
</tr>
<tr>
<td>Figure 4.13</td>
<td>Joint shear and normal displacement histories at joint beneath dam foundation at Stage 3</td>
<td>4-10</td>
</tr>
<tr>
<td>Figure 4.14</td>
<td>Displacements, principal stresses and joint shear displacement at Stage 3</td>
<td>4-10</td>
</tr>
<tr>
<td>Figure 5.1</td>
<td>UDEC model and initial stress state for grouting simulation</td>
<td>5-2</td>
</tr>
<tr>
<td>Figure 5.2</td>
<td>Flow rates versus hole pressure</td>
<td>5-4</td>
</tr>
<tr>
<td>Figure 5.3</td>
<td>Flow rates for grouting simulation at 10,000 Pa</td>
<td>5-4</td>
</tr>
<tr>
<td>Figure 5.4</td>
<td>Flow rates for grouting simulation at 12,000 Pa</td>
<td>5-5</td>
</tr>
<tr>
<td>Figure 5.5</td>
<td>Flow rates for grouting simulation at 16,000 Pa</td>
<td>5-5</td>
</tr>
<tr>
<td>Figure 5.6</td>
<td>Grout pressures in joints for grouting simulation at 10,000 Pa</td>
<td>5-6</td>
</tr>
<tr>
<td>Figure 5.7</td>
<td>Grout pressures in joints for grouting simulation at 12,000 Pa</td>
<td>5-6</td>
</tr>
<tr>
<td>Figure 5.8</td>
<td>Grout pressures in joints for grouting simulation at 16,000 Pa</td>
<td>5-7</td>
</tr>
<tr>
<td>Figure 6.1</td>
<td>Conceptual model of vertical emplacement concept</td>
<td>6-1</td>
</tr>
<tr>
<td>Figure 6.2</td>
<td>UDEC geometry for thermomechanical analysis of a waste emplacement drift</td>
<td>6-4</td>
</tr>
<tr>
<td>Figure 6.3</td>
<td>Close-up view of waste emplacement drift</td>
<td>6-4</td>
</tr>
<tr>
<td>Figure 6.4</td>
<td>Principal stress distribution after excavation of the drift</td>
<td>6-5</td>
</tr>
<tr>
<td>Figure 6.5</td>
<td>Displacement vectors after excavation of the drift</td>
<td>6-6</td>
</tr>
<tr>
<td>Figure 6.6</td>
<td>Temperature distribution in the rock at 50 years</td>
<td>6-6</td>
</tr>
<tr>
<td>Figure 6.7</td>
<td>Principal stress distribution in the rock at 50 years</td>
<td>6-7</td>
</tr>
<tr>
<td>Figure 6.8</td>
<td>Shear displacement along the joints at 50 years</td>
<td>6-7</td>
</tr>
<tr>
<td>Figure 7.1</td>
<td>UDEC model for inflow into a tunnel</td>
<td>7-1</td>
</tr>
<tr>
<td>Figure 7.2</td>
<td>Initial total stresses in UDEC model</td>
<td>7-3</td>
</tr>
<tr>
<td>Figure 7.3</td>
<td>Initial pore pressures in UDEC model</td>
<td>7-3</td>
</tr>
<tr>
<td>Figure 7.4</td>
<td>Pore pressures after undrained deformation</td>
<td>7-4</td>
</tr>
<tr>
<td>Figure 7.5</td>
<td>Undrained deformation of the model</td>
<td>7-5</td>
</tr>
<tr>
<td>Figure 7.6</td>
<td>Locations of the history points</td>
<td>7-6</td>
</tr>
<tr>
<td>Figure 7.7</td>
<td>Histories of flow rates during consolidation (SET flow compressible)</td>
<td>7-7</td>
</tr>
<tr>
<td>Figure 7.8</td>
<td>Histories of flow rates during the steady-flow calculation (SET flow steady)</td>
<td>7-7</td>
</tr>
<tr>
<td>Figure 7.9</td>
<td>Histories of flow rates during consolidation (SET flow incompressible)</td>
<td>7-8</td>
</tr>
<tr>
<td>Figure 7.10</td>
<td>Pore pressures at steady-state flow</td>
<td>7-8</td>
</tr>
<tr>
<td>Figure 7.11</td>
<td>Flow rates in the joints at steady-state flow</td>
<td>7-9</td>
</tr>
<tr>
<td>Figure 8.1</td>
<td>Problem geometry for example problem involving flow through jointed rock slope</td>
<td>8-1</td>
</tr>
<tr>
<td>Figure 8.2</td>
<td>Problem geometry for example problem involving flow through jointed rock slope</td>
<td>8-3</td>
</tr>
<tr>
<td>Figure 8.3</td>
<td>Water level at right-hand side equal to 6 m (maximum flow rate = $5.6 \times 10^{-5}$ m$^3$/sec)</td>
<td>8-3</td>
</tr>
</tbody>
</table>

**UDEC Version 5.0**
Example Applications

Figure 8.4  Water level at right-hand side equal to 8 m (maximum flow rate = $7.4 \times 10^{-5}$ m$^3$/sec) .......................................................... 8-4
Figure 8.5  Water level at right-hand side equal to 9 m (maximum flow rate = $8.1 \times 10^{-5}$ m$^3$/sec) .......................................................... 8-4
Figure 8.6  Water level at right-hand side equal to 10 m (maximum flow rate = $9.4 \times 10^{-5}$ m$^3$/sec) .......................................................... 8-5
Figure 8.7  Slope failure with water level at 10 m .................................. 8-5
Figure 9.1  UDEC model of a borehole in a rock mass containing two orthogonal joint sets .............................................................. 9-2
Figure 9.2  Flow in joints at 15 seconds after fluid flow initiates from the borehole .................. 9-3
Figure 9.3  Joint hydraulic apertures at 15 seconds after fluid flow initiates from the borehole .............................................................. 9-3
Figure 9.4  Fluid pressure histories at points 1, 2, 3, 4 and 5 .......................... 9-4
Figure 10.1  UDEC model of deep longwall excavation .............................. 10-1
Figure 10.2  Cap pressure variation with plastic volumetric strain ................. 10-3
Figure 10.3  Close-up view of the zoning for the 2.5-m thick seam and the roof .......... 10-4
Figure 10.4  Second excavation (ZONE model null) step and first fill (ZONE model dy) step ............ 10-5
Figure 10.5  Stress state, 10 m excavated ................................................. 10-6
Figure 10.6  Stress state, 20 m excavated and 10 m backfilled .................. 10-6
Figure 10.7  Stress state, 30 m excavated and 20 m backfilled .................. 10-7
Figure 10.8  Stress state, 40 m excavated and 30 m backfilled ................. 10-7
Figure 10.9  Stress state, 50 m excavated and 40 m backfilled ................. 10-8
Figure 10.10 Stress state, 50 m excavated and 40 m backfilled with Mohr-Coulomb material ....... 10-8
Figure 11.1  UDEC model of a circular excavation in a rock mass containing a high-angle joint set and a vertical fault – tunnel support with a shotcrete lining and cable bolts .............................................................. 11-1
Figure 11.2  y-displacement history recorded at a location on the roof wedge – unstable movement .............................................................. 11-4
Figure 11.3  Close-up view of tunnel showing sliding of roof wedge and failure of shotcrete lining .............................................................. 11-4
Figure 11.4  y-displacement history recorded at a location on the roof wedge – movement stops .............................................................. 11-5
Figure 11.5  Close-up view of tunnel showing the stable roof wedge and axial forces in the cables and shotcrete lining .............................................................. 11-5
Figure 12.1  UDEC model of blocks to be dropped onto the slope ..................... 12-3
Figure 12.2  Position and velocities of the blocks after 25,000 cycles ............... 12-3
Figure 12.3  Position and velocities of the blocks after 50,000 cycles ............... 12-4
Figure 13.1  Geometry of simple slope containing en-echelon joints .................. 13-1
Figure 13.2  En-echelon joints in slope model ......................................... 13-3
Figure 13.3  Continuous joints in slope model ......................................... 13-3
Figure 13.4  Step-path failure of slope – factor-of-safety = 1.29 ....................... 13-5
<table>
<thead>
<tr>
<th>Figure 13.5</th>
<th>Step-path failure depicted by joint maximum shear displacement and rock maximum shear strain</th>
<th>13 - 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 13.6</td>
<td>Continuum failure of slope – factor-of-safety = 1.38</td>
<td>13 - 6</td>
</tr>
<tr>
<td>Figure 13.7</td>
<td>Continuous joint shear failure of slope – factor-of-safety = 1.01</td>
<td>13 - 6</td>
</tr>
<tr>
<td>Figure 14.1</td>
<td>UDEC model – finite difference zones and fluid pressure in the crack</td>
<td>14 - 2</td>
</tr>
<tr>
<td>Figure 14.2</td>
<td>Fracture opening along the fracture for Case 1 – line: analytical solution; crosses: UDEC solution</td>
<td>14 - 3</td>
</tr>
<tr>
<td>Figure 14.3</td>
<td>Displacement vectors at the end of the simulation – Case 1</td>
<td>14 - 4</td>
</tr>
<tr>
<td>Figure 14.4</td>
<td>UDEC finite difference zones in a 2 m × 2 m portion of the model</td>
<td>14 - 5</td>
</tr>
<tr>
<td>Figure 14.5</td>
<td>Fluid pressure at the well (MPa) versus time (sec)</td>
<td>14 - 6</td>
</tr>
<tr>
<td>Figure 14.6</td>
<td>Width at the well (m) versus time (sec)</td>
<td>14 - 7</td>
</tr>
<tr>
<td>Figure 14.7</td>
<td>Fluid pressure (MPa) versus distance (m) along the fracture at 10 sec.</td>
<td>14 - 8</td>
</tr>
<tr>
<td>Figure 14.8</td>
<td>Fracture width (m) versus distance (m) along the fracture at 10 sec.</td>
<td>14 - 9</td>
</tr>
<tr>
<td>Figure 14.9</td>
<td>Displacement vectors near the fracture (half fracture shown) at 10 sec.</td>
<td>14 - 10</td>
</tr>
</tbody>
</table>
## EXAMPLES

<table>
<thead>
<tr>
<th>Example</th>
<th>Description</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>SEISMIC.DAT</td>
<td>1-9</td>
</tr>
<tr>
<td>2.1</td>
<td>STOPE.DAT</td>
<td>2-5</td>
</tr>
<tr>
<td>3.1</td>
<td>TUNNEL.DAT</td>
<td>3-7</td>
</tr>
<tr>
<td>4.1</td>
<td>DAM.DAT</td>
<td>4-11</td>
</tr>
<tr>
<td>5.1</td>
<td>GROUT.DAT</td>
<td>5-8</td>
</tr>
<tr>
<td>6.1</td>
<td>DRIFT.DAT</td>
<td>6-8</td>
</tr>
<tr>
<td>7.1</td>
<td>INFLOW.DAT</td>
<td>7-9</td>
</tr>
<tr>
<td>8.1</td>
<td>SLOPEFLO.DAT</td>
<td>8-6</td>
</tr>
<tr>
<td>9.1</td>
<td>BH.DAT</td>
<td>9-5</td>
</tr>
<tr>
<td>10.1</td>
<td>LONGWALL.DAT</td>
<td>10-9</td>
</tr>
<tr>
<td>11.1</td>
<td>SUPPORT.DAT</td>
<td>11-6</td>
</tr>
<tr>
<td>11.2</td>
<td>SUP_CAB.FIS</td>
<td>11-8</td>
</tr>
<tr>
<td>12.1</td>
<td>BOUNCE.DAT</td>
<td>12-4</td>
</tr>
<tr>
<td>13.1</td>
<td>STEPPATH.DAT</td>
<td>13-7</td>
</tr>
<tr>
<td>14.1</td>
<td>HF_CASE1.DAT</td>
<td>14-12</td>
</tr>
<tr>
<td>14.2</td>
<td>HF_CASE2.DAT</td>
<td>14-15</td>
</tr>
</tbody>
</table>
1 Seismic-Induced Groundfall

1.1 Problem Statement

A demonstration simulation of a seismic-induced groundfall is presented to illustrate the use of UDEC for analyzing this type of problem. The model shown in Figure 1.1 was developed based on the configuration and dimensions of the 34-1-554 over-cut shown on a section drawing for Fraser Mine, Falconbridge Limited, Sudbury, Ontario. A two-dimensional, plane-strain representation was chosen normal to the axis of the over-cut. The over-cut was modeled as being 5 m high and 10 m wide.

![UDEC model for seismic-induced groundfall](image)

**Figure 1.1 UDEC model for seismic-induced groundfall**

It was assumed that two continuous joint sets intersect the plane of analysis: one with an orientation of $45^\circ$, and the other with an orientation of $-9^\circ$. Both sets have a joint spacing of 5 m. For demonstration purposes, a near vertical “artificial” joint was also added to the block in the roof of the excavation to enhance the instability.

From the average laboratory test values provided for the intact rock, several material properties were assumed for the rock blocks:

- density $\quad 3000 \text{ kg/m}^3$
- Young’s modulus $\quad 75,000 \text{ MPa}$
- Poisson’s ratio $\quad 0.18$

_UDEC Version 5.0_
The blocks were assumed to behave elastically only. Coulomb slip behavior was assumed for the joints, and typical textbook values were chosen for joint properties:

- Joint normal stiffness: 20,000 MPa/m
- Joint shear stiffness: 20,000 MPa/m
- Friction angle: 30°
- Cohesion: 0

The in-situ stress state was estimated to be isotropic at 24 MPa (assuming vertical loading due to overlaying rock at a depth of approximately 800 m).

1.2 UDEC Analysis

The UDEC modeling sequence was performed in three stages. First, the model without the over-cut excavation was consolidated under the in-situ stresses. Next, the excavation was introduced and the model cycled to an equilibrium state. The stress distribution around the over-cut at this stage is illustrated in Figure 1.2. The blocks immediately above and below the over-cut have slipped and then stabilized.

In the third stage, two different seismic events with different peak velocities were evaluated. For all seismic simulations, viscous boundaries were introduced around the outer perimeter of the problem domain to eliminate wave reflections, thereby simulating an infinite rock mass. Seismic events were represented by a sinusoidal y-directed stress wave applied at the top of the model. The applied stress wave was superimposed on the existing in-situ stresses.

In the first simulation, a peak stress of 1.25 MPa was applied. Note that, due to the viscous boundary conditions in effect at the top of the model, the “effective” applied stress is 1.25 MPa/2, or 0.625 MPa. The stress distribution in the roof of the excavation after 0.02 second is shown in Figure 1.3. Displacements were monitored at two points. Point 1 is located in the left corner of the excavation; Point 2 is located at the right corner of the roof block. Displacement versus time plots (Figure 1.4) for these points essentially show an elastic response.
**Figure 1.2** Stress distribution around excavation at end of excavation stage

**Figure 1.3** Stress distribution in roof of excavation after 0.02 sec. (applied stress $= 1.25 \times \cos(2\pi \times 100t)$)
It is interesting to compare estimated applied velocities with calculated velocities at the top of the model. The following equation can be used to estimate the applied velocity.

$$V = \frac{\sigma}{2 (\rho C_p)}$$  \hspace{1cm} (1.1)

where $C_p = \left[ \frac{(K + (4/3)G)}{\rho} \right]^{1/2}$.

Using this equation, the applied maximum velocity is found to be approximately 0.04 m/sec. Figure 1.5 shows a peak velocity of approximately 0.06 m/sec. Differences between estimated velocities and measured velocities result from using the intact rock modulus instead of the equivalent deformation modulus, which takes into account the joint deformation.
In the second example, a stress wave with peak stress of 12.5 MPa (“effective” stress = 6.25 MPa) was applied. The stress distribution in the roof of the excavation after 0.02 second is shown in Figure 1.6. This figure shows that the roof block is unstressed, indicating that the block has loosened. Displacement versus time plots (Figure 1.7) also indicate that the block has loosened and is falling. As a matter of interest, the problem geometry and stress distribution at three later times are presented in Figures 1.8 through 1.10.

The predicted velocity (from the equation above) at the top of the problem is 0.4 m/sec. The velocity calculated from the model is shown in Figure 1.11. Again, differences between predicted and measured velocities result from using intact rock modulus instead of rock mass deformation modulus.

