1 INTRODUCTION

1.1 Overview

The Universal Distinct Element Code (UDEC) is a two-dimensional numerical program based on the distinct element method for discontinuum modeling. UDEC simulates the response of discontinuous media (such as a jointed rock mass) subjected to either static or dynamic loading. The discontinuous medium is represented as an assemblage of discrete blocks. The discontinuities are treated as boundary conditions between blocks; large displacements along discontinuities and rotations of blocks are allowed. Individual blocks behave as either rigid or deformable material. Deformable blocks are subdivided into a mesh of finite-difference elements, and each element responds according to a prescribed linear or nonlinear stress-strain law. The relative motion of the discontinuities is also governed by linear or nonlinear force-displacement relations for movement in both the normal and shear directions. UDEC has several built-in material behavior models, for both the intact blocks and the discontinuities, which permit the simulation of response representative of discontinuous geologic (or similar) materials. UDEC is based on a “Lagrangian” calculation scheme that is well-suited to model the large movements and deformations of a blocky system.

UDEC also contains the powerful built-in programming language FISH (short for FLACish; FISH was originally developed for our two-dimensional, finite-difference, continuum program: FLAC). With FISH, you can write your own functions to extend UDEC’s usefulness. FISH offers a unique capability to UDEC users who wish to tailor analyses to suit specific needs.

The formulation and development of the distinct element method embodied in UDEC has progressed for a period of 40 years, beginning with the initial presentation by Cundall (1971). In 1985, Dr. Cundall and Itasca staff adapted UDEC specifically to perform engineering calculations on an IBM-compatible microcomputer. The software is designed for high-speed computation of models containing several thousand blocks. With the advancements in floating-point operation speed and the ability to install additional RAM at low cost, increasingly larger problems can be solved with UDEC. For example, UDEC can solve a model containing up to 3000 deformable blocks with 8 degrees of freedom per block on a personal computer with only 12 MB RAM. The solution speed for a model of this size is roughly 6000 calculation steps per minute on a 2.9 GHz Pentium i7-870 computer.* The calculation speed is essentially a linear function of the number of blocks in a model, and the number of blocks is a linear function of the available RAM on the computer. (See Table 2.1.)

For typical models, consisting of roughly 1500 rigid blocks (or 500 deformable blocks) or fewer, the explicit solution scheme in UDEC requires approximately 2000 to 4000 steps to reach a solved state.** For example, a 500 deformable block model run on the computer described above would require roughly 6 seconds to perform 4000 calculation steps. Consequently, typical engineering

* See Section 5 for a comparison of UDEC runtimes on various computer systems.

** This will vary depending on the amount of relative motion that occurs between blocks. The explicit solution scheme is explained in Section 1 in Theory and Background.
problems involving several hundred blocks and multiple solution stages can be solved with UDEC on a microcomputer in a matter of minutes or a few hours.

A comparison of UDEC to other numerical methods, a description of general features and new updates in UDEC Version 5.0, and a discussion of fields of application are provided in the following sections. If you wish to try UDEC right away, the program installation instructions and a simple tutorial are provided in Section 2.
1.2 Comparison with Other Methods

Common questions asked about UDEC: Is UDEC a distinct element or discrete element program? What is the difference, and what is UDEC’s relation to other programs? We provide a definition here which we hope will clarify these matters.

Many finite element, boundary element and Lagrangian finite difference programs 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. First, the logic may break down when many intersecting interfaces are used; second, there may not be an automatic scheme for recognizing new contacts; and third, the formulation may be limited to small displacements and/or rotation. Such programs are usually adapted from existing continuum programs.

The name “discrete element method” applies to a computer program only if it

(a) allows finite displacements and rotations of discrete bodies, including complete detachment; and

(b) recognizes new contacts automatically as the calculation progresses.

Without the first attribute, a program cannot reproduce some important mechanisms in a discontinuous medium; without the second, the program 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 (not the transformed, modal equations).

