ITASCA
Consulting Group, Inc.

# UDEC™ VERSION 6.0

## Distinct-Element Modeling of Jointed and Blocky Material in 2D

ICG14-BRO-UDEC600-US-3

CIVIL • MANUFACTURING • MINING • OIL & GAS • POWER GENERATION

# ABOUT *UDEC*

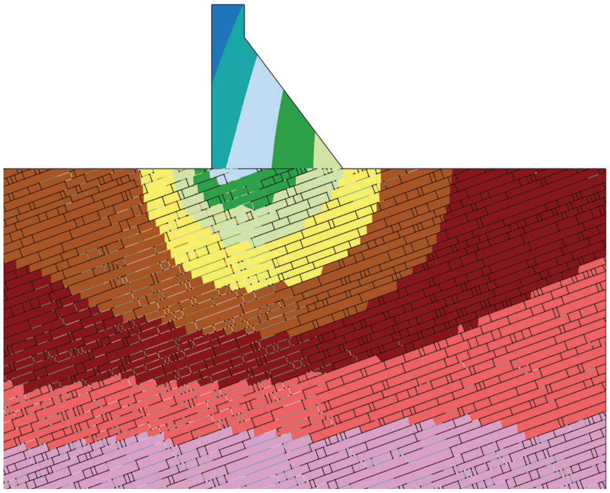
The **Universal Distinct Element Code** (*UDEC*) is two-dimensional numerical software that simulates the response of loading on materials containing multiple, intersecting joint structures.

The material is represented as an assemblage of discrete blocks while the joints are treated as boundary conditions between blocks. Large block displacements along the joints and rotations of blocks can occur. As such, *UDEC* can model complex, non-linear behaviors.

Models may contain a mixture of rigid and deformable blocks. Deformable blocks are meshed using finite-difference zones, with each zone behaving according to its material model (e.g., elastic, Mohr-Coulomb, modified Cam-Clay, etc.) Joint models and properties also can be assigned separately to individual, or sets of discontinuities that govern their relative motion in both the normal and shear directions.

*UDEC* is different from conventional numerical programs in the way the model geometry is created. A single block or domain is created first, with a size that encompasses the physical region being analyzed. Then, this block is cut into smaller blocks whose boundaries represent both geologic features, engineered structures or construction geometry.

> "Because it is not limited to a particular type of problem or initial condition, *UDEC* may be applied to a wide variety of physical behaviors."



This cutting process is termed collectively as joint generation. Joints can be defined through commands, simple shapes, coordinate tables, a statistical joint generator and a Voronoi tessellation generator.

**WHAT IS THE DIFFERENCE BETWEEN DISTINCT ELEMENT AND DISCRETE ELEMENT?**

Many finite element, boundary element and finite difference software have interface elements or "slide lines" that enable them to model a discontinuous material to some extent. However, their formulation is usually restricted in one or more of the following ways:

1. The logic may break down when many intersecting interfaces are used;
2. There may not be an automatic scheme for recognizing new contacts; and
3. The formulation may be limited to small displacements and/or rotation.

The term "discrete element method" applies to software only if it:

1. Allows finite displacements and rotations of discrete bodies, including complete detachment; and
2. Recognizes new contacts automatically as the calculation progresses.