Section 1.3 contains a listing of the data file for this model. A movie of the groundfall can be made by including the commands at the end of this file. The movie view is a block plot including velocity vectors and stress tensors. The movie is run with the MOVIE command, which results in a movie file named “SEISMIC.DCX” that can then be run with the movie viewer file, “MOVIE.EXE,” located in the “ITASCA\Shared\Utility” folder.
Figure 1.6  Stress distribution in roof of excavation after 0.02 sec. (applied stress = $12.5 \times \cos(2\pi \ 100t)$)

Figure 1.7  y-displacement histories for two points on excavation boundary (applied stress = $12.5 \times \cos(2\pi \ 100t)$)
Figure 1.8  Stress distribution around excavation after 0.25 sec. (applied stress = $12.5 \times \cos(2\pi \times 100t)$)

Figure 1.9  Stress distribution around excavation after 0.50 sec. (applied stress = $12.5 \times \cos(2\pi \times 100t)$)
Figure 1.10  Stress distribution around excavation after 0.75 sec. (applied stress = 12.5 × cos(2π 100t))

Figure 1.11  Plot of y-velocity at top of model (applied stress = 12.5 × cos(2π 100t))
1.3 Listing of Data File

Example 1.1 SEISMIC.DAT

;File:seismic.dat
;Title:SEISMIC INDUCED ROOF COLLAPSE
new
round 0.01
edge 0.02
; define original boundary of modeled region
; generate joint pattern over entire original region
jset angle 45 spacing 5 origin 0,0
jset angle 351 spacing 5 origin 0,0
; put in joints needed for the later excavation
crack (-5.0,-2.5) (5.0,-2.5)
crack (-5.0,2.5) (5.0,2.5)
crack (-5,-2.5) (-5,2.5)
crack (5,-2.5) (5,2.5)
crack (2.25,2.5) (1.93,5)
; generate fdef zones and assign block and joint properties
gen edge 9.0 range -30,30 -30,30
zone model elastic density 0.003 bulk 3.906E4 shear 3.178E4 range group '&mat1'
group joint 'jmat1'nnjoint model area jks 2E4 jkn 2E4 jfriction 30 range group '&jmat1'
; new contact default
set jcondf joint model area jks=2E4 jkn=2E4 jfriction=30
; apply boundary conditions and initial conditions to
; consolidate model under field stresses
boundary stress -24.0,0.0,0.0,-24.0 xgrad 0.0,0.0,0.0 ygrad -0.03,0.0,-0.03
insitu stress -24.0,0.0,0.0,-24.0 xgrad 0.0,0.0,0.0 ygrad -0.03,0.0,-0.03
boundary yvelocity 0 range -26,26 -21,-19
set gravity=0 -10
; track the x-displacement, and y-displacement over time
history xdisplace 0.0,7.0
history ydisplace 0.0,7.0
solve ratio 1.0E-5
save seismic1.sav
; make excavation
delete range -5.5 -2.5,2.5
solve ratio 1.0E-5
save seismic2.sav

; apply seismic load from top (peak velocity=0.04 m/sec)
; set up nonreflecting boundary
boundary xvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
-26,-23 -21,21
boundary xvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
23,26 -21,21
boundary yvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
-26,26 -21,-19
boundary yvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
-26,26 19,21
; apply sinusoidal stress wave
boundary stress 0.0,0.0,-1.25 yhistory=cosine(100,1.95E-2) range -26,26 &
19,21
reset rot hist time
reset disp
history ncyc 1
history ydisplace -4.56,2.57
history ydisplace 0.0,2.57
history yvelocity 0.0,2.57
history yvelocity 4.0,2.57
history yvelocity -4.48,2.57
history yvelocity 0.0,20.0
history yvelocity 25.0,10.0
history yvelocity 25.0,-10.0
history yvelocity 0.0,-20.0
history yvelocity -25.0,-10.0
history yvelocity -25.0,10.0
history sxx 25.0,10.0
history sxx 25.0,-10.0
history sxx -25.0,-10.0
history sxx -25.0,10.0
history syy 0.0,20.0
damping 0.1 1.0 mass
; 0.02 sec.
cycle time 0.02
save seismic3.sav

restore ‘seismic2.sav’
; apply seismic load from top (peak velocity=0.4 m/sec)
; set up nonreflecting boundary
boundary xvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
-26,-23 -21,21
boundary xvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
23,26 -21,21
boundary yvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range &
-26,26 -21,-19
boundary yvisc ff_bulk=39060.0 ff_shear=31780.0 ff_density=0.0030 range & -26,26 19,21
; apply sinusoidal stress wave
boundary stress 0.0,0.0,-12.5 yhistory=cosine(100,1.95E-2) range -26,26 & 19,21
reset rot hist time
reset disp
history ydisplace -4.56,2.57
history ydisplace 0.0,2.57
history yvelocity 0.0,2.57
history yvelocity 4.0,2.57
history yvelocity -4.48,2.57
history yvelocity 0.0,20.0
history yvelocity 25.0,10.0
history yvelocity 25.0,-10.0
history yvelocity 0.0,-20.0
history yvelocity -25.0,-10.0
history yvelocity -25.0,10.0
history sxx 25.0,10.0
history sxx 25.0,-10.0
history sxx -25.0,-10.0
history sxx -25.0,10.0
history syy 0.0,20.0
damping 0.1 1.0 mass
save seismic4.sav

restore 'seismic4.sav'
; 0.02 sec.
hist ncyc 1
cycle time 0.02
save seismic5.sav

; 0.25 sec.
cycle time 0.23
save seismic6.sav

; 0.50 sec.
cycle time 0.25
save seismic7.sav

; 0.75 sec
cycle time 0.25
save seismic8.sav

; restore 'seismic4.sav'
; make a movie of the groundfall
;wind -12 12 -12 12
;set ovtol 0.05
;plot block vel max 5.0 blue stress max 50
;movie on
;movie file = seismic.dcx
;movie step 1000
;step 40000
2 Open Stoping Using Vertical Retreat

2.1 Problem Statement

A distinct element simulation of a large blasthole, open stoping operation is shown to demonstrate the ability of UDEC to model sequential mining steps. The model is for a quartzite orebody for which the potential instability in the stope back is to be evaluated. Of particular concern is the stress concentration in the crown pillar after mining of the stope is completed.

The geometry for this example is illustrated in Figure 2.1. A steeply dipping orebody (average dip of 80°) is analyzed between the 990 m level and the 1190 m level of the mine. A low-angle discontinuous joint set is also oriented at 10° dip, with average spacing of 30 m. The average thickness of the orebody is 14 m. The upper stope, above the 1090 m level, is mined first; then, mining of the lower level is completed, leaving a 10 m crown pillar.

![Figure 2.1 Initial geometry for blasthole open stoping operation](image)

Four rock types are defined for the analysis: hanging wall quartzite, footwall quartzite, banded ore and a weaker schistose ore. Based on the average laboratory test values, the following properties for these rock types were assumed.
Table 2.1  Properties of rock types

<table>
<thead>
<tr>
<th></th>
<th>Hanging Wall</th>
<th>Banded Ore</th>
<th>Schistose Ore</th>
<th>Footwall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus (GPa)</td>
<td>62</td>
<td>56</td>
<td>40</td>
<td>67</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.29</td>
<td>0.28</td>
<td>0.33</td>
<td>0.28</td>
</tr>
<tr>
<td>Unconfined compressive</td>
<td>186</td>
<td>168</td>
<td>96</td>
<td>198</td>
</tr>
<tr>
<td>strength (MPa)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Several joint properties were estimated:

- joint normal stiffness 5 GPa/m
- joint shear stiffness 5 GPa/m
- joint friction angle 27°
- joint cohesion 0

The pre-mining state of stress was estimated to be hydrostatic. The stress is 33 MPa at the 1190 m level.
2.2 UDEC Analysis

The UDEC model was created with constitutive models and properties assigned to the blocks using the **ZONE** command. This facilitates the assignment of the different rock types to the model. **Figure 2.2** shows the different cohesion properties that were assigned to the hanging wall, footwall, banded ore and schistose ore.

![Figure 2.2 Cohesion properties assigned to the UDEC model](image)

The model was first consolidated at the initial stress state by applying a stress boundary condition. This boundary condition was then replaced with a boundary-element boundary to represent an infinite elastic medium in the far field.

After model consolidation, the mining progressed in five stages. First, the upper level blocks were removed for a stope height of 45 m. The lower stope was then mined in four stages of 17 m, 15 m, 15 m and 18 m, leaving the 10 m crown pillar. The final stress concentration is depicted in **Figure 2.3**. At this stage, most of the stress is transferred to the abutments. The stress build-up in the crown pillar is shown in the stress history plot in **Figure 2.4**.

Although backfilling was not simulated in this example, the model can simulate backfill emplacement after excavation, and stress histories in the backfill can be monitored like they were for the crown pillar (see **Figure 2.4**).
Figure 2.3  Principal stresses at end of mining sequence

Figure 2.4  Changes in xx- and yy-stresses in crown pillar with mining stages

History 2: xx-stress, History 3: yy-stress (note: compressive stresses are negative)
2.3 Listing of Data File

Example 2.1 STOPE.DAT

;File:STOPE.dat
;Title:OPEN STOPING USING VERTICAL RETREAT
new
config

;establish initial geometry
round 0.2
edge 0.4
block 0,0 0,200 150,200 150,0

jset angle 80 trace 300 spacing 14 origin 0,0
delete area 1210.0
crack (67.864,0) (104.129,200)

;put in discontinuous cross joints
jset angle 350 trace 14 gap 14 spacing 30 origin 0,0
jset angle 350 trace 14 gap 14 spacing 30 origin 49.48,200

;put in joints needed for later excavations
crack (74,100) (89,100)
crack (84,155) (99,155)
crack (63,35) (78,35)
crack (65,52) (81,52)
crack (68,67) (83,67)
crack (71,82) (86,82)

; create finite difference triangles in all blocks
gen quad 100.0
gen edge 100.0

;assign material properties
;hanging wall rock
group zone ‘hangwall’
;banded ore
group zone ‘bandedore’ range region (56.86,0) (92.13,200) &
(104.13,200) (67.86,0)
;schistose ore
group zone ‘schisore’ range region (67.86,0) (104.13,200) &
(106.3,200) (71.05,0)
;foot wall
group zone ‘footwall’ range region (71.05,0) (106.3,200) (160,200) &
(160,0)
zone model mohr density 0.002 bulk 4.8E4 shear 2.4E4 cohesion 93 range &
group ‘hangwall’
zone model mohr density 0.002 bulk 4.2E4 shear 2.2E4 cohesion 84 range &
group ‘bandedore’
zone model mohr density 0.002 bulk 3.8E4 shear 1.5E4 cohesion 48 range &
group ‘schisore’
zone model mohr density 0.002 bulk 5.1E4 shear 2.6E4 cohesion 99 range &
group ‘footwall’

; joint
group joint ‘jmat1’
joint model area jks 4E3 jkn 5E3 jfriction 27 range group ‘jmat1’
; new contact default
set jcondf joint model area jks=4000 jkn=5000 jfriction=27

; specify stress field
boundary stress -33.0,0.0,-33.0 xgrad 0.0,0.0,0.0 ygrad 0.02,0.0,0.02
insitu stress -33.0,0.0,-33.0 xgrad 0.0,0.0,0.0 ygrad 0.02,0.0,0.02 szz &
-33.0 zgrad 0.0,0.0,0.02
set gravity=0 -10
history solve_ratio
history sxx 80.0,105.0
history syy 80.0,105.0
solve ratio 1.0E-5
save st1.sav

be gen -1,160 -1,201
be fix -100,-100 260,300
be ff_dens=0.002 ff_bulk=44000 ff_shear=26000
be stiff

; begin excavation sequence with upper stope
delete range 78,98 110,155

solve ratio 1.0E-5
save st2.sav

; excavate lowest part of lower stope
delete range 63,81 35,52

solve ratio 1.0E-5
save st3.sav

; excavate next part of lower stope
delete range 65,83 52,67

UDEC Version 5.0
solve ratio 1.0E-5
save st4.sav

; excavate next part of lower stope
delete range 67,86 67,82
solve ratio 1.0E-5
save st5.sav

; excavate last part of lower stope
delete range 71,89 82,100
solve ratio 1.0E-5
save st6.sav
3 Tunnel Support Loading

3.1 Problem Statement

This simulation demonstrates the application of UDEC to examine lined tunnels, with specific emphasis on loads developed in the concrete liners. This example also illustrates the procedure to model the individual stages of a sequential construction operation.

The idealized geometry of the tunnel system is shown in Figure 3.1. The system consists of two tunnels on 12-meter centers at a depth of roughly 70 meters (centerline) beneath the sea bed. The water level is initially 110 meters above the tunnel centerline. The small (service) tunnel is 5.24 meters in diameter, with a 37 cm-thick concrete liner. The main tunnel is 8.22 meters in diameter, with a 46 cm-thick concrete liner. The service tunnel is driven and lined prior to excavation and lining of the main tunnel. After installation of the main tunnel lining, the water level is raised an additional 100 meters.

![Figure 3.1 Idealized geometry of service tunnel and main tunnel](image)

The sequence of construction activities is

1. excavation of the service tunnel;
2. lining of the service tunnel;
3. excavation of the main tunnel;
4. lining of the main tunnel; and
5. raising of the water level.

The objective of the analysis is to evaluate the tunnel support for the service tunnel and the main tunnel at each construction stage.
The material properties for this example are listed below.

**Rock** – The tunnels are excavated in rock that has the following material properties.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic modulus</td>
<td>0.89 GPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.35</td>
</tr>
<tr>
<td>Uniaxial compressive strength</td>
<td>3.5 MPa</td>
</tr>
<tr>
<td>Cohesion</td>
<td>1 MPa</td>
</tr>
<tr>
<td>Density</td>
<td>1340 kg/m³</td>
</tr>
</tbody>
</table>

**Concrete Liner** – The elastic modulus for the concrete liner is 24 GPa, and the Poisson’s ratio is 0.19. The liner is assumed to behave as a linear-elastic material.

### 3.2 UDEC Analysis

The **UDEC** model created for this analysis is shown in Figure 3.2. The dimensions are selected so that the centerline of the tunnels corresponds to the location $y = -70$ m. Note that the model boundaries are very close to the tunnel excavations. This model is only intended to provide fast calculations for demonstration purposes. A larger model would be required for practical solutions.

Construction joints are used to create the tunnel boundaries by adding the `join` keyword to the `CRACK` and `ARC` commands. In this way, it is not necessary to assign joint models and properties for this example.

The lower and side boundaries of the **UDEC** model are fixed with rollers. The weight of the sea water above the sea bed is simulated by applying the equivalent pressure of 30 m of water head to the top surface of the model. The tunnels are assumed to be lined with a waterproof liner. Thus, there is no need to perform a transient groundwater flow analysis. The pore water pressure is accounted for by setting the unit weight of the rock to the submerged unit weight. The vertical-to-horizontal stress ratio is assumed to be hydrostatic.

For this example, the five construction activities listed above are simulated by three modeling stages. Excavation of the main tunnel and lining of the service tunnel are modeled as one activity that occurs instantaneously. Lining of the main tunnel and raising the water level are also assumed to occur instantaneously. For a more realistic simulation, these activities should be simulated separately by reducing the tractions around the tunnels gradually and installing the support after some prescribed relaxation takes place.

In the first modeling stage, after gravity stresses have been initialized in the body, the service tunnel is mined, and **UDEC** is cycled until equilibrium is achieved. The resulting elastic displacements are given in Figure 3.3.
Figure 3.2  UDEC model zoning with service tunnel excavated

Figure 3.3  Elastic displacements due to excavation of service tunnel
In the second stage, the service tunnel is lined and the main tunnel is excavated. Sixteen beam elements are used to model the concrete liner for the service tunnel. Figures 3.4 and 3.5 illustrate the displacements and principal stress distribution resulting from excavation of the main tunnel. Note that the entire service tunnel translates toward the main tunnel.

In the third stage, an additional load is applied to the top of the model to simulate the weight of an additional 100 meters of water. The STRUCT apply pressure command is used to apply hydrostatic loads (to the tunnel liner) representing the water table 210 m above the tunnel center line. Figures 3.6 and 3.7 display the thrust and moment distributions, respectively, that develop after the additional load is applied. The maximum thrust in the service tunnel is approximately $5.5 \times 10^6$ N, and that in the main tunnel is approximately $9.0 \times 10^6$ N.
**Figure 3.4** Displacements after mining of main tunnel

**Figure 3.5** Principal stress distribution after mining of main tunnel
Figure 3.6  Thrust in liners after water level is raised

Figure 3.7  Moment in liners after water level is raised
3.3 Listing of Data File

Example 3.1 TUNNEL.DAT

;File:tunnel.dat
config
round 0.1
edge 0.2
block 0,-90 0,-30 69,-30 69,-90
crack (0,-70) (60,-70) join
crack (30,0) (30,-90) join
crack (42,0) (42,-90) join
arc (30,-70) (34.11,-70) 360 12 join
arc (42,-70) (44.62,-70) 360 8 join
arc (30,-70) (35.5,-70) 360 12 join
arc (42,-70) (47.5,-70) 360 12 join
gen edge 2.0
; rock properties
group zone 'rock'
zone model mohr density 1.34E3 bulk 9.9E8 shear 3.3E8 friction 30 cohes &
1E6 range group 'rock'
; initial stress state
boundary stress 102000.0,0.0,102000.0 xgrad 0.0,0.0,0.0 ygrad &
13400.0,0.0,13400.0
insitu stress 102000.0,0.0,102000.0 xgrad 0.0,0.0,0.0 ygrad &
13400.0,0.0,13400.0 szz 102000.0 zgrad 0.0,13400.0
set gravity=0.0 -10.0
solve ratio 1.0E-5 elastic
save tun1.sav
;
; excavate service tunnel
delete range annulus (42,-70) 0 2.62
boundary xvelocity 0 range -1,1 -91,0
boundary xvelocity 0 range 68,70 -91,0
boundary yvelocity 0 range -1,90 -91,-89
history ydisplace 42.0,-67.0
history ydisplace 42.0,-73.0
history xdisplace 39.0,-70.0
history xdisplace 45.0,-70.0
reset jdisp disp
solve ratio 1.0E-5
save tun2.sav
;
; line service tunnel and excavate main tunnel
struct gen begin 39.5747,-69.5376 end 39.5747,-69.5376 max=100.0 min=0.2 &
mat 1
property mat 1 st_density 2.4E3 st_prat 0.2 st_ycomp 1E10 st_yield 1E10 & st_ymod 2.4E10 st_area 0.37 st_inertia 4.221E-3 st_shape 0.8333 st_spac & 1 st_thickness 0.37 st_width 1 if_cohesion 1E10 if_kn 1E8 if_ks 1E7 delete range annulus (30,-70) 0 4.11
reset disp
solve ratio 1.0E-5
save tun3.sav
;
; line main tunnel and raise water level
struct gen begin 26.3077,-68.4617 end 26.3077,-68.4617 max=1.0E9 min=0.2 & mat 2
property mat 2 st_density 2.43E3 st_prat 0.2 st_ycomp 1E10 st_yield 1E10 & st_ymod 2.4E10 st_area 0.46 st_inertia 8.111E-3 st_shape 0.8333 st_spac & 1 st_thickness 0.46 if_cohesion 1E10 if_kn 1E8 if_ks 1E7
boundary stress 0.0,0.0,-1000000.0 range -1,91 -31,-29
struct apply pressure 0.0 2060000.0
reset disp
solve ratio 1.0E-5
save tun4.sav
4 Gravity Dam: Fluid Flow and Dynamic Loading

4.1 Problem Statement

This demonstration problem involves analysis of a 100 m-high concrete gravity dam on a jointed rock foundation. The average joint spacing is 50 m, with joints oriented at $20^\circ$ and $-70^\circ$. Two loading conditions are studied. First, the effects of filling the reservoir are studied, including an analysis of fluid flow through the rock joints. Second, a dynamic wave is applied to the base of the model to study potential effects of an earthquake-type loading.