There are four main classes of computer programs that conform to the proposed definition of a discrete element method. (The classes and representative programs are discussed in more detail in Section 1 in Theory and Background.)

1. Distinct Element Programs – These programs use explicit time-marching to solve the equations of motion directly. Bodies may be rigid or deformable (by subdivision into elements); contacts are deformable. UDEC falls into this category.

2. Modal Methods – The method is similar to the distinct element method in the case of rigid bodies but, for deformable bodies, modal superposition is used.

3. Discontinuous Deformation Analysis – Contacts are rigid, and bodies may be rigid or deformable. The condition of no-interpenetration is achieved by an iteration scheme; the body deformability comes from superposition of strain modes.

4. Momentum-Exchange Methods – Both the contacts and the bodies are rigid: momentum is exchanged between two contacting bodies during an instantaneous collision. Frictional sliding can be represented.
There are several published schemes that appear to resemble discrete element methods, but which are different in character or are lacking one or more essential ingredients. For example, many publications are concerned with the stability of one or more rigid bodies, using the *limit equilibrium method* (e.g., Hoek 1973, Warburton 1981, Goodman and Shi 1985, and Lin and Fairhurst 1988). This method computes the static force equilibrium of the bodies and does not address the changes in force distribution that accompany displacements of the bodies.
1.3  General Features

*UDEC* is primarily intended for analysis in rock engineering projects, ranging from studies of the progressive failure of rock slopes to evaluations of the influence of rock joints, faults, bedding planes, etc. on underground excavations and rock foundations. *UDEC* is ideally suited to study potential modes of failure directly related to the presence of discontinuous features.

The program can best be used when the geologic structure is fairly well-defined (e.g., from observation or geologic mapping). Both manual and automatic joint generators are built into *UDEC* to create individual and sets of discontinuities which represent (in two dimensions) jointed structure in a rock mass. A wide variety of joint patterns can be generated in the model. A screen-plotting facility allows the user to instantly view the joint pattern. Adjustments can easily be made before the final pattern is selected for analysis.

Different representations of joint material behavior are also available. The basic model is the Coulomb slip criterion, which assigns elastic stiffness, frictional, cohesive and tensile strengths, and dilation characteristics to a joint. A modification to this model is the inclusion of displacement weakening as a result of loss in cohesive and tensile strength at the onset of shear failure. A more complex model, the continuously yielding joint model, is also available and simulates continuous weakening behavior as a function of accumulated plastic shear displacement. As an optional feature, the Barton-Bandis joint model is also available at an additional cost. Joint models and properties can be assigned separately to individual, or sets of, discontinuities in a *UDEC* model. Note that the geometric roughness of a joint is represented via the joint material model, even though the plot of discontinuities shows the joint as a straight-line segment.

Blocks in *UDEC* can be either rigid or deformable. There are seven built-in material models for deformable blocks, ranging from the “null” block material (which represents holes – excavations), to the shear and volumetric yielding models (which include strain-hardening/softening behavior and represent nonlinear, irreversible shear failure and compaction). Thus, blocks can be used to simulate backfill and soil materials as well as intact rock.

The basic formulation for *UDEC* assumes a two-dimensional plane-strain state. This condition is associated with long structures or excavations with constant cross-section acted on by loads in the plane of the cross section. Discontinuities, therefore, are considered as planar features oriented normal to the plane of analysis. In addition, *UDEC* offers a plane-stress option, in which the stresses normal to the cross section are zero. This is encountered, for example, in masonry structures loaded only in the plane of the structure. For plane-strain analysis, blocks may exhibit plastic yield, and failure can occur in the out-of-plane direction if the out-of-plane stress, $\sigma_{zz}$, becomes a major or minor principal stress.

The explicit solution algorithm in *UDEC* permits either dynamic or static analysis. For dynamic calculations, user-specified velocity or stress waves can be input directly to the model either as an exterior boundary condition or interior excitation to the model. A library of simple dynamic wave forms is also available for input. *UDEC* contains non-reflecting and free-field boundary conditions for dynamic analysis.