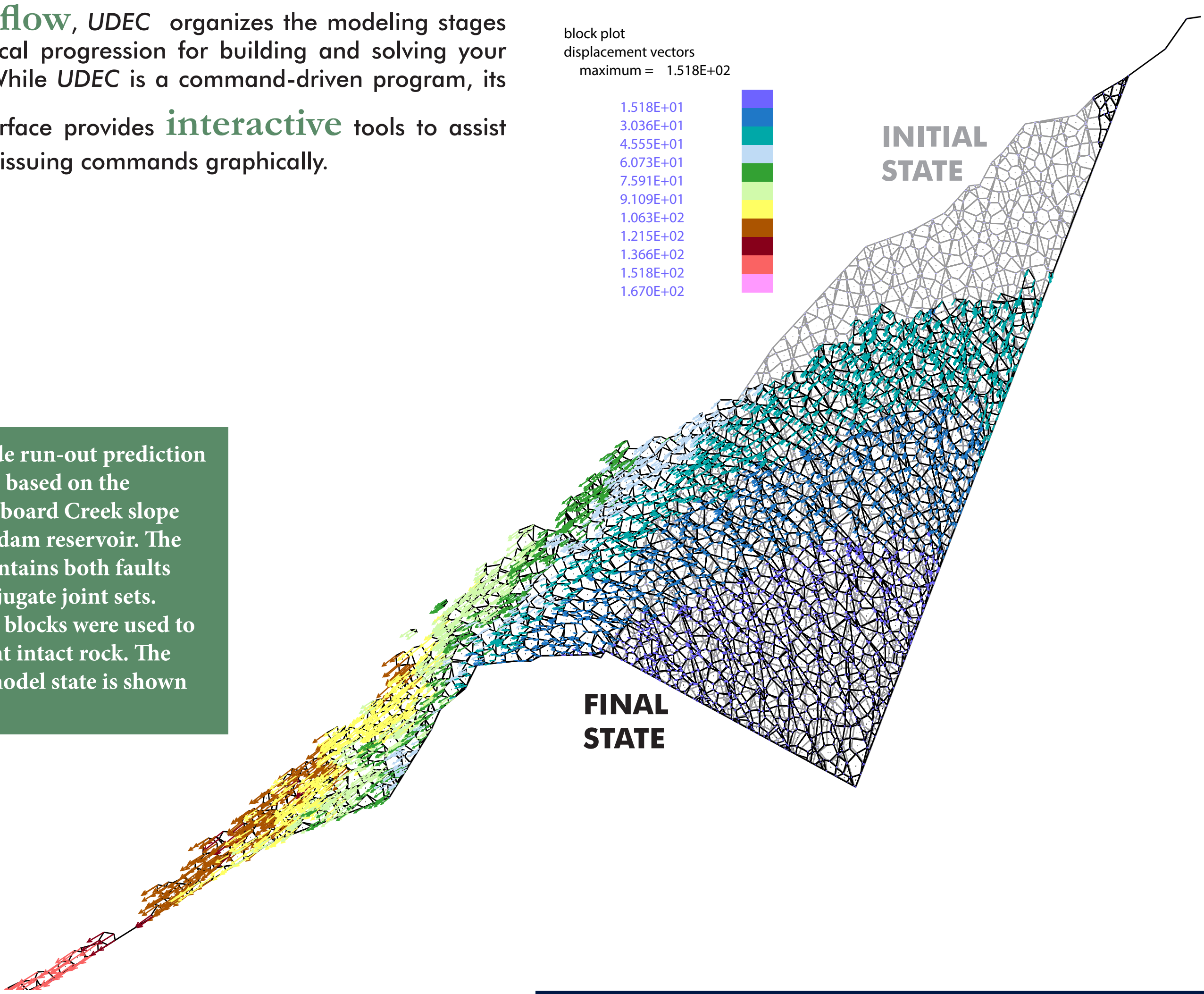
Without the first attribute, the model cannot reproduce some important mechanisms in a discontinuous medium; without the second, the model is limited to small numbers of bodies for which the interactions are known in advance.

The term **"distinct element method"** was coined by Cundall and Strack (1979) to refer to the particular discrete element scheme that uses deformable contacts and an explicit, time-domain solution of the original equations of motion.

*UDEC*'s user interface provides a complete **interactive modeling environment**, project management facilities, a built-in library of materials, easy specification of boundary conditions and structural elements, movie pre-processing, extensive plotting capabilities and run-time

UDEC plot showing displacements and joint shear displacements of a 100-m high gravity dam subjected to water reservoir forces and undergoing an earthquake.

monitoring of results. Designed to **facilitate your work flow**, *UDEC* organizes the modeling stages in a logical progression for building and solving your model. While *UDEC* is a command-driven program, its user interface provides **interactive** tools to assist you with issuing commands graphically.




Rockslide run-out prediction example based on the Checkerboard Creek slope above a dam reservoir. The slope contains both faults and conjugate joint sets. Voronoi blocks were used to represent intact rock. The initial model state is shown in gray.

Slope weathered rocks and prominent jointing.

Itasca Consulting Group, Inc. is an Itasca International company. Itasca is an engineering consulting and software development company founded in Minneapolis, Minnesota with 16 offices worldwide. Itasca specializes in solving complex geomechanical, hydrogeological and microseismic issues in civil, manufacturing, mining, oil & gas and power generation.

Itasca's software are among the most widely used and respected tools available. Use of numerical simulation software is a core part of our consulting work and our software development benefits from the dynamic interplay between Itasca's practical consulting and cutting-edge contract research activities.

# FEATURES

### MULTITHREADED

In order to take advantage of multiple cores, the main calculations in *UDEC* have been modified to run on multiple CPU threads, providing much faster model runs in most cases. The multithreading has been implemented for the following calculations:

- Zone stress/strain calculations;
- Contact force/displacement calculations; and
- Fluid-flow calculations.

### 64-BIT VERSION

*UDEC*'s new 64-bit version eliminates the memory limitations and much larger models, limited only by the physical memory in the computer, can be run on a Windows 64-bit operating system.