4.2 UDEC Analysis

The UDEC model for this problem is illustrated in Figure 4.1. The model is an idealized representation of a gravity dam on a jointed rock foundation, and is intended to demonstrate the recommended solution procedure for this type of problem. The data file is listed in Section 4.3. The analysis is performed in the following sequence.

*Figure 4.1 UDEC model of gravity dam with principal stresses plotted at Stage 1*
Stage 1: Gravity Loads – Empty Reservoir

An in-situ state of stress with an effective stress ratio $\sigma_H/\sigma_V = 0.69$ is assumed in the rock mass. The water table is assumed at $y = 0$. The initial stress state resulting from the weight of the dam, and with the reservoir empty, is shown in Figure 4.1. Note that stresses specified with the INSITU command are total stresses. These are assigned to the blocks. For the joints, UDEC calculates effective stresses, and sets the domain pressures to the hydrostatic stresses.

Stage 2: Full Reservoir

During this stage, the water table is assumed to be at the top of the dam, exerting hydrostatic pressure on the upstream side of the dam and rock foundation. The horizontal reaction due to the load applied on the dam is taken by the rollers on the lateral boundaries.

Several conditions are assumed with respect to fluid flow:

1. Joint contacts along the bottom and sides of the model are assumed to have zero permeability.

2. On the rock surface upstream of the dam, the head is fixed at 100 m (0.98 MPa) by using the BOUNDARY pp command. Downstream, the head is set to zero.

3. The interface between the dam and rock foundation is assumed to have low permeability.

4. The algorithm for steady flow (SET flow steady) is used.

Selected results for Stage 2 are shown in Figures 4.2 through 4.6. The displacements resulting from reservoir filling are shown in Figure 4.2, and the histories of the $x$- and $y$-displacements at the crest of the dam are shown in Figure 4.3. The latter figure indicates that the model is in equilibrium with the reservoir full. The plot of flow rates in Figure 4.4 shows that most of the flow is concentrated in the joint directly beneath the dam foundation. Figure 4.5 shows the shear and normal displacements along this joint (at $x = -33.42, y = -30.37$). The positive normal displacement indicates that the joint opens during this stage. The fluid pressure history at location $x = -22.1, y = -26.3$ along this joint is plotted in Figure 4.6.
**Figure 4.2**  Principal stress state and displacements at Stage 2

**Figure 4.3**  x- and y-components of displacement at crest of dam during Stage 2 reservoir filling
**Figure 4.4** Flow rates at Stage 2

**Figure 4.5** Shear and normal displacements along joint at $x = -33.42$, $y = -30.37$
Stage 3: Dynamic Loading

In this stage, a vertical, propagating, sinusoidal shear wave (frequency = 5 Hz) is applied to the base of the model for 10 seconds. The following boundary conditions are used.

1. The bottom boundary is assumed to be a non-reflecting boundary in the shear direction, and is fixed in the vertical direction. The dynamic input is applied in the form of a shear stress history.

2. A free-field boundary condition is applied to the sides of the model (using `BOUNDARY ffield`). The free field is automatically prescribed the same material behavior as the adjacent blocks along the side boundaries.

3. The static reactions resulting from the flow stage are still applied to the blocks during the dynamic stage, so that the blocks will be in equilibrium in the absence of dynamic loads.

Rayleigh damping is applied to simulate the hysteretic response of the jointed rock foundation. A damping value of 5% of critical damping at a center frequency of 5 Hz is assumed. A shear modulus reduction factor was not applied for this simple demonstration example.

The fluid flow calculation is switched off during this stage (`SET flow off`). This is analogous to assuming that no flow occurs during the 10 second dynamic loading.

*Figures 4.7 through 4.10 show the results of the Stage 3 analysis after 1.5 seconds of the dynamic loading. The \( x \)-velocity histories at the dam crest and model base are plotted in Figure 4.7, and illustrate the amplification of the wave at the surface. The displacement histories at the dam*
crest (given in Figure 4.8) show an accumulation of horizontal displacement with each load cycle. The joint shear and normal displacements at the joint beneath the dam (in Figure 4.9) show an accumulation of shear displacement. In Figure 4.10, note the large amount of slip that occurs along the first joint beneath the dam foundation; the joint upstream at 70° is open (i.e., no effective stress).

The dynamic phase is repeated to evaluate the influence of hydrodynamic pressure on the response of the dam. Hydrodynamic pressure is simulated by using the Westergaard procedure described in Section 4.3.1.6 in Special Features. The FISH function wester is called to apply the hydrodynamic pressure along the upstream face of the dam. Figures 4.11 through 4.14 show the results after 1.5 seconds when hydrodynamic pressures are included.

A substantial increased in displacement is shown. The large displacement of the wedge beneath the dam, along with the accumulation of displacements shown in Figures 4.12 and 4.14, indicate the probable failure of the dam.
Figure 4.7  x-velocity histories at dam crest and model base at Stage 3

Figure 4.8  x- and y-displacements at dam crest at Stage 3
**Example Applications**

**Figure 4.9** Joint shear and normal displacement histories at joint beneath dam foundation at Stage 3

**Figure 4.10** Displacements, principal stresses and joint shear displacement at Stage 3
Figure 4.11  x-velocity histories at dam crest and model base at Stage 3

Figure 4.12  x- and y-displacements at dam crest at Stage 3
**Figure 4.13** Joint shear and normal displacement histories at joint beneath dam foundation at Stage 3

**Figure 4.14** Displacements, principal stresses and joint shear displacement at Stage 3
4.3 Listing of Data File

Example 4.1 DAM.DAT

;Project Record Tree export
;File: dam.dat
;Title: Gravity Dam
;
; --- dam --- discontinuous joints : 20 and -70 deg.
; --- insitu stresses (k=0.5)
; --- free-field (20 nodes) --- applied only in phase 3
;
; --- phase 1 --- gravity loading; x-fixed boundaries
;
config fluid
set fluid clear steady off
round 0.5
edge 1
block -200,-200 -200,100 200,100 200,-200
crack (-200,0) (200,0)
crack (-40,0) (-40,100)
crack (40,0) (-40,100)
delete range -200,-40 0,100
delete range 40,200 0,100
jregion id 1 -200.0,-200.0 -200.0,0.0 200.0,0.0 200.0,-200.0
jset angle 20 spacing 50 origin 50,0 range jregion 1
jset angle 290 trace 50 gap 50 spacing 50 origin -50.99,16.45 &
    range jregion 1
jset angle 290 trace 50 gap 50 spacing 50 origin -10.4,-21.98 &
    range jregion 1
gen edge 60.0 range -200,200 -200,0
gen edge 30.0 range -40,40 0,100
;
; block and joint properties
group zone 'dam:rock block' range -200,200 -200,0
group zone 'dam:dam block' range -40,40 0,100
zone model elastic density 2.65E-3 bulk 3.333E4 shear 2E4 range group &
    'dam:rock block'
zone model elastic density 2.4E-3 bulk 1.667E4 shear 1.25E4 range group &
    'dam:dam block'
group joint 'dam:rock joint' range -200,200 -200,1
group joint 'dam:cohesive joint' range -200,200 -200,-150
group joint 'dam:foundation joint' range -40,40 -1,1
group joint 'dam:rock joint - zero perm' range -201,-199 -150,1
group joint 'dam:rock joint - zero perm' range 199,201 -150,1
group joint 'dam:cohesive joint - zero perm' range -201,-199 -200,-150

UDEC Version 5.0
Example Applications

group joint 'dam:cohesive joint - zero perm' range 199,201 -200,-150
group joint 'dam:cohesive joint - zero perm' range -200,200 -201,-199
joint model area jks 1E3 jkn 1E3 jfriction 30 jperm 3E8 ares 5E-4 &
  azero 1E-3 range group 'dam:rock joint'
joint model area jks 1E3 jkn 1E3 jfriction 30 jcohesion 2 jtension 2 &
  jperm 3E8 ares 1E-4 azero 2E-4 range group 'dam:foundation joint'
joint model area jks 1E3 jkn 1E3 jfriction 30 jcohesion 2 jperm 3E8 &
  ares 5E-4 azero 1E-3 range group 'dam:cohesive joint'
joint model area jks 1E3 jkn 1E3 jfriction 30 jperm 0 ares 5E-4 &
  azero 1E-3 range group 'dam:rock joint - zero perm'
joint model area jks 1E3 jkn 1E3 jfriction 30 jcohesion 2 jperm 0 &
  ares 5E-4 azero 1E-3 range group 'dam:cohesive joint - zero perm'
fluid density=0.0010
;
; boundary conditions : lateral : x-fixed ; bottom : y-fixed
boundary xvelocity 0 range -201,-199 -201,1
boundary xvelocity 0 range 199,201 -201,1
boundary yvelocity 0 range -201,201 -201,-199
;
; set gravity ; set in-situ stresses (total)
set gravity=0.0 -9.81
insitu stress 0.0,0.0,0.0 xgrad 0.0,0.0,0.0 ygrad 0.017885,0.0,0.02597 &
  szz 0.0 zgrad 0.0,0.017885 ywtable 0.0 range -200,200 -200,0
history xdisplace -40.0,100.0
history ydisplace -40.0,100.0
solve ratio 1.0E-5 elastic
save dam1.sav
;
; --- phase 2 --- water loads and flow
;
boundary pp 0.981 range -201,-39 -1,1
boundary stress 0.0,0.0,-0.981 range -201,-39 -1,1
boundary stress -0.981,0.0,0.0 xgrad 0.0,0.0,0.0 ygrad 0.00981,0.0,0.0 &
  range -40.1,-39 -0.1,100.1
boundary xvelocity 0 range -201,-199 -1,1
reset jndisp hist time disp
set ftime=0
history xdisplace -40.0,100.0
history ydisplace -40.0,100.0
history sdisplace -33.42,-30.37
history ndisplace -33.42,-30.37
history pp -22.1,-26.3
history flowrate -33.42,-30.37
history flowtime
set flow=on

UDEC Version 5.0
solve ratio 1.0E-5
save dam2.sav

; --- phase 3 --- dynamic loading: shear wave at base of model
; free-field lateral boundaries
; with constant domain pressures; no fluid flow by setting flow off
;
set flow=off
damping 0.05 5.0
boundary ffield

boundary xvisc ff_bulk=33333.0 ff_shear=20000.0 ff_density=0.00265 range &
-201,201 -201,-199
boundary stress 0.0,0.4,0.0 history=sine(5,10) range -201,201 -201,-199
boundary yvelocity 0 range -201,201 -201,-199
reset history time disp vel
history xvelocity -40.0,100.0
history xvelocity 0.0,-200.0
history yvelocity 0.0 100.0
history yvelocity -40.0,100.0
history xdisplace -40.0,100.0
history ydisplace -40.0,100.0
history ndisplace -33.42,-30.37
history sdisplace -33.42,-30.37
history sstress -33.42,-30.37
history ffxvel 0 1
history ffxvel 0 2
history xdisplace -31.765602,-29.082699
history xdisplace -41.061577,100.286316
history xdisplace -39.51225,75.49704
history xdisplace -41.061577,52.25709
history xdisplace -38.737583,23.594501
history xdisplace -38.737583,1.9038887
save dam3.sav
cycle time 1.5
save dam4.sav

; repeat dynamic analysis including hydrodynamic pressure
; using wester.fis FISH function
;
restore 'dam2.sav'
set flow=off
damping 0.05 5.0
boundary ffield
call 'wester.fis'
set dx=-1 dy=0 height=100 yb=0 c_m=0.743 den_w=0.001
westergaard
boundary xvisc ff_bulk=33333.0 ff_shear=20000.0 ff_density=0.00265 range &
-201,201 -201,-199
boundary stress 0.0,0.4,0.0 history=sine(5,10) range -201,201 -201,-199
boundary yvelocity 0 range -201,201 -201,-199
reset history time disp vel
history xvelocity -40.0,100.0
history xvelocity 0.0,-200.0
history xvelocity 0.0 100.0
history yvelocity -40.0,100.0
history ydisplace -40.0,100.0
history ndisplace -33.42,-30.37
history sdisplace -33.42,-30.37
history sstress -33.42,-30.37
history ffxvel 0 1
history ffxvel 0 2
history xdisplace -31.765602,-29.082699
history xdisplace -41.061577,100.286316
history xdisplace -39.51225,75.49704
history xdisplace -41.061577,52.25709
history xdisplace -38.737583,23.594501
history xdisplace -38.737583,1.9038887
save dam5.sav
cycle time 1.5
save dam6.sav
5 Cement Grouting Simulation

5.1 Problem Statement

The Bingham fluid model is widely accepted as an appropriate model for cement-based grouts (see, for example, Littlejohn 1982, Hassler et al. 1987 and Lombardi 1985). This simulation demonstrates use of the Bingham fluid model in UDEC.

The problem geometry represents a horizontal section in a regularly jointed rock mass in which a cylindrical hole (1.2 m diameter) has been made. The rock mass is assumed to be subject to an initial biaxial in-situ stress ($\sigma_{xx} = 0.2$ MPa and $\sigma_{yy} = 0.1$ MPa). Grout injection is simulated by maintaining specified pressures within the hole. The pressure is increased in 2000 Pa increments, and flow conditions are examined at each stage. The hypothetical properties used for the rock, discontinuities and grout are shown below.

<table>
<thead>
<tr>
<th>Intact Rock</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>bulk modulus</td>
<td>10 GPa</td>
</tr>
<tr>
<td>shear modulus</td>
<td>3 GPa</td>
</tr>
<tr>
<td>density</td>
<td>3000 kg/m$^3$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Joints</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>normal stiffness</td>
<td>10 GPa/m</td>
</tr>
<tr>
<td>shear stiffness</td>
<td>10 GPa/m</td>
</tr>
<tr>
<td>friction angle</td>
<td>45°</td>
</tr>
<tr>
<td>joint permeability constant</td>
<td>$1 \times 10^8$ Pa$^{-1}$ sec$^{-1}$</td>
</tr>
<tr>
<td>aperture at zero normal stress</td>
<td>0.1 mm</td>
</tr>
<tr>
<td>residual aperture at high stress</td>
<td>0.05 mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Grout</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>cohesion</td>
<td>0.1 Pa</td>
</tr>
<tr>
<td>density</td>
<td>1000 kg/m$^3$</td>
</tr>
</tbody>
</table>
5.2 UDEC Analysis

The UDEC model and initial stress state are illustrated in Figure 5.1. The threshold pressure gradient for flow is defined for the Bingham fluid model by specifying a yield strength limit with the FLUID cohw command.

In command-line mode, the fluid pressure can be applied to the hole in 2000 Pa increments with a FISH function, press_hole. See the data file for this problem in Section 5.4.

![Figure 5.1 UDEC model and initial stress state for grouting simulation](image)

No steady state flow occurs until the pressure in the hole exceeds 8000 Pa. This is indicated in the plot of flow-rate histories at different contact locations versus hole pressure in Figure 5.2.

The flow plots for hole pressures of 10,000 Pa, 12,000 Pa and 16,000 Pa are shown in Figures 5.3, 5.4 and 5.5. The corresponding changes in grout pressure in joints are shown in Figures 5.6 through 5.8.
5.3 References


**Figure 5.2**  Flow rates versus hole pressure

**Figure 5.3**  Flow rates for grouting simulation at 10,000 Pa
Figure 5.4  Flow rates for grouting simulation at 12,000 Pa

Figure 5.5  Flow rates for grouting simulation at 16,000 Pa
Figure 5.6  Grout pressures in joints for grouting simulation at 10,000 Pa

Figure 5.7  Grout pressures in joints for grouting simulation at 12,000 Pa
Figure 5.8  Grout pressures in joints for grouting simulation at 16,000 Pa
5.4 Listing of Data File

**Example 5.1 GROUT.DAT**

```plaintext
; ; grouting simulation with ; bingham fluid flow logic ;
; File:GROUT.dat
config fluid
round 0.01
;edge 0.02
block 0,0 10,10 10,0
;crack (0,5) (10,5)
crack (5,0) (5,10)
arc (5,5) (5.6,5) 360 12
jset angle 0 trace 10 spacing 2 origin 0,0
jset angle 90 trace 10 spacing 2 origin 0,0
gen edge 1.25
;
; material properties
; intact rock
group zone 'User:mat1'
zone model elastic density 3E3 bulk 1E10 shear 3E9 range group 'User:mat1'
; discontinuities
group joint 'User:jmat1'
joint model area jks 1E10 jkn 1E10 jfriction 45 jperm 1E8 ares 0.00005 &
zejero 0.0001 range group 'User:jmat1'
; new contact default
set jcondf joint model area jks=1E10 jkn=1E10 jfriction=45 jperm=1E8 &
ares=0.00005 azero=0.0001
; insitu stress and boundary conditions
insitu stress -200000.0,0.0,-100000.0 nodis
boundary stress -200000.0,0.0,-100000.0

; conditions during solution
; create hole
delete range 4.5,5.5 4.5,5.5
; cycle to equilibrium
history unbalanced
solve ratio 1.0E-5
save gr0.sav

; use steady state fluid logic
set flow steady
```

UDEC Version 5.0
reset hist
hist n=1000
history pp 5.0,5.0
history flowrate 9.0,5.0
history flowrate 5.0,9.0

; grout properties (density - cohesion)
fluid density=1000.0
fluid cohw=0.1

; pressurize hole in 0.2e4 increments

; use the following fish function only in command line mode
; Name: press_hole
; def press_hole
; loop i (1,8)
; nam_t = 'gr' + string(i) + '.sav'
; p_hole = float(i) * 0.2e4
; command
; pfix pres p_hole range 4.9, 5.1 4.9, 5.1
; cycle 1000
; save nam_t
; endcommand
; endloop
; end
; press_hole

pfix pres 2e3 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro1.sav

pfix pres 4e3 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro2.sav

pfix pres 6e3 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro3.sav

pfix pres 8e3 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro4.sav

UDEC Version 5.0
pfix pres 1e4 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro5.sav

pfix pres 1.2e4 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro6.sav

pfix pres 1.4e4 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro7.sav

pfix pres 1.6e4 range 4.9, 5.1 4.9, 5.1
cy 1000
save gro8.sav
6 Thermomechanical Analysis of a Waste Emplacement Drift

6.1 Problem Statement

This problem involves the transient thermal-mechanical simulation of the behavior of a waste emplacement drift in which heat-producing waste is placed vertically beneath the floor. The specific problem presented here is adapted from Christianson (1989)*.