Both stress (force) and fixed displacement (zero velocity) boundary conditions are available for static analysis. Boundary conditions may be different at different locations. In addition, a boundary
element model is available to link to the UDEC model to simulate the boundary as an infinite elastic body. A half-plane solution is also available to represent the effect of a free surface.

UDEC is able to simulate the flow of fluid through the discontinuities and voids in the model. At present, blocks are impermeable. A fully coupled mechanical-hydraulic analysis is performed in which fracture conductivity is dependent on mechanical deformation of the joint aperture; conversely, joint water pressures affect the mechanical behavior. Flow is idealized as laminar viscous flow between parallel plates. A visco-plastic flow model is also available to simulate flow of cement grout in the joints.

Structural element logic is implemented to simulate rock reinforcement and surface support in the model. Reinforcement includes point-anchored and fully grouted cables and bolts. Surface support simulates structures such as shotcrete, concrete linings, ribs and other forms of tunnel support, and stabilizing lining for open cuts or natural slopes.

There is also a thermal model available in UDEC. This model simulates the transient flux of heat in materials and the subsequent development of thermally induced stresses. The heat flux is modeled by either isotropic or anisotropic conduction. Heat sources can be added, and can be made to decay exponentially with time.

UDEC contains a powerful built-in programming language, FISH, which enables the user to define new variables and functions. FISH is a compiler; programs entered via a UDEC data file are translated into a list of instructions stored in UDEC’s memory space. These are executed whenever a FISH function is invoked. FISH permits:

- user-prescribed property variations in the grid (e.g., nonlinear increase in modulus with depth);
- plotting and printing of user-defined variables (custom-designed plots);
- implementation of special joint generators;
- servo-control of numerical tests;
- specification of unusual boundary conditions; variations in time and space; and
- automation of parameter studies.

An extensive plotting facility is built directly into UDEC. This allows the user to generate plots of virtually any problem variable in the UDEC model, either on the screen or a hardcopy device. Several variables can be plotted as overlays on a plot of the model or, alternatively, histories of the change in a variable as a function of calculation step can be plotted. The history plots are especially helpful in ascertaining when an equilibrium state or failure state has been reached, and monitoring the change in variables during transient calculations, such as fluid flow in joints. As mentioned previously, plots can be custom-designed via FISH to meet the user’s need.
1.4 What’s New in UDEC 5.0

UDEC 5.0 is faster, and it also has an updated user interface. The new release version of the double-precision version is 30% faster than the previous double-precision version. The user interface has been improved, is easier to use and is more comprehensive. Many new plotting features have been added. A list of improvements follows.

1.4.1 Improvements to the Graphical User Interface

The graphical user interface has been extensively redesigned to be more functional and easier to use, and it now includes all UDEC features available through the command line. Configuration switches have been added to enable the interface to provide comprehensive coverage but only show the features currently in use. In this way, experienced UDEC users will be supported while new users will not be overwhelmed.

1.4.2 Rockbolt Elements

New and improved rockbolt elements have been added to UDEC:

a. Elements include both bending and shear resistance.

b. Local stress-dependent pullout strength is based on the change in stress that has occurred since the bolt was installed.

c. The connection to the zone gridpoints in both the normal and shear directions is via coupling springs.

d. Softening as a function of shear displacement for the coupling spring can be prescribed using user-defined strength tables.

e. The rockbolt may yield in both tension and compression.

f. Rockbolt rupture is based on a user-defined tensile strain criterion.

1.4.3 Structural Liners

The structural liner logic has been improved by the addition of new commands. These commands make it easier to add simple liners in tunnels, and to more precisely control portioning of structural element nodes. The user can also import node placement in the form of tables.

Moment-thrust and thrust-shear plots for structural liners have been added.