### CONSTRUCTIONS JOINTS

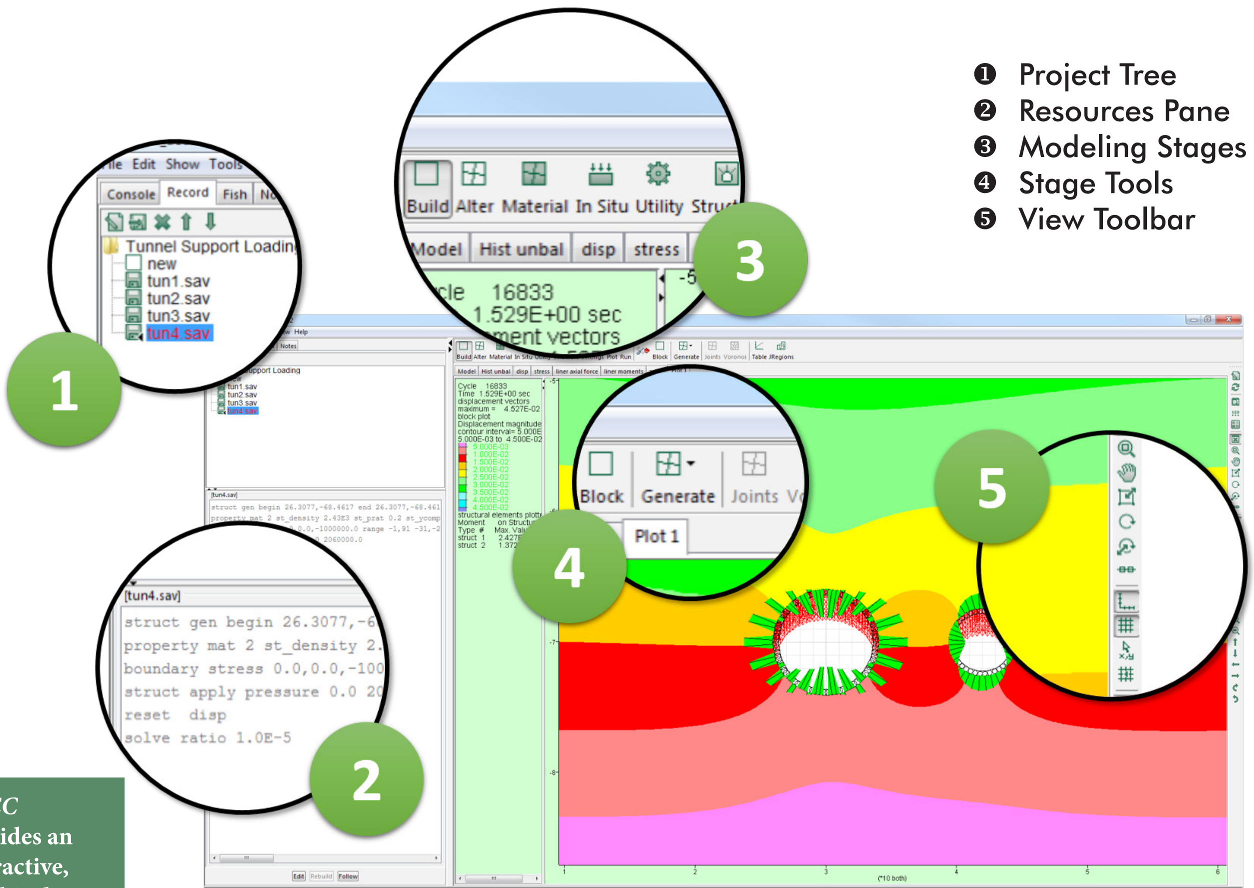
Any joint can be welded (joined) together to create construction joints so that they no longer influence the model behavior. They are generally used to define excavation boundaries, control zone density and create discrete joints that terminate inside intact material.

### STATISTICAL JOINT-SET GENERATOR

The joint generator in *UDEC* defines a joint-set in terms of its angle, gap length, spacing, trace length and origin. In addition to a mean value, the maximum random uniform deviation from the mean can be specified. Joint generation can be limited to selected portions of the model by defining a range or hiding blocks. By default, all joint segments terminating within a block will be deleted once the blocks are zoned or model calculations begin. Embedded joints can be created using continuous joints with intact rock properties intermittently along the joint length or using construction joints to weld blocks together.