The emplacement drift under study is in the center of an emplacement panel. Spent fuel (SF) canisters and defense high-level waste (DHLW) canisters are alternately placed in the floor of the drift at a pitch of 2.3 m. The emplacement of waste in the panel is assumed to be instantaneous.

Figure 6.1 illustrates the conceptual representation of the vertical waste emplacement. Because of symmetry, only one-half of the disposal room and pillar needs to be included in the analysis. The thermal boundary conditions are adiabatic. The top and bottom horizontal boundaries are moved sufficiently far from the heat-generating waste to remain at the initial temperature of 26°C for the time period simulated.

* This section was prepared for the U.S. Nuclear Regulatory Commission under U.S. NRC Contract No. 02-85-002.
Using two-dimensional models requires that the discrete location of the waste containers be distributed uniformly along the disposal room. In the case of vertical emplacement, this means the location of a vertical heat-generating trench at the center of the floor along the axis of the room. Because of the transient nature of the problem, as well as the geometric layout of the waste, the “trench” concept is expected to be an adequate idealization of the emplacement.

The tributary heating area for the emplacement panel is reported by Christianson (1989) to be 8194.5 m². The average thermal loading is considered to be 14.1 W/m². For the panel geometry, this results in an initial heat-generating power per meter of room length of 713.5 W.

The initial power of an SF container at the time of emplacement is set to 3.2 kW. The initial power of the DHLW container is chosen as 0.42 kW. The power output of the two waste types is combined and treated as spent fuel, as given by Peters (1983):

\[
P(t) = 0.54 \exp(-\ln(0.5)t/89.3) + 0.44 \exp(-\ln(0.5)t/12.8) \tag{6.1}
\]

where \( P(t) \) = normalized power, and \( t \) = time in years.

Eq. (6.1) is similar to the expression for normalized power as a function of time given by Mansure (1985) for SF.

The thermal and mechanical material properties used in this example are shown in Table 6.1:

<table>
<thead>
<tr>
<th>Property</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Intact Rock</strong></td>
<td></td>
</tr>
<tr>
<td>Bulk density</td>
<td>2.34 g/cc</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>15.1 GPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.20</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>2.07 W/m °C</td>
</tr>
<tr>
<td>Specific heat</td>
<td>961 J/kg °C</td>
</tr>
<tr>
<td>Thermal expansion coefficient</td>
<td>( 10.7 \times 10^{-6} ) °C</td>
</tr>
<tr>
<td><strong>Joints</strong></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness</td>
<td>100 GPa/m</td>
</tr>
<tr>
<td>Shear stiffness</td>
<td>100 GPa/m</td>
</tr>
<tr>
<td>Cohesion</td>
<td>1.0 MPa</td>
</tr>
<tr>
<td>Friction</td>
<td>38.7°</td>
</tr>
<tr>
<td>Dilation</td>
<td>0.0°</td>
</tr>
</tbody>
</table>

**Table 6.1** Thermal and mechanical properties used in thermomechanical analysis of a waste emplacement drift

_UDEC Version 5.0_
6.2 UDEC Analysis

In UDEC, each joint is explicitly modeled with variable spacing and persistence. The blocks are assumed to behave elastically. This means that inelastic behavior is allowed to occur only along the joints. Figures 6.2 and 6.3 illustrate the pattern of joints represented in this example.

The analysis ignores any effects of the jointing on the thermal conductivity of the rock mass. Based on the results of field tests involving thermal conductivity of rock masses, this assumption appears reasonable. The analysis also ignores the effects of fluid (i.e., air and water) convection in the rock mass and emplacement room, and the analysis ignores effects of boiling of pore water, which could affect heat transfer rates. The thermal properties assume fully saturated conditions.

A linear-stiffness Coulomb joint model is used in this analysis. While more complex models (such as the continuously yielding model or the Barton-Bandis model) could be used, these models vary in detail of the behavior, but the fundamental effects are similar.

The kinematic boundary conditions used in the UDEC model are shown in Figure 6.1, and are such that the two vertical boundaries are restricted from moving in the horizontal direction, but are free to move in the vertical direction. The lower horizontal boundary is restricted from moving in the vertical direction, but free to move in the horizontal direction. The upper horizontal boundary is a free-to-move pressure boundary. The initial vertical and horizontal stresses applied to the models are $-7 \text{ MPa}$ and $-3.5 \text{ MPa}$.

The decaying heat source is applied as a heat flux boundary condition (THAPP flux) to a 4 m length of the boundary beneath the center of the floor. The initial power is divided by the 4 m length, and then split according to the constants given in Eq. (6.1). Note that only half of this power is applied, because of symmetry.

The modeling sequence is as follows.

Excavation of the Drift

Deformations and stresses throughout the jointed rock are determined after the drift is excavated.

Thermomechanical Response at 50 Years

The thermal-mechanical process is performed by alternately calculating the thermal loading and then the mechanical response. The thermal calculation increment is made in 1000 steps with a thermal timestep of $8 \times 10^4 \text{ sec}$, using the implicit solution algorithm. Then, a maximum of 2000 mechanical steps are made to reduce the unbalanced force ratio to approximately $10^{-5}$. This two-step process is repeated until the 50-year thermal time is reached. The emplacement drift is not ventilated during this period; adiabatic boundaries are assumed for the drift.
Example Applications

Figure 6.2 UDEC geometry for thermomechanical analysis of a waste emplacement drift

Figure 6.3 Close-up view of waste emplacement drift
6.3 Results

The results of the analyses are shown in Figures 6.4 to 6.8. Figures 6.4 and 6.5 show the stress and displacement distributions that result from drift excavation. The temperature distribution at 50 years is shown in Figure 6.6. Figure 6.7 shows the stress distribution at 50 years. The extent of joint shear displacement is shown in Figure 6.8. In UDEC, shear displacement magnitudes are expressed by plotting multiple parallel lines along a joint. The thicker lines represent more shear displacement than thinner lines.

Figure 6.4 Principal stress distribution after excavation of the drift
Figure 6.5  Displacement vectors after excavation of the drift

Figure 6.6  Temperature distribution in the rock at 50 years
Figure 6.7  Principal stress distribution in the rock at 50 years

Figure 6.8  Shear displacement along the joints at 50 years
6.4 References


6.5 Listing of Data File

Example 6.1 DRIFT.DAT

```
; ; T H E R M A L / M E C H A N I C A L A N A L Y S I S ; ;
; ; Input file for determining emplacement room behavior. ; ;
; Vertical emplacement scheme ...
; ;
; config thermal
round 0.005
edge 0.01
block 0,-40 0,100 19.2,100 19.2,-40
; ; ; ;
; large block cracks
; ; ; ;
crack (0,43) (19.2,43)
crack (0,16) (19.2,16)
; ; ; ;
; ; ;
; emplacement room cracks
; ; ;
crack 0.0,36.5 1.0,36.5
crack 1.0,36.5 2.0,36.0
crack 2.0,36.0 2.5 35.0
crack 2.5,27.0 2.5,40.0
crack 0.0,30.0 6.0,30.0
; ; ; ;
; ; heavily jointed region
; ; ;
jregion id 1 0.0,16.0 0.0,43.0 7.0,43.0 7.0,16.0
jset angle 0 trace 1 gap 1 spacing 2 origin 1,0 range jregion 1
```

UDEC Version 5.0
jset angle 0 trace 1 gap 1 spacing 2 origin 0,1 range jregion 1
jset angle 90 spacing 1 origin 1,0 range jregion 1

; make crack for heaters

; additional fine cracks

; bottom region

; top region

; right side

UDEC Version 5.0
Example Applications

; Example Applications;

; crack 10.5,16 10.5,43
; crack 14,16 14,43
; crack 7,19 10.5 19
; crack 7,23 19.2 23
; crack 7,27 19.2 27
; crack 7,31 19.2 31
; crack 7,35 19.2 35
; crack 7,39 10.5 39
;
; generate zones
;
; gen edge 1.4 range 0,7 16,43
; gen edge 4.2 range 7,20 16,43
; gen edge 4.2 range 0,20 4,16
; gen edge 14 range 0,20 -40,4
; gen edge 4.2 range 0,20 43,55
; gen edge 14 range 0,20 55,100
;
;--- Assign Material Properties (Ref: SCP-CDR Chap. 2, Sec. 2.3.1)
;--- USING THE JOINT PROPERTIES AND "ROCK MASS" PROPERTIES.
;--- USING THE 'DESIGN' VALUES FROM
;--- TABLES 2-4, 2-6, AND 2-7.
;--- THE ROCK IS CHARACTERIZED AS AN ELASTIC/PLASTIC MATERIAL
;--- WITH VERTICAL AND HORIZONTAL. A COULOMB FAILURE CRITERION
;--- IS USED FOR THE JOINTS ...
;
;--- Thermal Properties of the Rock ...
; (Ref: SCP-CDR Chap. 2, Sec. 2.3.1.9, Table 2-9)

; group zone 'intact rock'
zone model elastic density 2.34E3 bulk 8.39E9 shear 6.29E9 cond 2.07 &
  specheat 961 thexp 1.07E-5 range group 'intact rock'
;
;--- Rock Joints:
; group joint 'joints'
joint model residual jks 1E11 jkn 1E11 jfriction 39 jcohesion 1E6 range &
  group 'joints'
; new contact default
set jcondf joint model residual jks=1E11 jkn=1E11 jfriction=39 &
  jcohesion=1E6
;
;--- Define the Initial Stress Field (MPa)...
;--- Reference: SCP-CDR Chap. 2, Sec. 2.3.1.9
; (The initial vertical stress is about -7 MPa at
; the disposal room horizon. The horizontal stress

UDEC Version 5.0
is determined as 0.5 x SYY.

; in situ stress -3500000.0,0.0,-7000000.0 xgrad 0.0,0.0,0.0 ygrad &
  11700.0,0.0,23400.0
initemp 26.0
set gravity=0.0 -9.81

; --- SET KINEMATIC BOUNDARY CONDITIONS ...
; (The two vertical boundaries are symmetry planes, thus,
; they are restricted from moving in the horizontal (x)
; direction. The bottom horizontal boundary is restricted
; from moving in the vertical (y) direction. The top
; horizontal boundary is a free-to-move pressure boundary.
; The pressure is acting downward, and is equal to the
; initial vertical stress.)

; boundary xvelocity 0 range -0.1,0.1 -40.1,100.1
boundary yvelocity 0 range -0.1,19.3 -40.1,-39.9
boundary stress -3500000.0,0.0,-7000000.0 xgrad 0.0,0.0,0.0 ygrad &
  11700.0,0.0,23400.0 range -0.1,19.3 99.9,100.1
save cycle0.sav

; solve ratio 1.0E-5
save m0e.sav

; --- EXCAVATE THE DISPOSAL ROOM ...

; reset jdisp disp
delete range 0,2.5 30,35
delete range 2,2.3 35,35.5
delete range 0,1.55 35,36.2
solve ratio 1.0E-5
save m0d.sav

; --- ASSIGN THE DECAYING HEAT SOURCE WHICH SIMULATES THE
; --- COMMINGLED SF AND DHLW ...
; (The thermal decay characteristics are from Peters, 1983,
; SAND-2497. The initial heat generating power per meter
; of room length is 713.5 W. Because of symmetry only half
; of this power is applied. Note that the decay coefficients
; have dimension 1/sec and not 1/year, which is commonly
; used in the literature ...)
; decay constants for SF are also used for the DHLW.
; thapp flux 48.16 -2.46E-10 range -0.1,0.1 23,27
thapp flux 41.03 \(-1.72E-9\) range \(-0.1,0.1\) 23,27
reset hist
history temperature 0.0,30.0
history temperature 0.0,36.7
history temperature 2.5,30.0
history temperature 1.0,25.0
history temperature 2.0,25.0
history temperature 3.0,25.0
history temperature 5.0,25.0
history temperature 9.0,25.0
history temperature 18.0,25.0
history ydisplace 0.0,36.5
history ydisplace 0.0,30.0
history xdisplace 2.5,33.0
history ydisplace 1.5,36.2
history sxx 0.0,36.5
history sxx 0.0,30.0
history syy 2.5,33.0

;--- START THE HEAT TRANSFER SOLUTION USING THE IMPLICIT SCHEME ...

; set nmech=2000
set ntherm=1000
set thdt=80000.0
run implicit age 1.58E9 step 100000 temp 200.0
solve age 0.0 ratio 1.0E-5
save t50c.sav
7 Inflow into a Tunnel

7.1 Problem Statement

Solid deformation and groundwater flow are interdependent in many situations in soil and rock mechanics engineering practice: (1) groundwater exerts pressure on the solid portion of the rock mass; while (2) deformation of the rock mass controls conditions of groundwater flow (i.e., changes hydraulic apertures of joints through which groundwater moves). Ignoring coupling between two processes (solid deformation and groundwater flow) may sometimes lead to incorrect predictions of the response of the rock mass to mechanical perturbations (e.g., tunnel excavation, pumping from a well and construction of a dam).

A simple problem of excavation of a tunnel below the water table in a jointed rock mass is analyzed in this example (see Figure 7.1). Material properties (e.g., initial joint hydraulic aperture, joint stiffness and stiffness of the whole assemblage of blocks) and perturbation to the system (e.g., change of total stress and pore pressure due to excavation of the tunnel) are chosen such that full, two-way coupling must be taken into consideration.

The drainage of groundwater into the excavated tunnel causes significant drawdown of the groundwater table and formation of a phreatic surface at a distance above the tunnel comparable to the radius of the tunnel. Therefore, the logic for unsaturated flow (see Section 2 in Special Features) is used in this example to provide accurate modeling of unconfined flow.

Figure 7.1 UDEC model for inflow into a tunnel
7.2 UDEC Analysis

The model is two-dimensional. Two continuous, orthogonal (horizontal and vertical) sets of joints form the blocks of the rock mass (as shown in Figure 7.1). Joint spacing is 10 m in both directions. Several material properties are assumed in the example:

- Shear modulus of rock \((G)\) = 15 GPa
- Bulk modulus of rock \((K)\) = 20 GPa
- Density of rock \((\rho)\) = 2700 kg/m\(^3\)
- Joint normal stiffness \((k_n)\) = 10 GPa/m
- Joint shear stiffness \((k_s)\) = 10 GPa/m
- Zero stress hydraulic aperture \((a_o)\) = \(10^{-3}\) m
- Residual hydraulic aperture \((a_r)\) = \(5 \times 10^{-4}\) m
- Joint permeability factor \((k_j = \frac{1}{12\mu})\) = \(5 \times 10^8\) MPa\(^{-1}\)s\(^{-1}\) (\(\mu\) is the groundwater viscosity)
- Bulk modulus of groundwater \((K_w)\) = 0.2 GPa
- Density of groundwater \((\rho_w)\) = 1000 kg/m\(^3\)

Note that the bulk modulus of water is 2.0 GPa. We used a lower value (0.2 GPa) for two reasons: (1) there are gasses dissolved in the groundwater that increase the compressibility of the groundwater; and (2) a lower value of bulk modulus speeds convergence to steady state of the numerical calculation when using the compressible flow option.

The initial total stresses are isotropic, and increase from the top to the bottom of the model as a function of the density of the rock mass and gravity. The initial hydrostatic pore pressures are defined by the groundwater surface, which is also at the top of the model. The far-field boundary conditions for both the solid and the groundwater models are at equilibrium with the initial conditions. The effective stresses in the joints are initialized as the difference between the normal component of the initial, total block-stress vector in the joint plane and the initial pore pressure in the joints. Accordingly, deformation and hydraulic apertures of the joints are calculated as a function of the initial effective stresses and the normal stiffness of the joints. (Initial joint deformation can be inhibited, and in that case the zero stress hydraulic aperture, \(a_o\), becomes the initial stress state hydraulic aperture.) The initial state of stress in the model is shown in Figure 7.2, while the initial pore pressure distribution is shown in Figure 7.3. Some calculational stepping is necessary due to the irregular geometry.
**Figure 7.2** Initial total stresses in UDEC model

**Figure 7.3** Initial pore pressures in UDEC model
After establishing an initial stress state in both the solid and groundwater models, the tunnel is excavated. Excavation of the tunnel introduces a perturbation in both the solid model (the total radial stress at the boundary of the tunnel is reduced to zero) and the groundwater model (the pore pressure at the boundary of the tunnel is reduced to zero). However, the time scales of the mechanical process in the rock mass and the fluid flow process in the joints are of different orders of magnitude. The response of the solid model is at a much shorter time scale; in this example, the response is instantaneous. Therefore, the first stage of the response of the model to the tunnel excavation is undrained deformation: the solid model deforms, while pore pressure in the joints changes as a function of deformation of the solid model and the bulk modulus, $K_w$, of the groundwater only – there is no flow. Pore pressures and deformation of the model after undrained deformation are shown in Figures 7.4 and 7.5.