New fixed degrees-of-freedom for structural nodes allow lines of symmetry to be used with liners.
1.4.4 Plasticity in Zones

More accurate plasticity solutions are available using nodal mixed discretization. This technique averages the zone strains at common nodes, which leads to a better plasticity solution by reducing the locking effect of lower-order elements. This is particularly useful in cases where `GENERATE quad` cannot be used.

1.4.5 Construction Joints

The `JOINT` command and associated keywords are used for the creation of “fictitious” construction joints. New keywords have been added to allow cracks to be specified as welded construction joints. These cracks are normally used to define excavation boundaries, create internal cracks and provide zone density control. This new feature allows construction joints to be included while minimizing the effect on model behavior. The joined edges created using the new commands are hidden when blocks are plotted. A new `PLOT joined` command has been added to show the joined edges.

1.4.6 New Interactive Help

A compiled HTML help file is now accessible through the graphical user interface. Explanations for all commands from the `Command Reference` and `FISH in UDEC` volumes are available online through the GUI.

1.4.7 Restricted Movement for Rigid Blocks

The centroid velocities of rigid blocks can now be fixed in specified directions. Previously, the centroid velocities could only be fixed in all directions.

1.4.8 Gas Flow in Joints

The joint fluid-flow logic in `UDEC` has been expanded to allow the modeling of gas flow. In gas flow, the bulk modulus of a gas is equal to its pressure.

1.4.9 Built-in FISH Editor

A new `FISH` editor with parameters has been added to the GUI. This makes it easier to include `FISH` functions in GUI project files.

1.4.10 Virtual Models

Several virtual models have been included in the GUI. The virtual models allow the modification of several base classes of geometries, and then generate the `UDEC` commands to build them. Since the geometry is defined outside of `UDEC`, it is easier to modify the geometry graphically.
1.4.11 Expanded Range Control

The ranges available to perform functions in the GUI have been expanded to include all of the UDEC command-line ranges. This adds significant flexibility to model building.

1.4.12 Changes to the SOLVE elastic command

The SOLVE elastic command has been modified to function the same way it does in FLAC. In SOLVE elastic, the model is given high strength properties and cycled until equilibrium is obtained. Then the original properties are restored and the model is again cycled to equilibrium.

1.4.13 New FISH Functions

Reaction-force FISH function have been added to allow the retrieval of the boundary forces generated at fixed velocity boundaries.

FISH functions to retrieve the index strains $\text{exx}$, $\text{exy}$ and $\text{eyy}$ have been added.

1.4.14 New Plotting Commands and Features

Joint normal and shear stress plots have been added.

All vector quantity plots are now displayed with colors scaled to the vector magnitude. This makes it easier to identify areas of high and low activity.

Joint attribute plots are now displayed with colors scaled to the joint attribute magnitude.

Maximum shear strain (SSI) and maximum shear strain rate (SSR) contour plots have been added to UDEC.

Boundary condition plotting has been improved to include fluid settings.

A new displacement magnitude contour plot has been added.

A new velocity magnitude contour plot has been added.

There is a new moment-thrust plot for structural liners.

There is a new thrust-shear plot for structural liners.

1.4.15 Use of Tables for Ground Water Surface

The ability to use a table to define the ground water surface has been added to the PFIX command.
1.4.16 Improvements to Voronoi Block Generator

A new algorithm that reduces Voronoi block-generation time by a factor of approximately 100 has been added.

The Voronoi block-generation range control has been modified to include any range command that can be used to select blocks. Previously, only the JREGION range command was used.

In UDEC 4.0, there was a problem with finely discretized Voronoi block patterns showing a texture or preferred orientation. This texture problem has been eliminated in the new version.

1.4.17 Gridpoint History Control

Gridpoint histories are now selected based on closeness, with preference given to the zone that includes the coordinate. Previously, the user was not able to control which gridpoint was selected for the history in the case where more than one gridpoint was at the same location in the model.