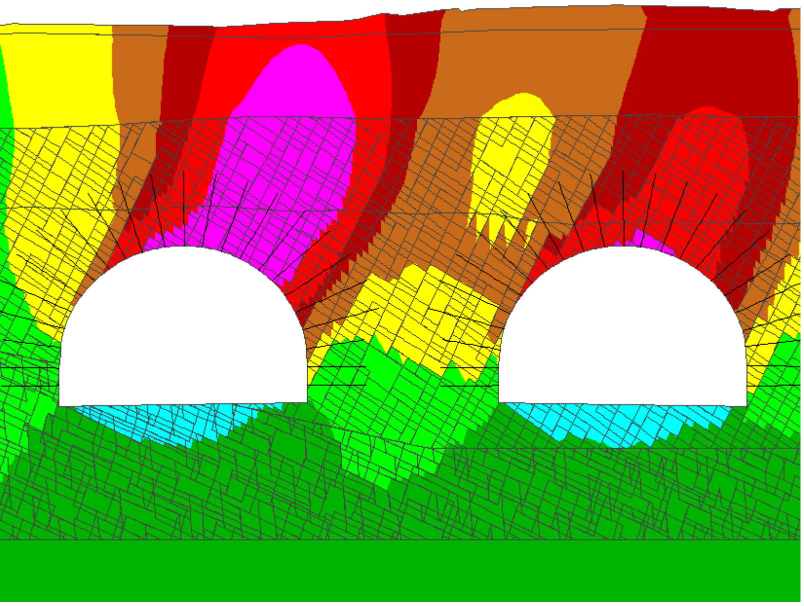
### VORONOI TESSELLATION GENERATOR

Voronoi tessellation creates randomly sized polygonal or triangular blocks. One or more blocks in a *UDEC* model can be subdivided into Voronoi sub-blocks of arbitrary size. This joint generator is useful to simulate crack propagation; "fracturing" occurs when the joint strength between Voronoi blocks is exceeded. Such a model often is referred to as a *UDEC*-DM (damage model) and used to simulate intact rock undergoing micro-damage.




❶ Project Tree
❷ Resources Pane
❸ Modeling Stages
❹ Stage Tools
❺ View Toolbar

*UDEC* provides an interactive, graphical user interface.



Vertical displacement contours for a *UDEC* model of twin, parallel road tunnels excavated in poor to fair quality rock.

The Voronoi algorithm begins by distributing points randomly within the tessellation region. The interior points then are allowed to move. An iteration procedure moves the points; the higher the number of iterations, the more uniform the spacing between points will be. Next, triangles are created between all points. Finally, the Voronoi polygons are created by constructing perpendicular bisectors of all triangles that share a common side. The polygons are truncated at the boundaries of the tessellation region.

**AUTOMATIC ZONING**

A model may contain both rigid and deformable blocks at the same time. Blocks are rigid by default, but can be made deformable by discretized them automatically or manually into triangular finite-difference zones. This allows simulation of soils, intact rock, engineered materials and rock masses as a continuum using *UDEC*.

**SOLVE ELASTIC**

Automatically sets joints and zone constitutive models to infinite strength for initial model equilibrium. This speeds-up modeling time and prevents artificial plastic deformations due to numerical shock. Afterwards the model is solved again with the strength values specified.

**SOLVE RELAX**

Automatically reduces the forces on the inside of an excavation gradually to avoid dynamic failure in zones that would not normally fail under the static stresses caused by the excavation. SOLVE relax can also be used to reduce the boundary forces on the internal boundary of an excavation down to a prescribed level, to simulate the 3D effect of a tunnel advance. A ground reaction curve can also be generated automatically.

**SOLVE FOS**

*UDEC* provides a manual or automatic factor-of-safety solution using Shear Strength Reduction (SSR) that can be used for stability analyses of models. The SSR progressively reduces the shear strength of the material to bring the slope to a state of limiting equilibrium. The automated method can be applied to the Mohr-Coulomb, ubiquitous-joint and Hoek-Brown material models. The Coulomb joint model and certain strength properties for structural elements also can be incorporated into the automated factor-of-safety calculation. Manually, the SSR approach can be applied to essentially any material failure model to evaluate a factor of safety based upon the reduction of a specified strength property or property group.

**ENERGY CALCULATION**

When the outer boundary of the model is free to deform (stress boundary or boundary-element), energy changes can be measured for a *UDEC* model (intact rock, the joints and for the work done on boundaries). Energy components calculated include:

- Total boundary loading work;
- Potential energy;
- Kinetic energy;
- Damped energy;
- Strain energy (stored in the rock mass);
- Viscous boundary work (dynamic); and
- Plastic work.

**BUILT-IN SCRIPTING LANGUAGE**

*FISH* is a scripting language embedded within *UDEC* that allows you to create new model variables, customized functionality and interact with the model. *FISH* was developed in response to requests from users who wanted to do things with Itasca software that were either difficult or impossible to do with existing program structures. Rather than incorporate many new and specialized features into the standard software, an embedded language was provided so that users could write their own functions. For example, *FISH* functions can be written to:

- Prescribed unusual property distributions within the model;
- Calculate, plot and print user-defined variables;
- Implement custom model geometries and joint generators;
- Simulate numerical tests of servo-control material strength testing (e.g., UCS, triaxial, shear);
- Specify unusual boundary conditions (variations in time and space); and
- Automate parametric and sensitivity studies.

## MOMENT-THRUST DIAGRAMS

Moment-Thrust yield envelopes can be plotted directly in *UDEC*. These diagrams illustrate the maximum force that can be applied to a typical section for various eccentricities and are commonly used in the design of concrete columns.