**Figure 7.4** Pore pressures after undrained deformation
After reaching mechanical equilibrium in the first, undrained stage of the simulation, the second stage is simulated: deformation due to drainage of the groundwater into the tunnel (i.e., consolidation). This process is time-dependent, controlled by dissipation of the groundwater. This stage is simulated in real (flow) time until steady-state flow is reached and the solid model is at an equilibrium state.

Consolidation (time-dependent deformation due to drainage of the groundwater into the tunnel) is simulated using three different options in UDEC for calculation of coupled flow: (1) transient compressible flow (SET flow compressible); (2) steady flow (SET flow steady); and (3) transient incompressible flow (SET flow incompressible). If the steady-state solution is path-independent (e.g., a linear model), these three options should yield the same results in the steady-state condition. The steady-flow option does not consider unsaturated flow: the groundwater table is determined from pore pressures only. However, in this problem, infiltration from the unsaturated zone above the water table must become zero in the limit. Thus, the three models should converge in the limit. It can be verified from Figures 7.7, 7.8 and 7.9 that the results obtained from the compressible and incompressible flow options approach those obtained using the steady-flow option. (Locations of the history points are indicated in Figure 7.6.) Note that the calculation times for the steady-flow and the incompressible-flow options are an order of magnitude faster than for the compressible-flow option. The compressible-flow simulation is only run for a flow time of approximately 20 sec. The results of the incompressible-flow simulation indicate that almost 200 sec. are required to reach steady state.

It appears that the incompressible transient-flow logic is the best method for the solution of this problem, since calculation time is almost the same as for the steady-flow option, while it simulates
complete transient response of the model. (The steady-flow calculation is only correct at the steady state.)

Pore pressures at steady-state flow are shown in Figure 7.10. The phreatic surface at steady state is indicated by the limit of the pore-pressure distribution in this plot. Flow rates in joints are shown in Figure 7.11.

Figure 7.6 Locations of the history points
Figure 7.7  Histories of flow rates during consolidation (SET flow compressible)

Figure 7.8  Histories of flow rates during the steady-flow calculation (SET flow steady)
**Figure 7.9** Histories of flow rates during consolidation (SET flow incompressible)

**Figure 7.10** Pore pressures at steady-state flow
7.3 Listing of Data File

Example 7.1 INFLOW.DAT

```
;File:inflow.dat
;Title:Inflow into a Tunnel
config gwflow
set flow steady on
round 0.1
edge 0.2
block 0,100 0,200 200,200 200,100

; add tunnel
arc (200,150) (210,150) 360 40

; two orthogonal joint sets
jset angle 90 trace 200 spacing 10 origin 5,0
jset angle 0 trace 200 spacing 10 origin 0,5
gen edge 20.0

; material properties
```
Example Applications

```
group zone 'tunnel:rock'
zone model elastic density 2.7E3 bulk 2E10 shear 1.5E10 range group & 
     'tunnel:rock'
group joint 'tunnel:joint'
joint model area jks 1E10 jkn 1E10 jfriction 30 jperm 300 ares 0.0005 & 
     azero 0.001 range group 'tunnel:joint'
set jmatdf=1
prop jmat=1 jks=1E10 jkn=1E10 jfriction=30 jperm=300 ares=0.0005 & 
     azero=0.001
fluid density=1000.0

; boundary conditions
boundary stress -5400000.0,0.0,0.0 xgrad 0.0,0.0,0.0 & 
     ygrad 27000.0,0.0,0.0 range -0.1,0.1 99.9,200.1
boundary yvelocity 0 range -0.1,200.1 99.9,100.1
boundary xvelocity 0 range 199.9,200.1 99.9,200.1
boundary impermeable range 199.9,200.1 99.9,200.1
boundary impermeable range -0.1,200.1 199.9,200.1
boundary pp 2000000.0 pygrad -10000.0 range -0.1,0.1 99.9,200.1
boundary pp 1000000.0 range -0.1,200.1 99.9,100.1

; initial conditions
set gravity=0 -10
insitu stress -5400000.0,0.0,-5400000.0 xgrad 0.0,0.0,0.0 ygrad & 
     27000.0,0.0,27000.0 ywtable 200

; mechanical / steady state flow
; ------------------------------
history ydisplace 100.0,200.0
history unbalanced
solve ratio 1.0E-5
save inflow1.sav

; compressible fluid / no flow
; -----------------------------
delete range annulus (200,150) 0 10
set flow compressible
fluid bulkw=2.0E8
reset jdisp disp
set flow=off
solve ratio 1.0E-5
save inflow2.sav

; compressible transient flow
; -----------------------------
reset jdisp hist time disp
```

UDEC Version 5.0
set ftime=0
history ncyc 100
history flowtime
history flowrate 191.0,155.0
history flowrate 191.0,145.0
history flowrate 196.0,158.0
history flowrate 196.0,142.0
history pp 150.0,150.0
history ydisplace 200.0,200.0
set flow=on
set nfmech=10
cycle time 31.0
set nfmech=1
cycle time 110.0
save inflow3.sav

; steady state flow
;-----------------
restore 'inflow2.sav'
reset jdisp hist time disp
set ftime=0
history ncyc 100
history flowtime
history flowrate 191.0,155.0
history flowrate 191.0,145.0
history flowrate 196.0,158.0
history flowrate 196.0,142.0
history pp 150.0,150.0
history ydisplace 200.0,200.0
set flow steady
set flow=on
cycle 5000
save inflow4.sav

; incompressible transient flow
;-----------------------------
restore 'inflow2.sav'
reset jdisp hist time disp
set ftime=0
history ncyc 1
history flowtime
history flowrate 191.0,155.0
history flowrate 191.0,145.0
history flowrate 196.0,158.0
history flowrate 196.0,142.0
history pp 150.0,150.0
history ydisplace 200.0,200.0
set flow incompressible
set flow=on
set maxmech=1000
set voltol=1.0E-4
set dtflow=0.01
cycle 20000
save inflow5.sav
8 Flow through a Jointed Rock Slope

8.1 Problem Statement

The stability of a slope in jointed rock is affected by the water level behind the slope. In this example, the water level is raised in stages until the slope becomes unstable. The failure of the slope occurs when the fluid pressure in the joints increases (and the effective normal stress in the joints decreases) such that the limiting shear strength of the joints at the slope face is reached.

The problem geometry, shown in Figure 8.1, consists of a slope in regularly jointed rock. The water level is raised in four stages to elevations of 6 m, 8 m, 9 m and 10 m above the slope toe. A steady-state flow analysis is performed at each stage.

![Figure 8.1 Problem geometry for example problem involving flow through jointed rock slope](image)

The following material properties are assumed for the jointed rock slope.

Rock Properties:
- density: 2500 kg/m$^3$
- bulk modulus: 16.7 GPa
- shear modulus: 10.0 GPa
Joint Mechanical Properties:

- normal stiffness: 10 GPa/m
- shear stiffness: 10 GPa/m
- friction angle: 25°

Joint Hydraulic Properties:

- permeability factor: $1 \times 10^8$ MPa$^{-1}$ sec$^{-1}$
- residual hydraulic aperture: $2 \times 10^{-4}$ m
- aperture at zero normal stress: $5 \times 10^{-4}$ m

Fluid Properties:

- density: 1000 kg/m$^3$

8.2 UDEC Analysis

The UDEC model is shown in Figure 8.2. The problem is modeled as a steady-state flow analysis by specifying SET flow steady. The water level is raised by changing the fluid pressure gradient for each stage with the BOUNDARY pp pygrad command. The data file is listed in Section 8.3.

Initially, the slope is brought to an equilibrium state under gravity loading. Then, the water level at the right-hand side is raised to 6 m above the slope toe; the water level on the left-hand side is maintained at the level of the slope toe. The slope is stable for this fluid pressure condition. The steady-state flow pattern for this condition is shown in Figure 8.3.

Next, the right-hand water level is raised to 8 m. The steady-state flow condition for the 8-m water height is shown in Figure 8.4. Again, the system is in equilibrium. The water level is then raised to 9 m. The flow condition is shown in Figure 8.5; the slope is still stable.

Finally, the water level is raised to the top of the slope. The flow pattern for this case is shown in Figure 8.6. With the water level at 10 m, the slope slides, as indicated by the displaced rock wedge shown in Figures 8.7.
Figure 8.2  Problem geometry for example problem involving flow through jointed rock slope

Figure 8.3  Water level at right-hand side equal to 6 m (maximum flow rate $= 5.6 \times 10^{-5}$ m$^3$/sec)
Figure 8.4  Water level at right-hand side equal to 8 m (maximum flow rate
= \(7.4 \times 10^{-5}\) m\(^3\)/sec)

Figure 8.5  Water level at right-hand side equal to 9 m (maximum flow rate
= \(8.1 \times 10^{-5}\) m\(^3\)/sec)
**Figure 8.6** Water level at right-hand side equal to 10 m (maximum flow rate $= 9.4 \times 10^{-5}$ m$^3$/sec)

**Figure 8.7** Slope failure with water level at 10 m
8.3 Listing of Data File

Example 8.1  SLOPEFLO.DAT

;==============================================================================
;
; --- fluid flow test run ---
;
; --- slope : 10 m high ---
;
; --- 2 joint sets : 20 and 80 deg.
;   friction = 30 deg.
;
; --- r.h.s. water level : 6 m --- no failure ---
;                      8 m
;                      9 m
;                      10 m --- failure ---
;
;==============================================================================

config gwflow
set flow clear steady off
round 0.05
edge 0.1
block 0,-5 0,0 5,0 11,10 23,10 23,-5
jset angle 20 spacing 2 origin 5,1
jset angle 80 spacing 3 origin 5,0
delete area 0.1
gen edge 10.0
group zone 'rock'
zone model elastic density 2.5E-3 bulk 1.66667E4 shear 1E4 range group & 'rock'
group joint 'joint'
joint model area jks 1E4 jkn 1E4 jfriction 45 jperm 1E8 ares 0.0002 azero &
                          0.0005 range group 'joint'
; new contact default
set jcondf joint model area jks=1E4 jkn=1E4 jfriction=45 jperm=1E8 &
                ares=2E-4 azero=5E-4
fluid density=0.0010
insitu stress -0.125,0.0,-0.25 xgrad 0.0,0.0,0.0 ygrad 0.0125,0.0,0.025
boundary xvelocity 0 range -0.1,0.1 -5.1,0.1
boundary yvelocity 0 range 22.5,23.1 -5.1,10.1
boundary yvelocity 0 range -0.1,23.1 -5.1,-4.9
set gravity=0 -10
history xdisplace 11.0,10.0
history ydisplace 11.0,10.0
history unbalanced

UDEC Version 5.0
solve ratio 1.0E-5 elastic
save slfl1.sav
;
; --------------------------------------
; flow --- r.h.s. water at y=6m
;
; --- no failure ---
;
; --------------------------------------
;
set flow on
boundary impermeable range -0.1,23.1 -5.1,-4.9
boundary pp 0.0 pygrad -0.01 range -0.1,0.1 -5.1,0.1
boundary pp 0.06 pygrad -0.01 range 22.5,23.1 -5.1,6
history xdisplace 5.92,1.54
history ydisplace 5.92,1.54

joint model area jks 1E4 jkn 1E4 jfriction 25 jperm 1E8 nwjperm 1 ares &
0.0002 azero 0.0005 empb 1 expa 3 range group 'weak joint'

prop jmat=1 jks=1E4 jkn=1E4 jfriction=25 jperm=1E8 nwjperm=1 ares=0.0002 &
azero=0.0005 empb=1 expa=3

solve ratio 1.0E-5
save slf12.sav
;
; --------------------------------------
; flow --- r.h.s. water raised to y=8m
;
; --- some slip, but no block failure ---
;
; --------------------------------------
;
boundary pp 0.08 pygrad -0.01 range 22.5,23.1 -5.1,8
solve ratio 1.0E-5
save slf3.sav
;
; --------------------------------------
; flow --- r.h.s. water raised to y=9m
;
; --- no failure ---
;
; --------------------------------------
Example Applications

boundary pp 0.09 pygrad -0.01 range 22.5,23.1 -5.1,9.1
solve ratio 1.0E-5
save slf14.sav
;
; --------------------------------------
; flow --- r.h.s. water raised to y=10m
;
; --- failure ---
;
; --------------------------------------
;
boundary pp 0.10 pygrad -0.01 range 22.5,23.1 -5.1,10.1
cyc 15000
save slf5.sav
ret
9 Flow from a Borehole in a Biaxial Stress Field

9.1 Problem Statement

A borehole is located in a rock mass containing two orthogonal sets of joints and subjected to a biaxial in-situ stress state. Fluid is injected into the borehole at a constant flow rate. The purpose of the analysis is to evaluate the influence of the in-situ stress state on flow into the joints.

The in-situ stress field is

\[
\begin{align*}
\sigma_{xx} &= -25 \text{ MPa} \\
\sigma_{yy} &= -20 \text{ MPa} \\
p &= 10 \text{ MPa}
\end{align*}
\]

in which \(\sigma_{xx}\) and \(\sigma_{yy}\) are total stresses in the \(x\)- and \(y\)-directions, and \(p\) is the fluid pressure in the joints.

The intact rock block properties are

- density: \(2500 \text{ kg/m}^3\)
- bulk modulus: \(66.667 \text{ GPa}\)
- shear modulus: \(40.0 \text{ GPa}\)

The joint spacing is 2.5 m. The joint properties are

- normal stiffness: \(2 \times 10^5 \text{ MPa/m}\)
- shear stiffness: \(2 \times 10^5 \text{ MPa/m}\)
- permeability factor: \(300 \text{ m} \cdot \text{sec/kg}\)
- residual hydraulic aperture: \(2 \times 10^{-5} \text{ m}\)
- aperture at zero normal stress: \(1 \times 10^{-4} \text{ m}\)
9.2 UDEC Analysis

The UDEC model for this problem is shown in Figure 9.1. The data file is listed in Section 9.3.

For this transient flow analysis, the flow is assumed to be incompressible (SET flow incompressible). Fluid is injected at a constant flow rate of $10^{-3}$ m$^3$/sec into the borehole via the WELL command.

Given the difference in in-situ stresses, the dominant flow is in the horizontal joint set. Figure 9.2 shows the flow pattern after 15 seconds. The change in hydraulic aperture after 15 seconds is illustrated by the aperture plot in Figure 9.3.

The fluid pressure histories at five points in the model (see Figure 9.1) are plotted in Figure 9.4. Points 1 and 3, located on the horizontal joint, indicate a faster response than do points 2 and 4, located on the vertical joint.

Figure 9.1  UDEC model of a borehole in a rock mass containing two orthogonal joint sets
Figure 9.2  Flow in joints at 15 seconds after fluid flow initiates from the borehole

Figure 9.3  Joint hydraulic apertures at 15 seconds after fluid flow initiates from the borehole
Figure 9.4  Fluid pressure histories at points 1, 2, 3, 4 and 5
9.3 Listing of Data File

Example 9.1  BH.DAT

; --------------------------------------------------
; pressurized borehole --- biaxial stress field
;               insitu sxx=-25 syy=-20 pp=10
; borehole radius = 0.2
; --------------------------------------------------

;File: bh.dat
>Title: Flow from a Borehole in a Biaxial Stress Field
config fluid
set flow clear incompressible off
round 1E-3
edge 2E-3
block -10,-10 -10,10 10,10 10,-10
jset angle 0 spacing 2.5 origin 0,0
jset angle 90 spacing 2.5 origin 0,0
arc (0,0) (0.2,0) 360 8 join
delete range annulus (0,0) 0 0.2

gen edge 10.0

group zone 'rock'
zone model elastic density 2.5E-3 bulk 6.667E4 shear 4E4 range group 'rock'
group joint 'joint'
joint model area jks 2E5 jkn 2E5 jfriction 30 jperm 3E8 ares 2E-5 azero & 1E-4 range group 'joint'
; new contact default
set jcondf joint model area jks=2E5 jkn=2E5 jfriction=30 jperm=3E8 & ares=2E-5 azero=1E-4

insitu stress -25.0,0.0,-20.0 pp 10
boundary xvelocity 0 range -10.1,-9.9 -10.1,10.1
boundary xvelocity 0 range 9.9,10.1 -10.1,10.1
boundary yvelocity 0 range -10.1,10.1 -10.1,-9.9
boundary yvelocity 0 range -10.1,10.1 9.9,10.1
solve ratio 1.0E-5 elastic
save bh0.sav
;
reset hist time
history nycn 1
history pp -5.0,0.0
history pp 0.0,-5.0
history pp -9.0,0.0
history pp 0.0,-9.0
history pp 0.0,0.0
history flowtime
history unbvol

set flow=on
boundary pp 10.0
well flow 0.0010 atdomain (0,0)
set maxmech=1000
set voltol=1.0E-4
set dtflow=0.5
cycle ftime 15.0
save bh1.sav
10 Influence of the Placement of Backfill in a Deep Longwall Excavation

10.1 Problem Statement

The use of the null, strain-softening and double-yield models is demonstrated in this simulation in which (1) the cohesion of the rock sharply decreases when failure occurs during staged excavation of a seam, and (2) the backfill yields in compression.

A five-meter thick seam is excavated in a bedded rock mass that is cut by horizontal joints with a vertical spacing of 5 m. Using symmetry, only half the seam thickness (2.5 m) is represented by the analysis. Four (4) horizontal joints are located at elevations of 2.5 m, 7.5 m, 12.5 m and 17.5 m above the center of the seam. Figure 10.1 shows the UDEC model for this geometry.

The analysis includes five excavation steps. Fictitious vertical joints are placed every 10 m in the seam to model the phased excavation. After the first 10 m section is excavated, it is then backfilled, and the second section is excavated. This pattern is repeated for the five steps.