1.4.18 Automatic Loading of Constitutive Models

Zone and joint DLL models are automatically loaded; the MODEL load command is no longer needed. This eliminates the annoyance of having to reload models prior to restoring save files.

1.4.19 Improved SOLVE fos command

The SOLVE fos command now works for the mhoek (Modified Hoek Brown) model and any constitutive model written to support property scaling.

New keywords have been added to allow more user control of the SOLVE fos process. The user can now specify the solve resolution and the bracket limits.

SOLVE fos can also test a single safety factor value and determine whether the model is stable or unstable.

1.4.20 SOLVE relax command

A new SOLVE relax command (similar to the FISH function zonk) has been added. This function is used to slowly reduce boundary forces in excavations to zero or any other specified value. The new command can also be used to automatically create a force ratio history table (similar to a ground reaction plot).

1.4.21 Expanded Default Joint Behavior

JMODEL and JOINT models are now used as the default constitutive models for newly formed contacts.
1.4.22  Windows File Open Dialog in Command-Line Mode

Using the command line interface, the CALL, RESTORE and SAVE commands will open a Windows file open dialog for visual file selection, without specifying a file name. This allows the user to open a file without having to type the file name and path.

1.4.23  Automatic Far-Field Dynamic Boundary

The dynamic far-field boundary can now be generated automatically using the BOUNDARY ff command. This sets up the far-field zones, transfers properties and constitutive models from the zones to the far field, and then cycles the far field to equilibrium. Any dynamic boundary conditions applied after the BOUNDARY ff command are automatically transferred to the far field.
1.5 Fields of Application

UDEC was originally developed to perform stability analysis of jointed rock slopes. The discontinuum formulation for rigid blocks and the explicit time-marching solution of the full equations of motion (including inertial terms) facilitate the analysis of progressive, large-scale movements of slopes in blocky rock.

UDEC has been applied most often in studies related to mining engineering. Both static and dynamic analyses for deep underground mined openings have been performed. Fault-slip induced failure around excavations is one example of analyses conducted with UDEC. Blasting effects have been studied by applying dynamic stress or velocity waves at model boundaries. Research in the area of fault-slip induced seismicity has also been conducted by use of the continuously yielding joint model. Structural elements have been employed to simulate various rock reinforcement systems, such as grouted rockbolting and shotcrete.

UDEC has also been applied in the fields of underground construction and deep underground storage of high-level radioactive waste. Through the use of the thermal model, UDEC has been used to simulate effects of thermal loading in connection with buried nuclear waste.

UDEC has been used to a limited extent as a computational design tool. However, the program is better-suited to investigate potential failure mechanisms associated with the response of a jointed rock mass. The nature of a jointed rock mass is that it is a “data-limited” system (i.e., the internal structure and stress state are, in large part, unknown and unknowable). Thus, it is impossible, in principle, to make a complete model of a rock mass system. Also, since UDEC is a two-dimensional program, the three-dimensional geometry of a joint structure cannot be represented except for special orientations. Nevertheless, an understanding of the response of underground openings in jointed rock can be achieved at a phenomenological level using UDEC. This methodology seeks to improve the engineering understanding of the relative impact of various phenomena on the rock mechanics design. In this way, the engineer can anticipate potential problem areas by identifying mechanisms that may lead to unacceptable states of deformation/loading (or failure) of the underground opening. The paper by Starfield and Cundall (1988) is recommended as a guide for using UDEC in rock engineering projects.

Section 6 presents a bibliography of published reports on the application of UDEC in the fields of mining and underground engineering. Additionally, UDEC has potential for application in other fields of engineering, as discussed below and listed in Section 6.

The fluid flow model in UDEC has been used for studies of fluid penetration from unlined pressure tunnels, storage of natural gas in rock caverns, and flow through jointed rock foundations beneath gravity dams. The visco-plastic formulation of the flow model has been employed to simulate flow of cement grout. Examples of these applications are provided in the Verification Problems.