## 3D SUPPORT

Although *UDEC* is a two-dimensional program, three-dimensional support can be simulated by averaging the effect in three-dimensions over the distance between the elements using a linear scaling of material properties in the out-of-plane model direction for a regularly spaced pattern of structural elements for reinforcement, cables, rockbolts, beams and supports.

## HISTORIES

Histories are recordings of any one of 55 pre-selected variables (e.g., pore pressure, centroids, stresses, temperature, displacements, etc.) at a user-defined location (some block, contact, zone or gridpoint). Histories of *FISH* function values can also be recorded. By default, histories are recorded every ten calculation cycles, but this can be selected by the user. A time history of as many as 500 variables can be made in one run.

Histories may be printed, plotted (versus time, steps or another history), converted to Tables and exported as an ASCII data file.

| | MODEL | REPRESENTATIVE MATERIAL | EXAMPLE APPLICATION |
|---|---|---|---|
| | Null | void | holes, excavations, regions in which material will be added at later stage |
| ELASTIC | Isotropic | homogeneous, isotropic continuum; linear stress-strain behavior | manufactured materials (e.g., steel) loaded below strength limit; factor-of-safety calculation |
| | Transversely Isotropic | thinly laminated material exhibiting elastic anisotropy (e.g., slate) | laminated materials loaded below strength limit |
| PLASTIC | Drucker-Pager | limited application; soft clays with low friction | common model for comparison to implicit finite-element programs |
| | Mohr-Coulomb | loose and cemented granular materials; soils, rock, concrete | general soil or rock mechanics (e.g., slope stability and underground excavation) |
| | Ubiquitous-Joint | thinly laminated material exhibiting strength anisotropy (e.g., slate) | excavation in closely bedded strata |
| | Strain-Hardening/Softening | granular materials that exhibit nonlinear material hardening or softening | studies in post-failure (e.g., progressive collapse, yielding pillar, caving) |
| | Bilinear Strain-Hardening/Softening Ubiquitous-Joint | laminated materials that exhibit nonlinear material hardening or softening | studies in post-failure of laminated materials |
| | Double-Yield | lightly cemented granular material in which pressure causes permanent volume decrease | hydraulically placed backfill |
| | Modified Cam-clay | materials for which deformability and shear strength are a function of volume change | geotechnical construction on clay |
| | Hoek-Brown | isotropic rock material | geotechnical construction in rock |
| | Modified Hoek-Brown | isotropic rock material | geotechnical construction in rock including factor-of-safety calculation |
| | Cysoil (cap-yield) | soils that exhibit decreasing stiffness as plastic strains develop | geotechnical construction in soft soils |
| | Simplified Cysoil | simplified version of Cysoil model to simulate hyperbolic strain-strain behavior | geotechnical construction in soft soils |

**All of the above material models (for deformable, zoned blocks) come standard.**

*many special features come standard*

# Structural Elements
# Fluid & Gas Flow in Joints
# Thermal
# Dynamic

## ZONE CONSTITUTIVE MATERIAL MODELS

The constitutive models provided in *UDEC* are arranged into null, elastic and plastic model groups. Input parameters to all of these built-in models can be entered directly as commands, interactively using the GUI or controlled via *FISH* to modify the behavior of the models.

The following features are included as part of the standard *UDEC* license.

## STRUCTURAL ELEMENTS

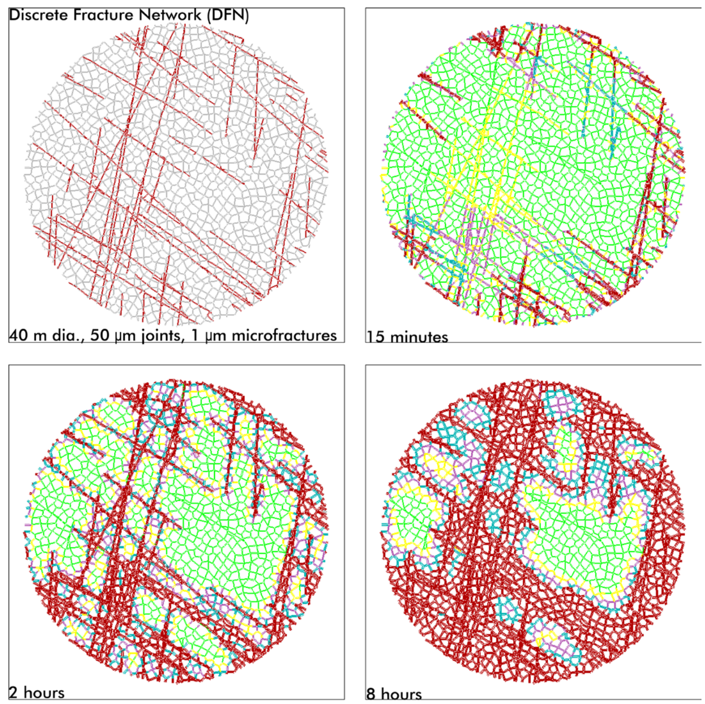
Provide the ability to model support in *UDEC*. Reinforcement is modeled using cables that may be part of a local or global reinforcement model. Surface support is provided through beam elements (which may represent concrete lining, shotcrete, steel sets, etc.) and one-dimensional support members (which may be hydraulic or wooden props or packs).