Figure 10.1 UDEC model of deep longwall excavation
The seam and surrounding rock mass are assigned the same properties:

- **density**: 2700 kg/m$^3$
- **bulk modulus**: 38.9 GPa
- **shear modulus**: 29.9 GPa
- **friction angle**: 45°
- **dilation**: 0
- **cohesion**: 20 MPa at zero plastic shear strain, decreasing linearly to 0 at 3% plastic shear strain

The joint properties are

- **normal stiffness**: 100 GPa / m
- **shear stiffness**: 10 GPa / m
- **friction angle**: 30°
- **cohesion**: 0.05 MPa
- **tensile strength**: 0

The backfill properties are

- **density**: 1000 kg/m$^3$
- **bulk modulus**: 0.45 GPa
- **shear modulus**: 0.6 GPa
- **friction angle**: 40°
- **dilation**: 5°
- **cohesion**: 0

At zero plastic volumetric strain, the backfill material yields in compression at a mean stress of 10 kPa. The compressive strength then increases with increasing volumetric strain, following the curve given in Figure 10.2.

The initial stress state before any excavation is

\[
\sigma_v = -67.5 \text{ MPa} \\
\sigma_h = -33.8 \text{ MPa}
\]
Figure 10.2  Cap pressure variation with plastic volumetric strain
10.2 UDEC Analysis

The purpose of this analysis is to investigate the response of the rock mass and the behavior of the backfill during staged excavation. A fictitious horizontal joint is added to the model immediately above the seam to create a finer zoning of this area (Figure 10.3), in order to obtain a clearer picture of the stress distribution in the rock and backfill as the excavation and backfilling progresses.

![Figure 10.3 Close-up view of the zoning for the 2.5-m thick seam and the roof](image)

The UDEC analysis begins at an equilibrium state defined by the initial stress conditions. The base boundary of the model is a “no vertical displacement” boundary (symmetry axis), while the two vertical boundaries have imposed zero horizontal displacement. Note that, in this example, the boundaries are too close to permit the accurate simulation of a single longwall excavation. The example is only intended to demonstrate the behavior of the various block constitutive models.

The seam is excavated by changing the seam blocks to a null material (*ZONE model null*), and backfilled by changing the null material to a double-yield material (*ZONE model dy*). The excavation phase begins by excavating (nulling) a 10-m long section of blocks at the lower-left corner of the model. After a 10-m section has been excavated, backfill is placed in the section while the next section is excavated. Note that the excavation and backfilling are performed instantaneously in this example. Figure 10.4 shows the material models assigned for the second excavation/fill step.
The vertical displacements in the roof, and the vertical stress in the backfill, are monitored while the excavation continues.

The stress states, after the seam has been excavated 10 m, 20 m, 30 m, 40 m and 50 m, are shown in Figures 10.5 through 10.9. The distribution of the vertical stress in the backfill is superimposed on the stress plots. These figures give a broad picture of the system response. A classical arching effect develops.

Very far from the face, the vertical stress sustained by the backfill must approach the in-situ stress (67.5 MPa). It can be seen in Figure 10.9 (40 m behind the face excavated for 50 m) that the backfill is sustaining only about one-third of the load. If a Mohr-Coulomb material is used in place of the double-yield model, the backfill would have already taken up most of the load (compare Figure 10.9 to Figure 10.10). The volumetric failure produced by the double-yield model results in the reduced load in the backfill.
Figure 10.5 Stress state, 10 m excavated

Figure 10.6 Stress state, 20 m excavated and 10 m backfilled
Figure 10.7  Stress state, 30 m excavated and 20 m backfilled

Figure 10.8  Stress state, 40 m excavated and 30 m backfilled
Example Applications

**Figure 10.9** Stress state, 50 m excavated and 40 m backfilled

**Figure 10.10** Stress state, 50 m excavated and 40 m backfilled with Mohr-Coulomb material
10.3 Listing of Data File

Example 10.1 LONGWALL.DAT

```
new
;Project Record Tree export
;File:longwall.dat
;Title:Backfill in Longwall Excavation
;Branch 1:fill0.sav
config
round 1E-2
edge 2E-2
set ovtol 1.0
block 0.0 0.50 80.50 80.0
crack (0,2.5) (80,2.5)
crack (0,7.5) (80,7.5)
crack (0,12.5) (80,12.5)
crack (10,0) (10,2.5)
crack (20,0) (20,2.5)
crack (30,0) (30,2.5)
crack (40,0) (40,2.5)
crack (50,0) (50,2.5)
crack (60,0) (60,2.5)
crack (70,0) (70,2.5)
crack (0,3.5) (80,3.5) join
gen quad 2.0,0.5 range 0.80 2.5.8.5
gen quad 2.0,1.0 range 0.80 3.5,7.5
gen quad 8.0,5.0 range 0.80 7.5,17.5
gen quad 16.0,16.5 range 0.80 17.5,50
zone 'rock'
zone model ss density 2.7E3 bulk 3.89E10 shear 2.99E10 friction 45 &
  cohesion 2E7 ctable 1 range group 'rock'
table 1 delete
  table 1 0 2E7 3E-2 0
  group joint 'joint'
joint model area jks 1E10 jkn 1E11 jfriction 30 jcohesion 5E4 &
  range group 'joint'
; new contact default
set jcondf joint model area jks=1E10 jkn=1E11 jfriction=30 jcohesion=5E4
insitu stress -3.38E7,0.0,-6.75E7 szz -5.0E7
boundary stress 0.0,0.0,-6.75E7 range -1E-2,80.01 49.99,50.01
boundary xvelocity 0 range -1E-2,2E-2 -1E-2,250.01
boundary yvelocity 0 range -1E-2,280.01 -1E-2,41E-2
history syy 10.05,2.2

UDEC Version 5.0
```
Example Applications

history syy 20.05,2.2
history syy 30.05,2.2
history syy 40.05,2.2
history syy 50.05,2.2
history ydisplace 5.0,2.5
history ydisplace 15.0,2.5
history ydisplace 25.0,2.5
history ydisplace 35.0,2.5
history ydisplace 45.0,2.5
solve ratio 1.0E-5 elastic
save fill0.sav

;Branch 2:fill1.sav
group zone 'Null:chamber 1' range atblock (5,2.25) (5,1)
zone model null range group 'Null:chamber 1'
solve ratio 1.0E-5
save fill1.sav

;Branch 3:fill2.sav
group zone 'backfill' range group 'Null:chamber 1'
zone model dy density 1E3 bulk 4.5E8 shear 6E8 friction 40 dilation 5 &
cap_pressure 1E4 ctable 2 range group 'backfill'
table 2 delete
table 2 0 1E4 2E-2 2E5 4E-2 8E5 6E-2 1.3E6 0.1 5E6 0.12 8.5E6 &
0.14 1.15E7 0.16 1.9E7 0.18 3.4E7 0.2 5E7
group zone 'Null:chamber 2' range atblock (15.0005,2.2473) (15.0004,0.9998)
zone model null range group 'Null:chamber 2'
solve ratio 1.0E-5
save fill2.sav

;Branch 4:fill3.sav
group zone 'backfill' range group 'Null:chamber 1'
zone model dy density 1E3 bulk 4.5E8 shear 6E8 friction 40 dilation 5 &
cap_pressure 1E4 ctable 2 range group 'backfill'
group zone 'Null:chamber 3' range atblock (25.005,2.2442) (25.0044,0.9995)
zone model null range group 'Null:chamber 3'
solve ratio 1.0E-5
save fill3.sav

;Branch 5:fill4.sav
group zone 'backfill' range group 'Null:chamber 3'
zone model dy density 1E3 bulk 4.5E8 shear 6E8 friction 40 dilation 5 &
cap_pressure 1E4 ctable 2 range group 'backfill'
group zone 'Null:chamber 4' range atblock (35.0044,1.2494)
zone model null range group 'Null:chamber 4'
solve ratio 1e-5

UDEC Version 5.0
save fill4.sav

;Branch 6:fill5.sav
group zone ‘backfill’ range group ‘Null:chamber 4’
zone model dy density 1E3 bulk 4.5E8 shear 6E8 friction 40 dilation 5 &
cap_pressure 1e4 ctable 2 range group ‘backfill’
group zone ‘Null:chamber 5’ range atblock (45.0044,1.2494)
zone model null range group ‘Null:chamber 5’
solve ratio 1e-5
save fill5.sav
set pline 0 2 40 2 40

;*** plots ****
;plot name: Hist unbal
plot hold history 0 line
;plot name: material
plot hold model block
;plot name: ctable
plot hold table 2 both
;plot name: Plot 3
plot hold block red stress

UDEC Version 5.0
11 Shotcrete and Cable Support

11.1 Problem Statement

This example application demonstrates the use of the structural element logic in UDEC to simulate the support of a circular excavation provided by combining a shotcrete lining and cable bolts.

The geometry for this example is illustrated in Figure 11.1. A circular tunnel is excavated in a rock containing a high-angle continuous joint set oriented at 50° dip, with an average spacing of 7 m. The excavation also intersects a vertical fault, creating a triangular wedge above the crown of the tunnel.

![Figure 11.1 UDEC model of a circular excavation in a rock mass containing a high-angle joint set and a vertical fault – tunnel support with a shotcrete lining and cable bolts](image-url)
The properties of the rock, joints and fault are summarized:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Intact Rock</strong></td>
<td></td>
</tr>
<tr>
<td>Bulk modulus</td>
<td>1.5 GPa</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>0.6 GPa</td>
</tr>
<tr>
<td>Density</td>
<td>2500 kg/m³</td>
</tr>
<tr>
<td><strong>Joints and Fault</strong></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness</td>
<td>2 GPa/m</td>
</tr>
<tr>
<td>Shear stiffness</td>
<td>2 GPa/m</td>
</tr>
<tr>
<td>Friction angle</td>
<td>10°</td>
</tr>
<tr>
<td>Cohesion</td>
<td>100 Pa</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>100 Pa</td>
</tr>
</tbody>
</table>

Several properties are used for the shotcrete lining and cable bolts:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Shotcrete:</strong></td>
<td></td>
</tr>
<tr>
<td>Thickness</td>
<td>0.1 m</td>
</tr>
<tr>
<td>Density</td>
<td>2500 kg / m³</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>21 GPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.15</td>
</tr>
<tr>
<td>Tensile yield strength</td>
<td>2 MPa</td>
</tr>
<tr>
<td>Residual yield strength</td>
<td>1 MPa</td>
</tr>
<tr>
<td>Compressive yield strength</td>
<td>4 MPa</td>
</tr>
<tr>
<td><strong>Rock/Shotcrete Interface:</strong></td>
<td></td>
</tr>
<tr>
<td>Normal stiffness</td>
<td>1 GPa / m</td>
</tr>
<tr>
<td>Shear stiffness</td>
<td>1 GPa / m</td>
</tr>
<tr>
<td>Friction</td>
<td>45°</td>
</tr>
<tr>
<td>Cohesive strength</td>
<td>1 MPa</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>1 MPa</td>
</tr>
<tr>
<td><strong>Cable Bolts:</strong></td>
<td></td>
</tr>
<tr>
<td>Cable area</td>
<td>$10^{-3}$ m²</td>
</tr>
<tr>
<td>Cable length</td>
<td>20 m</td>
</tr>
<tr>
<td>Cable modulus (E)</td>
<td>100 GPa</td>
</tr>
<tr>
<td>Cable ultimate tensile strength</td>
<td>10 MN</td>
</tr>
<tr>
<td>Grout bond stiffness</td>
<td>$10^9$ N/m/m</td>
</tr>
<tr>
<td>Grout cohesive strength</td>
<td>$10^6$ N/m</td>
</tr>
</tbody>
</table>

*UDEC Version 5.0*
For demonstration purposes, the tunnel is excavated and support installed instantaneously. Two support analyses are presented. In the first, only a shotcrete lining is applied. In the second, both shotcrete and cable bolts are used to provide support.

The parameters selected for this example are not intended to represent typical cable and shotcrete properties; they are chosen to provide a clear demonstration of the response of the cable and structural-element support models. For example, the compressive strength of the shotcrete is set to a very low value so that the liner fails in the first analysis; the support provided by the cables is then clearly seen in the second analysis.

11.2 UDEC Analysis

In the first analysis, the shotcrete lining is installed in the upper half of the tunnel. The model is brought to an equilibrium state prior to excavating the tunnel. Then the tunnel blocks are deleted, the lining is installed and the calculation is continued. A y-displacement history is recorded at one point in the tunnel crown. The history plot in Figure 11.2 indicates that the tunnel roof is collapsing. Figure 11.3 plots a close-up view of the tunnel region and lining; the wedge is falling and the lining is failing.

In the second analysis, cables are installed in the region of the roof wedge along with the shotcrete lining, as shown in Figure 11.1. In order to combine the support provided by both structural element types, the cables must be connected to the lining. This is accomplished by specifying the connect keyword with the CABLE command. This will position the cable node to coincide with the nearest structural element node on a lining.

The FISH function place_cables is used to place the cables around the lining, and position the starting node of each cable to coincide with a lining node. The cable pattern is defined by an origin location for the cable pattern (xOrigin,yOrigin), a radius to the remote end of the first cable in the pattern (radius1), a radius to the remote end of the last cable in the pattern (radius2), a starting angle for the cable pattern (theta1) and an ending angle for the cable pattern (theta2). See the FISH data file, “SUP_CAB.FIS,” in Example 11.2 for the listing of this function.

The following SET commands define the cable pattern to span the region of the roof wedge block.

```
set xOrigin=0.0 yOrigin=0.0
set radius1=20 radius2=20
set theta1=80.0 theta2=130.0
```

When the analysis is repeated with the shotcrete and cable bolts, the roof wedge is stabilized after the wedge has moved approximately 0.45 m. Figure 11.4 plots the y-displacement history in the crown; the wedge stabilizes after approximately 0.45 m movement. The axial forces in the cables and lining are shown in the plot in Figure 11.5.
**Figure 11.2** y-displacement history recorded at a location on the roof wedge – unstable movement

**Figure 11.3** Close-up view of tunnel showing sliding of roof wedge and failure of shotcrete lining
**Figure 11.4**  y-displacement history recorded at a location on the roof wedge – movement stops

**Figure 11.5**  Close-up view of tunnel showing the stable roof wedge and axial forces in the cables and shotcrete lining
11.3 Listing of Data Files

Example 11.1 SUPPORT.DAT

;File:support.dat
config
round 0.1
edge 0.2
block -50,-50 -50,50 50,50 50,-50
jset angle 310 spacing 7 origin 0,0
crack (-6,-50) (-6,50)
arc (0,0) (9,0) 360 16 join
gen edge 10.0

group zone ‘rock’
zone model elastic density 2.5E3 bulk 1.5E9 shear 6E8 range group ‘rock’
group joint ‘joint’
joint model area jks 1E9 jkn 2E9 jfriction 10 jcohesion 100 jtension 100 &
range group ‘joint’
; new contact default
set jcondf joint model area jks=1E9 jkn=2E9 jfriction=10 jcohesion=100 &
jtension=100

boundary stress 0.0,0.0,-1.0E7 range -50,50 49,50
boundary xvelocity 0 range -51,-49 -51,51
boundary xvelocity 0 range 49,51 -51,51
boundary yvelocity 0 range -51,51 -51,-49
set gravity=0.0 -10.0
solve ratio 1.0E-5 elastic
save supp1.sav
;
; shotcrete liner only
reset hist time disp
history ydisplace 0.0,9.0
delete range annulus (0,0) 0 9
struct gen begin 8.6183,1.4905 end -8.6471,1.2842 max=100.0 min=0.2 mat 1
property mat 1 st_density 2.5E3 st_prat 0.15 st_ycomp 4E6 st_yield 2E6 &
st_ymod 2E10 st_yresid 1E6 st_area 0.2 st_inertia 2.5E-2 st_shape 0.8333 &
st_spacing 1 st_thickness 0.1 st_width 1 if_friction 45 if_cohesion 1E6 &
if_tensile 1E6 if_kn 1E9 if_ks 1E9
cycle 5000
save supp2.sav
;
; shotcrete liner and cable bolts
restore ‘supp1.sav’
reset hist time disp

UDEC Version 5.0
history ydisplace 0.0,9.0
delete range annulus (0,0) 0 9
struct gen begin 8.6183,1.4905 end -8.6471,1.2842 max=100.0 min=0.2 mat 1
property mat 1 st_density 2.5E3 st_prat 0.15 st_ycomp 4E6 st_yield 2E6 &
   st_ymod 2E10 st_yresid 1E6 st_area 0.2 st_inertia 2.5E-2 st_shape 0.8333 &
   st_spacing 1 st_thickness 0.1 st_width 1 if_friction 45 if_cohesion 1E6 &
   if_tensile 1E6 if_kn 1E9 if_ks 1E9
call 'sup_cab.fis'
set radius1=20 radius2=20 xOrigin=0.0 yOrigin=0.0 theta1=80.0
set theta2=130
place_cables
property mat 3 cb_area 1E-3 cb_dens 7.5E3 cb_fstrain 1E30 cb_ycomp 1E10 &
   cb_yield 1E7 cb_ymod 1E11 cb_kbond 1E9 cb_sbond 1E6 cb_spacing 1
solve ratio 1.0E-5
save supp3.sav
Example 11.2 SUP.CAB.FIS

;Name:place_cables
;Input:radius1/float/20/remote radius for first cable
;Input:radius2/float/20/remote radius for last cable
;Input:xOrigin/float/0.0/x-origin of cable pattern
;Input:yOrigin/float/0.0/y-origin of cable pattern
;Input:theta1/float/80.0/starting angle of cable pattern
;Input:theta2/float/130/ending angle of cable pattern

call str.fin
;def setup
;; Create vars for later use
; xOrigin = 0.0 ; x-coord of cable radial centroid
; yOrigin = 0.0 ; x-coord of cable radial centroid
; theta1 = 0.0 ; starting angle for cables
; theta2 = 180.0 ; ending angle for cables
; radius1 = 0.0 ; starting radius for remote end of cables
; radius2 = 0.0 ; ending radius for remote end of cables
;end

def place_cables
; This example places cable elements at structural nodes along
; a given arc of tunnel.
; clean out any existing tables that are in our way...
command
    table 95 delete
    table 96 delete
    table 97 delete
    table 98 delete
endcommand
; start struct node address
nodeIdx = str_node_head
counter = 0