UDEC also has the potential for application in studies related to earthquake engineering. For example, the program may be used to provide explanations of phenomena related to fault movement.

Another area of application is the study of the behavior of reinforced concrete. Although UDEC does not include a model to simulate dynamic fracture growth through deformable blocks, progressive failure associated with crack propagation and spalling can be simulated by the breaking
of preexisting bonds between blocks which remain intact. A special joint generator which creates Voronoi polygons within the UDEC model is employed (Lorig and Cundall 1987).

It is important to note that UDEC is not a suitable program for particle flow studies or dynamic analysis of cratering phenomena in which the interaction of many microparticles is important. For these studies, programs such as the Itasca code PFC\textsuperscript{2D} are recommended. The new cell space detection logic in UDEC does allow for the analysis of flying blocks, but is not efficient for problems with thousands of tiny particles. Finally, in order to evaluate the importance of three-dimensional geometry on response of a system, a three-dimensional numerical program, such as the Itasca code 3DEC or PFC\textsuperscript{3D}, is required.
1.6 Guide to the Manual

The *UDEC* Version 5.0 manual consists of nine volumes.

**User's Guide**

**Section 1** Introduction

This section introduces you to *UDEC* and its capabilities and features.

**Section 2** Getting Started

If you are just beginning to use *UDEC*, or are only an occasional user, we recommend that you read **Section 2**. This section provides instructions on installation and operation of the program, as well as recommended procedures for running simple *UDEC* analyses.

**Section 3** Problem Solving with *UDEC*

**Section 3** is a guide to practical problem solving. Turn to this section once you are familiar with the program operation. Each step in a *UDEC* analysis is discussed in detail, and advice is given on the most effective procedures to follow when creating, solving and interpreting a *UDEC* model simulation.

**Section 4** *FISH* Beginner’s Guide

**Section 4** provides the new user with an introduction to the *FISH* programming language in *UDEC*. This includes a tutorial on the use of the *FISH* language. *FISH* is described in detail in **Section 1** in the *FISH* volume.

**Section 5** Miscellaneous

Various information is contained in this section, including the *UDEC* runtime benchmark and procedures for reporting errors and requesting technical support.

**Section 6** Bibliography

**Section 6** contains a bibliography of published papers describing some uses of *UDEC*.

**Command Reference**

**Section 1** Command Reference

All the commands that can be entered in the command-driven mode in *UDEC* are described in **Section 1** in the **Command Reference**.
**FISH in UDEC**

Section 1  
*FISH Beginner’s Guide*

The *FISH* programming language is described in detail in Section 1 in the *FISH volume*. This includes a tutorial on the use of the *FISH* language.

Section 2  
*FISH Reference*

Section 2 in the *FISH volume* contains a detailed reference to the *FISH* language. All *FISH* statements, variables and functions are explained, and examples are given.

Section 3  
Library of *FISH* Functions

A collection of *FISH* functions is provided for general application in *UDEC*.

Section 4  
Program Guide

*FISH* allows access to some of *UDEC*’s linked-list data structure. This section describes this access.

**Theory and Background**

Section 1  
Background – The 2D Distinct Element Method

The theoretical formulation for *UDEC* is described in detail in Section 1 in Theory and Background.

Section 2  
Factor-of-Safety Calculation

The steps and options for calculating factors of safety are described.

Section 3  
Energy Calculation

The stored and dissipated energy components that can be monitored in *UDEC* are described. An example application is given.

**Constitutive Models**

Section 1  
Block Constitutive Models

The theoretical formulation and implementation of the various block constitutive models are described.

Section 2  
Continuously Yielding Joint Model

The formulation of the continuously yielding joint model is described, and example applications are provided.
Section 3  Barton-Bandis Joint Model

The implementation of the Barton-Bandis joint model in *UDEC* is described. The input commands are given for this option, along with examples to illustrate the application of the model.