## FLUID AND GAS FRACTURE FLOW

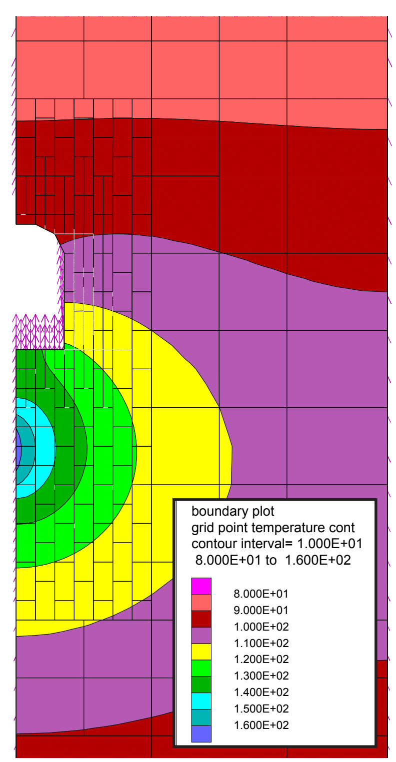
*UDEC* can be used to model fluid or gas flow through the fractures of a system of impermeable blocks. A fully coupled mechanical-hydraulic analysis is performed in which fracture conductivity is dependent on mechanical deformation, and in return joint water pressures affect the mechanical computations. Confined flow, transient flow, two-phase flow and flow with a free surface can be modeled using *UDEC*.

## THERMAL

*UDEC*'s thermal capability simulates transient heat conduction in materials and the development of thermally induced displacements and stresses. Heat transfer is modeled as conduction. Several different thermal boundary conditions may be imposed. Any mechanical model may be used with the thermal model. Heat sources may be inserted into the material and may be made to decay with time. Implicit and explicit calculation schemes are available and may be interchanged. The




**Evolution of pore pressures in rock mass for a *UDEC* circular domain model.**




**Temperature contours and displacement vectors for a coupled thermal-mechanical model of a waste repository.**

# OPTIONS

thermal analysis may be coupled to the mechanical and the fluid calculations.

**DYNAMIC**

*UDEC* can be used to simulate dynamic models for seismic, blasting and earthquake analyses. Absorbing (quiet) and free-field boundaries and seismic wave input are all available. The dynamic formulation can be coupled to structural element, fluid flow and thermal models.

Optional features are specialized modules that can be added to *UDEC* at an additional cost.

**BARTON-BANDIS JOINT MODEL**

The Barton-Bandis joint model utilizes a series of empirical relations for joint normal behavior and joint shear behavior based on the effects of surface roughness on discontinuity deformation and strength as decribed by Barton (1982) and Bandis et al. (1985). The Barton-Bandis joint model encompasses the following features.

**Joint Shear Behavior**

- Dilation as a function of normal stress and shear displacement.
- Joint damage due to post-peak shear.
- Reduced secondary peak shear upon post-peak shear reversal.

**Joint Normal Behavior**

- Hyperbolic stress-displacement path.
- Hysteresis due to successive load/unload cycles.
- Normal stiffness increase due to successive load/unload cycles.
- Normal stiffness change due to surface mismatch caused by shear displacement.
- Hydraulic aperture calculation based on joint closure and joint roughness.

**CREEP MATERIAL MODELS**

The creep option can be used to simulate the behavior of materials that exhibit creep (i.e., time-dependent material behavior). The major difference between creep and other constitutive models is the concept of problem time in the simulation. For creep runs, the problem time and timestep represent real time, while for static analyses (in the other constitutive models), the timestep is an artificial quantity, used only as a means of stepping to a steady-state condition. The timestep may be set by the user to a constant value, or controlled by *UDEC* to change automatically. If the timestep is changed automatically, it can be decreased whenever the maximum unbalanced force exceeds some threshold, and increased whenever it goes below some other level. For some of the creep models available, the creep rate is temperature-dependent. Temperatures may either be specified as a model property or they may be calculated during cycling using the thermal mode of the code. For either case, a temperature gradient may also be specified.

Eight creep models have been implemented in *UDEC*.