; collect relevant nodes and sort according to increasing theta
loop while nodeIdx # 0
    _x = fmem(nodeIdx + $SNDX) - xOrigin
    _y = fmem(nodeIdx + $SNDY) - yOrigin
    _t = atan2(_y, _x)
    if _t < 0
        _t = 2 * pi + _t
    endif
    th1 = degrad * theta1
    th2 = degrad * theta2
    if _t >= th1 then

if _t <= th2 then
  counter = counter + 1
  table(95, _t) = int(nodeIdx)
endif
dendif
nodeIdx = imem(nodeIdx + $SNDNEXT)
endloop

; we have an ordered table of indices, so...
loop ii (1, counter)
  _x = fmem(int(ytable(95, ii)) + $SNDX) - xOrigin
  _y = fmem(int(ytable(95, ii)) + $SNDY) - yOrigin
  _r = sqrt(_x^2 + _y^2)

  xtable(96, ii) = _x + xOrigin
  ytable(96, ii) = _y + yOrigin
  xtable(97, ii) = _r

  ; create other endpoint for cables and place
  _r = radius1+(radius2-radius1)*float((ii-1))/float((counter-1))
  if xtable(97, ii) < _r
    ytable(97, ii) = _r
    ; calculate (x, y) for cable end
    xtable(98, ii) = ytable(97, ii) * cos(xtable(95, ii)) + xOrigin
    ytable(98, ii) = ytable(97, ii) * sin(xtable(95, ii)) + yOrigin

    ; place the cable and connect to liner
    _x1 = xtable(96, ii)
    _y1 = ytable(96, ii)
    _x2 = xtable(98, ii)
    _y2 = ytable(98, ii)
    command
      cable _x1 _y1 _x2 _y2 20 3 3 connect
    endcommand
  endif
dendif
endloop
end
12 Blocks Bouncing down Slope

12.1 Problem Statement

Engineers are sometimes required to design slopes that must prevent loose rocks from falling onto roadways. The slopes are designed with benches or berms intended to catch falling debris. The geometry of these measures is difficult to determine by computer modeling. The path and final resting position of a falling block is highly dependent on size, shape, location and the properties of the surface on which it bounces. This is an example of a chaotic system where the final result is extremely sensitive to initial conditions. The final result is also sensitive to the timestep in UDEC. UDEC will update the timestep at each STEP or CYCLE command, so the results may vary if multiple cycle commands are used during an analysis.* It is often the case that this type of analysis is done by varying the input parameters and presenting a statistical result. The cell space contact-detection logic in UDEC can be used to look at problems where there may be many flying blocks.

The geometry in this case is shown in Figure 12.1. The blocks in this example are for demonstration purposes, and are much larger than would normally be used in this type of analysis. The blocks in this model are modeled as rigid blocks. All properties in this analysis are assumed. While the density of the blocks is fairly easy to determine, the stiffnesses of the joints may be difficult to obtain. A range in values should be used to determine the range of results.

The properties of rock and joints are summarized:

- **Rock**
  - density \(2500 \text{ kg/m}^3\)

- **Joints**
  - normal stiffness \(5 \text{ GPa/m}\)
  - shear stiffness \(5 \text{ GPa/m}\)
  - friction \(30^\circ\)

* If only point contacts ([JOINT model point](#)) are used, the timestep will remain constant.
12.2  **UDEC Analysis**

The first task in this type of analysis is to determine the damping parameters that will give an appropriate surface behavior. The bounce behavior of a block is often referred to as coefficient of rebound, and is the ratio of the height of the bounce to the height of the drop.

\[
Cr = \frac{Hr}{Hd}
\]

- \(Cr\) = coefficient of rebound
- \(Hr\) = height of rebound
- \(Hd\) = height of drop

In **UDEC**, the rebound can be controlled by manipulation of the damping parameters. Unfortunately, the damping parameters are global constants, so the rebound will vary for different size blocks. The damping should be set for the average block size. In this example, most blocks are 1.0 m square. Assume for this case that \(Cr\) is 0.5. The natural frequency of the system is

\[
\text{frequency} = \frac{1}{2\pi} \sqrt{\frac{kl}{m}} = 225 \text{ cycles/second}
\]

where
- \(l\) = joint length (1.0 m, in this case)
- \(k\) = joint stiffness (5 GPa, in this case)
- \(m\) = mass of upper block (2500 kg)

Using this frequency and dropping a single block from a height of 1.0 m, the fraction of critical damping required to get a rebound of 50% is 0.23. Therefore, the command

\[
damp 0.23 225 stiff
\]

will result in a coefficient of rebound of .5.

**Figure 12.2** shows the position and velocities after 25,000 cycles, and **Figure 12.3** shows the blocks and velocities after 50,000 cycles. It is clear that, for the parameters chosen in this case, the bench is not sufficient to catch the blocks falling down the slope.

The data file for this example is listed in **Section 12.3**. The file also demonstrates the **UDEC** commands to create a movie of the bouncing blocks.
Figure 12.1  UDEC model of blocks to be dropped onto the slope

Figure 12.2  Position and velocities of the blocks after 25,000 cycles
12.3 Listing of Data File

Example 12.1 BOUNCE.DAT

; File: bounce.dat
; cell logic example: slope rock fall with rigid blocks
; also demonstrates movie commands
;
config cell
round 0.01
block 0,0 0,15 3,10 8,9 11,2 15,2 15,0
block 2,12 2,15 4,15 4,12
crack 2,13 4,13
crack 2,13 3,15
crack 3,12, 3,15
crack 2,14 4,14
;
change mat 1
property mat 1 density 2.5e-3
joint joint 'area contact'
joint model area jkn 5E3 jks 5E3 jfriction 30 range group 'area contact'

Figure 12.3 Position and velocities of the blocks after 50,000 cycles
Blocks Bouncing down Slope

; new contact default
set jcond= joint model area jkn 5E3 jks 5E3 jfriction 30
;
fix 0 15 0 10
grav 0 -10
;
damping .23 225.0 stiffness
set ovtol .1
;
pl bl vel iw max 5
;
movie file = bounce.dcx
movie size 620 450
movie on step 1000
step 25000
step 25000
step 50000
movie off
;
pl bl vel hold
save bounce.sav
ret
13 Step-Path Failure of Rock Slopes

13.1 Problem Statement

While *UDEC* represents a jointed rock structure as a system of discrete blocks by default, it is also a straightforward matter to simulate the case in which joints terminate within intact rock. This is illustrated in this example application, which analyzes the stability of a simple slope containing non-continuous en-echelon jointing. If failure occurs both along the joints and within the intact rock, a “step-path failure” involving a combination of shear failure along the joints and shear and tensile failure within the intact rock bridging between the joints can develop.

Three en-echelon joints exist within the slope, as depicted in *Figure 13.1*. The three joints have a dip of 35°, and terminate within the rock near the slope surface. The slope dips at 50°, and has a height of 11.8 m.

![Figure 13.1 Geometry of simple slope containing en-echelon joints](image)

The properties of the intact rock are

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>density</td>
<td>2000 kg/m³</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>20 GPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>friction angle</td>
<td>25°</td>
</tr>
<tr>
<td>dilation angle</td>
<td>0</td>
</tr>
<tr>
<td>cohesion</td>
<td>25,000 Pa</td>
</tr>
<tr>
<td>tensile strength</td>
<td>0</td>
</tr>
</tbody>
</table>
The properties of the en-echelon joints are

- normal stiffness 100 GPa/m
- shear stiffness 10 GPa/m
- friction angle $35^\circ$
- cohesion 1000 MPa

The failure mode within the slope model is evaluated by performing factor-of-safety calculations based upon the shear strength reduction method (see Section 2 in Theory and Background). The failure mode can be identified when the strength properties are reduced to produce the state at which failure occurs.

### 13.2 UDEC Analysis

Discontinuous jointing can be simulated in UDEC by combining real joints with construction joints along the paths of the en-echelon joints. Note that construction joints effectively glue two blocks together so that the blocks behave as one intact block. The UDEC commands to create the en-echelon joints are described below, and the resulting joint structure is illustrated in Figures 13.2.

Cracks are introduced along the path of the three en-echelon joints by using the CRACK command to create real joints, and the CRACK ... join command to create a construction joint connecting the ends of real joints. Several commands are executed:

- `crack (17,8.2) (18.33,9.37)`
- `crack (18.33,9.37) (22.33,12.25)`
- `crack (22.33,12.25) (22.33,13.5)`
- `crack (22.33,13.5) (26.26,16)`
- `crack (26.26,16) (26.26,17.16)`
- `crack (26.26,17.16) (29.33,19.33)`
- `crack (29.33,19.33) (30,20)`

Each CRACK ... join command creates a construction joint that connects the ends of the real joints to produce a continuous joint through the model. Figure 13.2 displays the real joints, but not the construction joints.

For comparison, two separate models are created: the first contains no joints, and the second is created with three continuous joints located at the same position and orientation as the en-echelon joints. The three continuous joints are created with the commands

- `crack (17,8.2) (33.2,20)`
- `crack (15.5,8.2) (31.7,20)`
- `crack (14,8.2) (30.2,20)`

This joint structure is shown in Figure 13.3.
**Figure 13.2**  En-echelon joints in slope model

**Figure 13.3**  Continuous joints in slope model
A factor-of-safety calculation (SOLVE fos) is performed for all three models. The factor of safety for the en-echelon joint model is found to be 1.29. A step-path failure mode is indicated by the velocity vectors plot shown in Figure 13.4, and by the contour plot of joint maximum shear displacement and rock maximum shear strain shown in Figure 13.5.

For comparison, the factor of safety was also calculated for the model with no joints (FOS = 1.38, see Figure 13.6) and for the model with continuous joints through the slope (FOS = 1.01, see Figure 13.7). A circular failure mode within the intact rock is indicated in Figure 13.6. A planar shear failure mode along the joints is shown in Figure 13.7.
Figure 13.4  Step-path failure of slope – factor-of-safety = 1.29

Figure 13.5  Step-path failure depicted by joint maximum shear displacement and rock maximum shear strain
Figure 13.6  Continuum failure of slope – factor-of-safety = 1.38

Figure 13.7  Continuous joint shear failure of slope – factor-of-safety = 1.01
13.3 Listing of Data File

**Example 13.1 STEPPATH.DAT**

```
;File:steppath.dat
;Title:Step-Path Slope Failure
new
; model with en-echelon joint structure
round 4.5E-2
edge 9E-2
block 0,0 0,20 45,20 45,0
; create en-echelon joints
crack (0,8.2) (17,8.2)
crack (17,8.2) (26.9,20)
crack (17,8.2) (18.33,9.37) join
  crack (18.33,9.37) (22.33,12.25)
crack (22.33,12.25) (22.33,13.5) join
  crack (22.33,13.5) (26.26,16)
crack (26.26,16) (26.26,17.16) join
  crack (26.26,17.16) (29.33,19.33)
crack (29.33,19.33) (30,20) join
delete range 0,17 8,20
gen edge 1.0
save sp1.sav
; assign properties
group zone 'rock'
zone model mohr density 2E3 bulk 1.66667E10 shear 7.69231E9 friction 25 &
  cohesion 2.5E4 range group 'rock'
group joint 'joint'
joint model area jks 1E10 jkn 1E11 jfriction 35 jcohesion 1E3 range group &
  'joint'
; new contact default
set jcondf joint model area jks=1E10 jkn=1E11 jfriction=35 jcohesion=1000
; set boundary conditions
boundary xvelocity 0 range -0.1,0.1 -0.1,8.2
boundary xvelocity 0 range 44.9,45.1 -0.1,20.1
boundary yvelocity 0 range -0.1,45.1 -0.1,0.1
set gravity=0.0 -9.81
save sp2.sav
solve ratio 1.0E-5 elastic
save sp3.sav
; Factor of safety
solve fos no_restore file=FoSmodel.fsv
save FoSmodel.fsv
;
new
```
; model with continuous joint structure
round 4.5E-2
drive 9E-2
block 0,0 0,20 45,20 45,0
; create continuous joints
create (0,8.2) (17,8.2)
create (17,8.2) (26.9,20)
create (17,8.2) (33.2,20)
create (15.5,8.2) (31.7,20)
create (14,8.2) (30.2,20)
delete range 0,19 8,20
gen edge 1.0
save sp1a.sav

; assign properties
zone mohr density 2E3 bulk 1.66667E10 shear 7.69231E9 friction 25 & cohesion 2.5E4 range group 'rock'
zone mohr density 2E3 bulk 1.66667E10 shear 7.69231E9 friction 25 & cohesion 2.5E4 range group 'joint'
joint model area jks=1E10 jkn=1E11 jfriction=35 jcohesion=1E3 range group & 'joint'

; new contact default
set jcondf joint model area jks=1E10 jkn=1E11 jfriction=35 jcohesion=1000

; set boundary conditions
boundary xvelocity 0 range -0.1,0.1 -0.1,8.2
boundary xvelocity 0 range 44.9,45.1 -0.1,20.1
boundary yvelocity 0 range -0.1,45.1 -0.1,0.1
set gravity=0.0 -9.81
save sp2a.sav
solve ratio 1.0E-5 elastic
save sp3a.sav

; Factor of safety
solve fos no_restore file=FoSmode2.fsv
save FoSmode2.fsv

; new
; model with no joints
round 4.5E-2
drive 9E-2
block 0,0 0,8.2 17,8.2 26.9,20 45,20 45,0
gen edge 1.0
save sp1b.sav

; assign properties
zone mohr density 2E3 bulk 1.66667E10 shear 7.69231E9 friction 25 & cohesion 2.5E4 range group 'rock'

; set boundary conditions

boundary xvelocity 0 range -0.1,0.1 -0.1,8.2
boundary xvelocity 0 range 44.9,45.1 -0.1,20.1
boundary yvelocity 0 range -0.1,45.1 -0.1,0.1
set gravity=0.0 -9.81
save sp2b.sav
solve ratio 1.0E-5 elastic
save sp3b.sav
; Factor of safety
solve fos no_restore file=FoSmode3.fsv
save FoSmode3.fsv
;*** plots ****
;plot name: Hist unbal
plot hold history 0 line
;plot name: fos
plot hold fos block velocity green
14 Hydraulic Fracturing Simulation

14.1 Problem Statement

This is an example of using UDEC to simulate pressurized cracks and hydraulic fracturing. It documents the results of numerical simulations for fluid injection performed with UDEC. Two cases are presented. In Case 1, a uniform fluid pressure is applied inside a planar crack. The displacements of the crack surface are compared with an exact analytical solution. For Case 2, a viscous fluid is injected at a constant rate into a planar crack with zero toughness. The UDEC results for Case 2 are compared with both a zero toughness solution (Adachi and Detournay 2002) and a displacement discontinuity (DD) numerical solution.

14.1.1 Data for the Simulations

The medium is assumed to be elastic, Young’s modulus is 40 GPa and Poisson’s ratio is 0.22. The crack is straight with a length of 21.6 m.

The x-axis of reference is oriented along the fracture, with the origin located at mid-length. The UDEC sign convention is used in this section, whereby tension and extension are positive for the rock matrix. Joint opening is also positive. However, joint normal stress and fluid pressure are positive in compression.

The initial stress state, $\sigma_{yy} = -15$ MPa and $\sigma_{xx} = -30$ MPa, is applied. For Case 1, the fluid pressure is uniform and equal to 20 MPa. For Case 2, the injection rate is 0.0004 m$^2$/s, and fluid viscosity is 0.001 Pa · sec. The medium is initially dry. A total of 10 seconds on injection is considered for the numerical simulations, with intermediate results at 2.5, 5 and 7.5 seconds.

<table>
<thead>
<tr>
<th>case</th>
<th>E [MPa]</th>
<th>$\nu$</th>
<th>$\sigma_{yy}$ [MPa]</th>
<th>$\sigma_{xx}$ [MPa]</th>
<th>a [m]</th>
<th>p [MPa]</th>
<th>Q [m$^2$/s]</th>
<th>$\mu$ [Pa s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40,000</td>
<td>0.22</td>
<td>-15</td>
<td>-30</td>
<td>10.8</td>
<td>20</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>2</td>
<td>40,000</td>
<td>0.22</td>
<td>-15</td>
<td>-30</td>
<td>–</td>
<td>–</td>
<td>0.0004</td>
<td>0.001</td>
</tr>
</tbody>
</table>
14.2  **UDEC Analyses**

14.2.1  **Case 1**

The analysis for Case 1 considers the mechanical influence on stress and deformation of a uniform pressure being applied along a 21.6 meter crack. The fluid is simulated by specifying a domain pressure inside the crack.

14.2.1.1  **UDEC Model**

The UDEC model domain is 46.08 m by 46.08 m, with a zone size of 1.44 m. The matrix is elastic, with a Young’s modulus of 40,000 MPa and a Poisson’s ratio of 0.22. The embedded crack is modeled by preventing the ends of a throughgoing joint (located at mid-height in the model) from opening or sliding. This is achieved by assigning high strength properties to the contacts at the ends, and a value of joint stiffness equal to about 10 times the apparent stiffness of neighboring zones (see Eq. (3.1) in the User’s Guide). The in-situ stresses are specified in the model, and a boundary element representation is selected for the far field. A uniform domain fluid pressure is assigned inside the crack. The UDEC model and fluid pressure in the crack are shown in **Figure 14.1**:

![Figure 14.1 UDEC model – finite difference zones and fluid pressure in the crack](image)

Note that the crack tip is assumed to be located at the midpoint between “glued” and “unglued” nodes at the ends of the crack. Hence, the fracture being modeled is 21.6 m long and spans 14
zones. Fluid flow is turned off and the model is cycled to mechanical equilibrium. The data file for the simulation is provided in Example 14.1.