Section 4  Writing New Constitutive Models

This section describes the process that allows users to write their own zone and joint models, which are compiled and included as DLLs.

Creep Material Models

Section 1  Creep Material Models

The different creep models (available as an option in *UDEC*) are described, and verification problems are provided.

Special Features – Structures/Fluid Flow/Thermal/Dynamics

Section 1  Structural Elements

This section describes the various structural element models available in *UDEC*.

Section 2  Fluid Flow in Joints

The formulation for the joint fluid-flow model is described, and the various ways to model transient and steady-state flow in joints are illustrated.

Section 3  Thermal Analysis

The thermal analysis facility in *UDEC* is described, and several verification problems that illustrate its application (both with and without interaction with mechanical stress) are presented.

Section 4  Dynamic Analysis

The dynamic analysis facility is described, and considerations for running a dynamic model are provided. Several verification examples are also included in this section.

Verification Problems

This section contains a collection of *UDEC* verification problems. These are tests in which a *UDEC* solution is compared directly to an analytical (i.e., closed-form) solution.

Example Applications

This section contains example applications of *UDEC* that demonstrate the various classes of problems to which *UDEC* may be applied.
1.7 Itasca Consulting Group Inc.

Itasca Consulting Group Inc. is more than a developer and distributor of engineering software. Itasca is a consulting and research firm consisting of a specialized team of civil, geotechnical and mining engineers with an established record in solving problems in many areas:

- Civil Engineering
- Mining Engineering and Energy Resource Recovery
- Nuclear Waste Isolation and Underground Space
- Defense Research
- Software Engineering
- Seismic Engineering
- Groundwater Analysis and Dewatering
- Petroleum Engineering

Itasca was established in 1981 to provide advanced rock mechanics services to the mining industry. Today, Itasca is a multidisciplinary geotechnical firm with 80 professionals in offices worldwide. The corporate headquarters for Itasca is located in Minneapolis, Minnesota. Worldwide offices of Itasca are: Itasca Denver Inc. (Denver, Colorado); Itasca Geomekanik AB (Stockholm, Sweden); Itasca Consultants S.A.S. (Ecully, France); Itasca Consultants GmbH (Gelsenkirchen, Germany); Itasca Consultores S.L. (Llanera, Spain); Itasca S.A. (Santiago, Chile); Itasca Africa Ltd. (Johannesburg, South Africa); Itasca Consulting Canada Inc. (Sudbury, Canada); Itasca Consulting China Ltd. (Wuhan, China); Itasca Houston Inc. (Houston, Texas); Itasca Australia Pty. Ltd. (Melbourne, Australia); ASC (Shrewsbury, United Kingdom); and Itasca India Consulting Pvt. Ltd. (Nagpur, India).

Itasca’s staff members are internationally recognized for their accomplishments in geological, mining, petroleum, seismology and civil engineering projects. Itasca staff consists of geological, mining, hydrological, petroleum and civil engineers who provide a range of comprehensive services such as (1) computational analysis in support of geo-engineering designs, (2) design and performance of field experiments and demonstrations, (3) laboratory characterization of rock properties, (4) data acquisition, analysis and system identification, (5) groundwater modeling, and (6) short courses and instruction in the geomechanics application of computational methods. If you should need assistance in any of these areas, we would be glad to offer our services.
1.8 User Support

We believe that the support Itasca provides to code users is a major reason for the popularity of our software. We encourage you to contact us when you have a modeling question. We will provide a timely response via telephone, electronic mail or fax. General assistance in the installation of UDEC on your computer, plus answers to questions concerning capabilities of the various features of the code, are provided free of charge. Technical assistance for specific user-defined problems can be purchased on an as-needed basis.

If you have a question, or desire technical support, please contact us:

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
Email: software@itascacg.com
Web: www.itascacg.com

We also have a worldwide network of code agents who provide local technical support. Details may be obtained from Itasca.
1.9 References