- Classical viscoelastic (Maxwell) model
- Burgers viscoelastic model
- WIPP viscoelastic model
- Burgers viscoplastic model
- Two-component Power law
- Power law viscoplastic model
- WIPP viscoplastic model; and
- Crushed salt model.

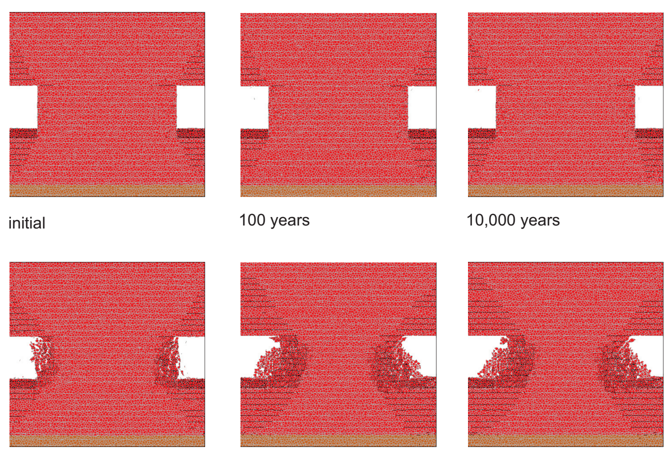
**USER-DEFINED MODELS (UDM)**

With this option, users may create their own contact or zone constitutive model for use in *UDEC*. The model must be written in C++ (CPP) and compiled as a DLL (dynamic link library) file using Microsoft Visual Studio 2010. The DLL then can be loaded whenever it is needed. The main function of the zone model is to return new stresses, given strain increments. The main function of the contact constitutive model is to return focres given displacements. However, the model must also provide other information (such as name of the model and material property names) and describe certain additional details about how the model interacts with the code. It is assumed that the user has a working knowledge of the C++ programming language.

UDMs may be used in other Itasca software provided this option is also available for that software.

DLL models can be obtained from an Itasca website devoted specifically to model development and exchange: www.itasca-udm.com

***UDEC*** **model showing the evolution of waste storage cavern and pillar damage assuming a glacial cycle and the effects of gas and pore pressure.**



# SYSTEM

## SYSTEM REQUIREMENTS

**Operating System:**
Windows XP (32/64), Windows Vista (32/64), Windows 7 (32/64), Windows 8 (32/64).

**USB Port:**
1 port required for USB local license key.
A TCP/IP connection to a server (with a USB Port) is required for a USB network license.

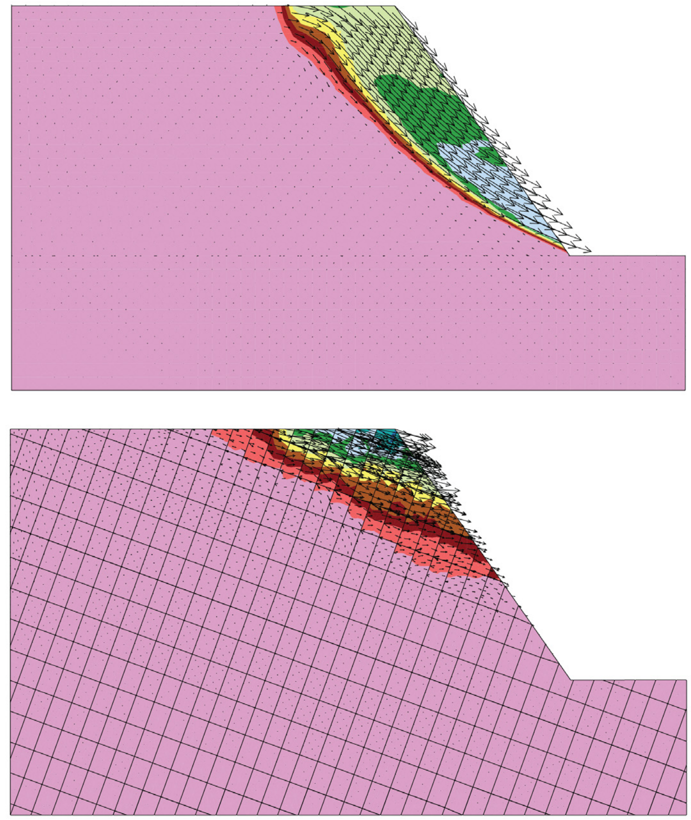
**Processor:**
Any Pentium or AMD equivalent compatible. Intel Pentium i7 or later and is recommended.

**Graphics Card:**
No specific requirement.

**Hard Disk:**
Installation requires 340 MB of free disk space. Save files are approximately proportional to the size of your model in RAM.