14.2.1.2 Analytical Solution

The analytical solution for fracture opening is given by (see, e.g., Parker 1981)

$$ w = \frac{\sigma_{yy} + p}{E} \frac{4(1 - v^2)}{\sqrt{a^2 - x^2}} $$

(14.1)

where $w$ is fracture opening.

14.2.1.3 UDEC Results

The UDEC results for fracture opening are compared to the analytical prediction in Figure 14.2:

The relative error on fracture opening at the center of the fracture is less than 1%.

The displacement vectors in the model at the end of the simulation are shown in Figure 14.3:
14.2.2 Case 2

The analysis for Case 2 involves full fluid-mechanical coupling. The UDEC modeling methodology is based on three components: 1) the joint is initially dry (saturation is zero); 2) it offers no resistance to opening, in accordance with the zero toughness condition being investigated; and 3) the condition of zero flow ahead of the fracture tip is enforced.

14.2.2.1 UDEC Model

The UDEC model for the Case 2 simulations is 23.04 m by 23.04 m. The zone size is uniform and equal to 0.18 m. The zoning is shown in Figure 14.4 for a 2 m by 2 m portion of the UDEC model. The matrix is elastic, with a Young’s modulus of 40,000 MPa and a Poisson’s ratio of 0.22. A throughgoing joint is specified at mid-height in the model. The initial stresses are specified, and stress boundary conditions are prescribed. The joint is being assigned a “small” initial aperture of $2 \times 10^{-5}$ m (which is done for numerical reasons), and saturation is initialized at zero.
A Coulomb slip with residual strength material model is assigned to the joint (joint model residual). This model has a special setting (SET j5flow on) that is switched on in this analysis, to allow flow to only occur in fractured segments of the joint. (A joint is fractured when the joint shear or tensile strength is exceeded. Also, after a joint is fractured, the residual friction, cohesion and tensile strength values are used). The joint stiffness is equal to about 10 times the apparent stiffness of neighboring zones. The joint has no cohesion (initial and residual values are zero). A high value of joint friction (45 degree) is assigned initially, to prevent premature fracturing during the transient phase experienced by the model as it reaches a quasi-static state. The residual value of joint friction is zero. To enforce the zero toughness condition, the joint tensile strength is set to an initial (negative) value (close to the initial normal stress, which is $-15$ MPa), but “slightly” smaller, to prevent fracture detection along the whole joint). The initial value of tensile strength is chosen to be $-14$ MPa for the runs; the residual value is zero.

The compressible flow algorithm is selected. A value of fluid bulk modulus of 100 MPa is specified. The value is high compared to fluid pressure changes in the model; it is adequate to simulate an “incompressible” fluid. The numerical simulation is carried out in quasi-static mode (a servo-control is used to enforce the condition that for each flow step enough mechanical steps are taken to maintain the model in quasi-static equilibrium). Fluid injection at a constant rate is specified for the well, located at the origin of axes (center of the model). The fluid-mechanical simulation is carried out for a total of 10 seconds of injection.
14.2.2.2  **UDEC Results**

The fluid pressure and hydraulic aperture are monitored at the well for the total simulation time covering 10 seconds of fluid injection. The model state is saved at intervals of 2.5 seconds for analysis of fracture pressure and width at intermediate times of 2.5, 5 and 7.5 seconds.

**Pressure and Width at the Well**

The fracture width is calculated from the hydraulic aperture, by subtracting the initial aperture which is \(2 \times 10^{-5}\) m for the runs. The fluid pressure and width at the well are plotted versus time in Figure 14.5 and Figure 14.6, respectively. The **UDEC** predictions are compared, in the same figure, with 1) the numerical DD solution, and 2) the first order approximation of the zero toughness solution (FMO) proposed by Adachi and Detournay (2002) (and Detournay 2004).

---

**Figure 14.5  Fluid pressure at the well (MPa) versus time (sec)**

---

**UDEC Version 5.0**
As may be observed from Figure 14.6, the UDEC solution for width at the well is bounded above by the analytical solution, and below by the DD solution.

**Pressure and Width Distribution in the Fracture**

The fracture pressure prediction from UDEC is compared to the first order approximation of the zero toughness solution (Adachi and Detournay 2002, and Detournay 2004) at 10 seconds of fluid injection in Figure 14.7.
It appears from Figure 14.7 that the UDEC pressure solution underestimates the value predicted by the analytical solution by about 7%.

The UDEC prediction for fracture width is compared to the FMO solution (Adachi and Detournay 2002, and Detournay 2004) at 10 seconds of fluid injection in Figure 14.8.
The fracture width predicted by *UDEC* in the reported simulations is underestimated when compared to the FMO solution by about 1%. However, the discrepancy for maximum value of width does not seem to grow with time in the reported results. On the other hand, the match for fracture length between the *UDEC* prediction and the FMO solution appears to be very good.

Displacement vectors in the vicinity of the fracture after 10 seconds of injection are shown in Figure 14.9.
Figure 14.9 Displacement vectors near the fracture (half fracture shown) at 10 sec.

The numerical simulation for 10 seconds of injection takes approximately 70 minutes to run on an Intel Core i7-870 with the model setup described in this section.
14.3 Summary

Numerical simulations have been carried out with UDEC to simulate fluid injection in a preexisting fracture with zero toughness. The numerical predictions for pressure and width at the well have been compared with available DD results and the analytical first order approximation of the zero toughness solution (FMO) of Adachi and Detournay (2002). Also, pressure and width along the fracture have been compared to the FMO solution. The reported UDEC results for fracture growth show an excellent match with the FMO solution. Fluid pressure at the well predicted by UDEC matches well with the FMO solution. The UDEC results for width at the well are bounded by the FMO solution and the DD solution.

14.4 References


14.5 Listing of Data Files

Example 14.1 HF_CASE1.DAT

new
;file hf_case1.dat
config fluid
; -------------------------------
; *** Case 1: Uniform pressure ***
; -------------------------------
title
  Uniform pressure
def_setup
  _young = 40e3 ; MPa
  _nu = 0.22
  _syy = -15.
  _sxx = -30.
  _pp = 20.
  _hl = 10.8 ; fracture half length
;
  _jkn = 3e4
  _ares = 5e-5
  _a0 = 5e-4
end
setup
;
round 0.01
crack -24 0 24 0
;
gen edge 1.6
;
join_contact
join_contact off range -10.5 10.5 -1 1
;
prop mat 1 dens 1e-3 ymod _young prat _nu
  --- crack ---
joint model resid jkn _jkn jks _jkn jperm 300 ares=_ares
;
fluid dens 1e-3
;
insitu str _sxx 0 _syy szz 0 _pp 0 aperture 0.0
;
bound stress _sxx 0 _syy
cycle 1
;
; --- boundary element representation of the far field ---
be gen -24 24 -24 24
be mat 1
be fix 0 -24 -24 0
be stiff
;
set dscan 100000 ; turn off scan for new contacts
       ; to speed calculation
; uniform pressure in fracture
pfix p _pp range -10.5 10.5 -.1 .1
;
; keep zero pp in impermeable joints
pfix p=0 range -50 -10.5 -.1 .1
pfix p=0 range 10.5 50 -.1 .1
;
hist n=1
hist type 1
hist ydis 0 0 ydis 2 0 ydis 4 0 ydis 6 0 ydis 8 0 ydis 10 0
label hist 1
Ydis at x=0
label hist 2
Ydis at x=2
label hist 3
Ydis at x=4
label hist 4
Ydis at x=6
label hist 5
Ydis at x=8
label hist 6
Ydis at x=10
save hf_case1_ft.sav
set flow steady
set flow off
set caprat 100
solve force 0 ratio 1e-6
save hf_case1.sav
rest hf_case1.sav

def fracop
; --- fracture opening ---
bpnt=block_head
loop while bpnt # 0
   pnt=b_gp(bpnt)
      loop while pnt # 0
         _x=gp_x(pnt)
Example Applications

\_y = \text{gp\_y}(pnt)
\begin{align*}
\text{if } \text{abs}(\_y) < 0.1 \text{ then} \\
\text{if } \text{abs}(\_x) < 10.5 \text{ then} \\
\quad \text{if } \text{bpnt} = \text{block\_head} \text{ then} \\
\quad \quad \text{table}(1, \_x) = \text{gp\_ydis}(pnt) \\
\quad \text{else} \\
\quad \quad \text{table}(2, \_x) = -\text{gp\_ydis}(pnt) ; \text{--(look: - sign)} \\
\text{endif}
\end{align*}
\text{endif}
\text{endif}
pnt = \text{gp\_next}(pnt)
endloop
bpnt = \text{b\_next}(bpnt)
endloop

\text{coe} = 2. \times (1. - _\text{nu} \times _\text{nu}) \times ((_\text{pp} + _\text{syy}) / _\text{young})
_hl = 10.8
nitem = \text{table\_size}(1)
\begin{align*}
\text{loop } \text{ii} (1, nitem) \\
\quad _x = \text{xtable}(1, \text{ii}) \\
\quad \text{xtable}(3, \text{ii}) = _x \\
\quad \text{ytable}(3, \text{ii}) = \text{ytable}(1, \text{ii}) + \text{ytable}(2, \text{ii}) \quad ; \text{numerical} \\
\quad \text{xtable}(4, \text{ii}) = _x \\
\quad \text{ytable}(4, \text{ii}) = 2. \times \text{coe} \times \text{sqrt}(_h_1*_hl-_x*_x) \quad ; \text{analytical opening}
\end{align*}
endloop

_\text{omax} = 2. \times \text{coe} \times _hl
_\text{err} = -100. \times (\text{table}(3, 0.) - _\text{omax}) / _\text{omax}
end
fracop
label table 3
Numerical opening
label table 4
Analytical opening
plot hold zone pp
plot hold table 3 cross 4 lin yw 0 5.5e-3
plot hold disp bound
print _\text{err}
ret
Example 14.2 HF_CASE2.DAT

new
;file hf_case2.dat
config fluid
; -----------------------------
; *** Case 2: viscous fluid ***
; -----------------------------
title
  Viscous fluid
def setup
  _young = 40e3 ; MPa
  _nu = 0.22
  _syy = -15.
  _sxx = -30.
  _hl = 10.8 ; fracture half length
  _edge = 0.2
;
  _bu = _young/(3.*(1.-2.*_nu))
  _sh = _young/(2.*(1.+_nu))
  _jkn = 3e6
  _jten = -14.
;
  _ares = 2e-5
  _a0 = -_syy/_jkn + _ares
  _amax = 6e-3
  _caprat = _amax/_ares
end
setup
;
round 0.01
block -11.52 -11.52 -11.52 11.52 11.52 11.52 11.52 -11.52
crack -12 0 12 0
;
gen edge _edge
;
prop mat 1 dens 1e-3 bulk _bu shear _sh
; --- crack ---
joint model res
joint jkn _jkn jks _jkn jcoh 0. jten _jten jfric 45.
joint jperm 83.33e6 azero _a0 ares _ares
;
fluid dens 1e-3
;
insitu str _sxx 0 _syy szz 0 pp 0
bound stress _sxx 0 _syy imperm
ini sat 0

set dscan 100000 ; turn off scan for new contacts
            ; to speed calculation
def setup2
    _cp0 = c_near(0.,0.)
end
setup2

set caprat _caprat

set flow compressible
fluid bulk 100
step 0
save hf_case2_ini.sav

def _kw
    _ap = _a0 + c_ndis(_cp0)
    _ap = max(_ares,_ap)
    _ap = min(_amax,_ap)
    _kw = 10.*(bu/1.44)*abs(_ap)
    _tdel = tdel
end

; injection well at x=0.
def in_flow
    in_flow=d_near(0.,0.)
end
well dom in_flow flow 4e-4

; hist n=100
hist type 9
hist ydis 0 0 ydis 2 0 ydis 4 0 ydis 6 0 ydis 8 0 ydis 10 0
hist unb
hist flowtime
hist pp .0 0 pp .72 0 pp 2.16 0 pp 3.60 0 pp 5.04 0 pp 6.48 0
hist pp 7.92 0 pp 9.36 0 pp 10.8 0
hist _kw _tdel
hist _ap
hist nstr .0 0 nstr .72 0 nstr 2.16 0 nstr 3.60 0 nstr 5.04 0 nstr 6.48 0
hist nstr 7.92 0 nstr 9.36 0 nstr 10.8 0
label hist 1
Ydis at x=0
label hist 2
Ydis at x=2
label hist 3
Ydis at x=4
label hist 4
Ydis at x=6
label hist 5
Ydis at x=8
label hist 6
Ydis at x=10
label hist 8
Flow Time
label hist 9
Fluid Pressure at x=.0
label hist 10
Fluid Pressure at x=.72
label hist 11
Fluid Pressure at x=2.16
label hist 12
Fluid Pressure at x=3.60
label hist 13
Fluid Pressure at x=5.04
label hist 14
Fluid Pressure at x=6.48
label hist 15
Fluid Pressure at x=7.92
label hist 16
Fluid Pressure at x=9.36
label hist 17
Fluid Pressure at x=10.80
label hist 18
Fluid bulk modulus
label hist 19
Fluid ? time step
label hist 20
Aperture at x=0

set j5flow=off

def _maxVel
    maxVel_ = 0.
iBlock_ = block_head
loop while iBlock_ # 0
    iGp_ = b_gp(iBlock_)
    loop while iGp_ # 0
        maxVel_ = max(maxVel_, sqrt(gp_xvel(iGp_)^2+gp_yvel(iGp_)^2))
iGp_ = gp_next(iGp_)
end_loop
iBlock_ = b_next(iBlock_)
end_loop
end

def _cycle
  nStep_ = nStep_
  loop while ftime < _ftime
    command
      cycle nStep_
    end_command
    tfdel_ = (ftime-ftimeOld_)/(nStep_*nf_)
    if tfdel_ < tfdelLimit_ then
      frac_ = frac_*tfdelLimit_/tfdel_
      command
        frac 0.1 1.0 frac_
      end_command
    endif
  maxVel
    if maxVel_ > upperBound_ then
      if nf_ > 1 then
        nfm_ = 1
        nf_ = nf_-1
      else
        nfm_ = nfm_+1
        nf_ = 1
      endif
    else
      if maxVel_ < lowerBound_ then
        if nfm_ > 1 then
          nfm_ = nfm_-1
          nf_ = 1
        else
          nfm_ = 1
          nf_ = nf_+1
        endif
      endif
    endif
    command
      set nfmech nfm_ nflow nf_
    end_command
  end_command
  ftimeOld_ = ftime
end_loop
end
set nStep_ 100 nfm_ 1 nf_ 1
set lowerBound_ 5e-4 upperBound_ 5e-3 tfdelLimit_ 0.5e-6 frac_ 1.0

set _ftime=2.5
_cycle
save hf_case2_2p5sec.sav
set _ftime=5.
_cycle
save hf_case2_5sec.sav
set _ftime=7.5
_cycle
save hf_case2_7p5sec.sav
set _ftime=10.
_cycle
save hf_case2_10sec.sav
; ------------------------------------------------------------------------
; fluid pressure and opening at the well
; comparison between:
; UDEC solution
; DD solution
; Analytical solution (Adachi and Detournay, 2002)
; ------------------------------------------------------------------------
;
res hf_case2_10sec.sav
hist write 8 table 1 ; time
hist write 9 table 2 ; pp
hist write 20 table 3 ; aperture
def ana_sol
  _num=table_size(1)
loop ii (2,_num)
  _tim = ytable(1,ii)
  _tim2 = _tim^1./3.
xtable(2,ii)=_tim
xtable(3,ii)=_tim
ytable(3,ii)=ytable(3,ii)-_ares
xtable(12,ii)=_tim
ytable(12,ii)=15.+1.508/_tim2
xtable(13,ii)=_tim
ytable(13,ii)=1.827*_tim2*1e-4
end_loop
end
ana_sol
label table 2
  Well p vs t: UDEC
label table 12
  Well p vs t: Analytical
label table 3

UDEC Version 5.0
Well width vs t: UDEC
label table 13
Well width vs t: Analytical
; --- import Displacement Discontinuity results ---
catable22.txt
catable23.txt

label table 22
Well p vs t: DD
label table 23
Well width vs t: DD

plot hold table 2 12 22
plot hold table 3 13 23
title
    Viscous fluid - 10sec

def _width
    pnt=contact_head
    loop while pnt# 0
        _x=c_x(pnt)
        _y=c_y(pnt)
        if _y < 0.1 then
            if _y > -0.1 then
                table(20,_x)=a0+c_ndis(pnt)-ares
            end_if
        end_if
        pnt=c_next(pnt)
    end_loop
    pnt=domain_head
    loop while pnt # 0
        _x=d_x(pnt)
        _y=d_y(pnt)
        if _y < 0.1 then
            if _y > -0.1 then
                if abs(_x) > 1e-2 then
                    table(30,_x)=d_pp(pnt)
                end_if
            end_if
        end_if
        pnt=d_next(pnt)
    end_loop
end
_width

catable50.txt
ca table51.txt

def ana_width
   _num=table_size(20)
   _time=10.
   _l=1.516*_time^(2./3.)
   _coew=1.623*_time^(1./3.)
   _coep=2.768*_time^(-1./3.)
   loop ii (1,_num)
      _x=xtable(20,ii)
      _xi=abs(_x)/_l
      _om=table(51,_xi)
      table(21,_x)=_coew*_om*1e-4
      _pi=table(50,_xi)
      _press=_coep*_pi + 15.
      if _pi = 0.0 then
         _press = 0.0
      end_if
      table(31,_x)=_press
   end_loop
end ana_width

label table 20
udec Fracture width
label table 21
Analytic Fracture width
label table 30
UDEC Fracture pressure
label table 31
Analytic Fracture pressure

plot hold table 20 21 xw -10 10 yw 0 4e-4
plot hold table 30 31 xw -10 10 yw 0 16.5
ret