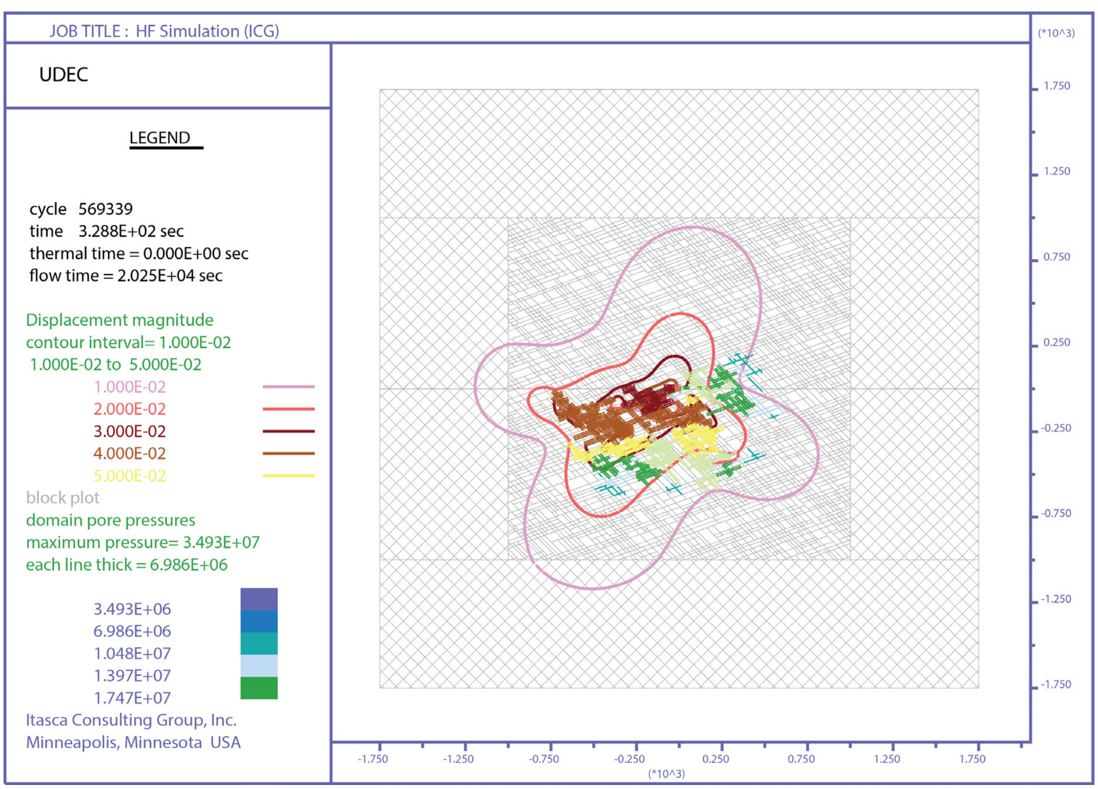
**Random Access Memory (RAM):**
1 GB of RAM memory minimum.

Total RAM is proportional to the size of your model. Total usable memory is limited by 32 bit Windows to approx. 800 MB. The 64-bit version is limited, for all practical purposes, only by the size of available memory hardware.



**In order to ascertain critical failure mechanisms, *UDEC* models can be simulated as a zoned continuum (top) or a blocky discrete system (bottom) or some combination.**

**_UDEC_ model showing DFN pore pressures and block displacement contours after undergoing water injection for over five hours.**



# BENEFITS

**POWERFUL**

- Model large displacements, even collapse.
- Very large models are possible (64-bit).
- Any material behavior can be simulated.
- Thermal, fluid and gas flow and dynamic analyses modes come standard and can be coupled together.
- Deep-scripting using FISH allows you to customize *UDEC* to your analyses.
- Barton-Bandis Joint Model, Creep and C++ User-Defined Models (UDM) are optionally available.

**EFFICIENT**

- Multi-core processing (for no additional cost).
- Optimized solution calculations.
- Customize material behaviors efficiently using optional C++ UDM.

**RELIABLE**

- Realistic physical solutions.
- Natural evolution of failure.
- Transparent methodology.
- Best-documented simulation software.
- Strong software support.
- Backed by an experienced, international team of consulting engineers and scientists.

**PROVEN**

- Tested against analytical solutions, laboratory testing and field measurements.
- Used by Itasca's own consulting engineers.
- 1000's of publications.
- Used worldwide by industry, universities and government agencies.

**FLEXIBLE**

- General by design.
- Interactive graphical user interface simplifies model construction and runs.
- Most model parameters are accessible by the built-in *FISH* scripting language.
- Import and export using ASCII.
- Human-readable data files.
- Define your own material behaviors using the optional C++ UDM.
- License is portable between computers and users.
- Network license(s) available.
- Software leases available.

ITASCA
Consulting Group, Inc.

111 Third Avenue South
Suite 450
Minneapolis, Minnesota USA 55401

Tel +1 (612) 371-4711
Fax +1 (612) 371-4717
www.itascacg.com
software@itascacg.com

Follow us online:

www.itascacg.com