Water Science and Engineering, 2009, 2(3): 71-77
doi:10.3882/j.issn.1674-2370.2009.03.007

http://kkb.hhu.edu.cn
e-mail: wse@hhu.edu.cn

Surrounding rock deformation analysis of underground caverns with multi-body finite element method

Wan-jin LIANG*, Chao SU, Fei WANG, Xiao-jun TANG

College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing 210098, P. R. China

Abstract: Discontinuous deformation problems are common in rock engineering. Numerical analysis methods based on system models of the discrete body can better solve these problems. One of the most effective solutions is discontinuous deformation analysis (DDA) method, but the DDA method brings about rock embedding problems when it uses the strain assumption in elastic deformation and adopts virtual springs to simulate the contact problems. The multi-body finite element method (FEM) proposed in this paper can solve the problems of contact and deformation of blocks very well because it integrates the FEM and multi-body system dynamics theory. It is therefore a complete method for solving discontinuous deformation problems through balance equations of the contact surface and for simulating the displacement of whole blocks. In this study, this method was successfully used for deformation analysis of underground caverns in stratified rock. The simulation results indicate that the multi-body FEM can show contact forces and the stress states on contact surfaces better than DDA, and that the results calculated with the multi-body FEM are more consistent with engineering practice than those calculated with DDA method.

Key words: *multi-body finite element method; discontinuous deformation; surrounding rock deformation; elastic contact; coordination displacement*

1 Introduction

Rock is a complex medium. Numerical analysis methods can consider anisotropy, complex boundary conditions, discontinuity, and characteristics varying with the time, so they are widely used in rock engineering. The basic purpose of numerical analysis is to study discrete objects and analyze the units (Zhong 1981; Zhong and Zhu 1985). At present, there are two major categories of numerical simulation methods dealing with rock engineering: techniques that consider the rock a continuous medium, such as the FEM and the boundary element method (BEM); and techniques that view the rock as a non-continuous medium and fully consider the characteristics of rock structure, such as the discrete element method, the rigid element method, and DDA method. Shi and Goodman (1988) used the block theory to conduct geotechnical stability analysis and studied the possibility of sliding with the balance conditions of blocks. The key of the block theory is to find the key blocks on the free surface,

*Corresponding author (e-mail: *hohaiwanjin@163.com*)
Received Apr. 14, 2009; accepted Jul. 16, 2009

and then to take timely engineering measures in order to ensure the stability of the rock. Zhuo and Zhao (1993) proposed the interface element method, which allows the interface components to reflect the deformation properties instead of the spring. Shi (1993) proposed the concept of numerical manifold, and proved the feasibility of this approach in theory. Then, using the manifold cover technology, he built a new numerical method including both the continuous deformation and DDA methods. Later, based on the key block theory, and the elastic and plastic theories, Ren and Yu (1999) developed the block element method, which further developed the DDA method.

As a theory of numerical calculation, the block system of DDA has been substantially developed in recent years. It reflects the discontinuity of rock deformation under the condition that it satisfies the basic equations of elasticity theory (Luan et al. 2000). It is as rigorous as the theory of FEM and can calculate large block displacements as the way the discrete element method does. It is therefore a promising numerical method.

DDA method is essentially a static analysis technique. It is similar to the FEM when it is used to solve the general equilibrium equations, while it is more like the discrete element method when it is used for calculation during a single time step. It builds the general balance equation through the principle of minimum potential energy. Taking the force-induced movement of blocks into account, the stress states are seen as a function of time (Krstulovic et al. 2002). The method of non-continuous mechanics is used to study the interaction among the units, and the naturally existing non-continuous surfaces cut the rock and form different block elements (Wang et al. 2002). Polyhedron and polygon block units can be used to overcome the limitations of continuum mechanics in the simulation of problems regarding jointed rock. They can also be used to take full consideration of the control effect of non-continuous surfaces and to combine the different constitutive relations, so this is a direct way of calculating and displaying the displacements, rotation, contingency, and sliding of the blocks, as well as the open and closed status of block interfaces.

The multi-body FEM is built on the basic theory of classical mechanics. It also has the advantages of DDA in the analysis process (Koo and Chern 1998). It views the non-continuous deformation problems as contact problems between the blocks in order to simulate the discontinuous deformation. For the internal blocks, it uses the FEM, while between the blocks it uses the classical mechanics method to set up coordinate equations. This model not only avoids rock embedding problems, but also improves the accuracy of simulation of internal block deformation (Zhang 2005).

Generally speaking, the main idea of the multi-body FEM is as follows: The global stiffness equation of the whole system is condensed into that of contact boundaries in order to form the flexibility equation, so that it is only necessary to modify the flexibility matrix during the whole contact iteration process. This simplification integrates the total stiffness equation and the flexibility equation (Kim et al. 1999). Because this method considers the contact internal force a basic unknown quantity, it is easy to determine the contact status. The multi-body FEM establishes FEM fundamental equations for each deformed block separately. Deformed blocks connect with each other through force and counterforce, and their surfaces satisfy the displacement coordinate conditions. Thus, this method can be used to solve the contact problems of deformed blocks (Li et al. 2000). Using basic mechanics theory, this paper developed a multi-body finite element formula from displacement coordination conditions of the contact surfaces and internal finite element equations of contact blocks. This method was successfully used in surrounding rock deformation analysis of underground caverns.

2 Basic principles of multi-body finite element method

Based on the assumption of the displacement field, the multi-body FEM disperses the elastic body and obtains the stiffness matrix (Panda and Natarajan 1981). On the basis of the aforementioned steps, we can obtain the influence coefficient matrix of internal forces. Then, we consider the contact force an unknown parameter, so that the continuity condition of the contact boundary can be established, which transforms the nonlinear problem into local problems. The control equation of the elastic contact problem is

$$\boldsymbol{KU}=\boldsymbol{P}+\boldsymbol{F} \tag{1}$$

where $\boldsymbol{K}$ is the global stiffness matrix, $\boldsymbol{U}$ is the nodal displacement column matrix, $\boldsymbol{F}$ is the overall external load column matrix, and $\boldsymbol{P}$ is the contact force column matrix, which is unknown. If we can obtain it by certain methods and make it satisfy all the conditions of contact status, the problem can be solved with Eq. (1). The key of the multi-body FEM is to solve these internal contact forces. Because the nonlinearity of the elastic problem is caused by a few contact points, it is reasonable to set up solution conditions for internal contact forces and obtain their values. This avoids solving the total stiffness matrix time and time again by condensing the iterative process into several contact points. Eq. (1) can be recast as

$$\boldsymbol{U} = \boldsymbol{K}^{-1}\boldsymbol{P} + \boldsymbol{K}^{-1}\boldsymbol{F} \tag{2}$$

If $\boldsymbol{C} = \boldsymbol{K}^{-1}$, and $\boldsymbol{U}_{\mathrm{F}} = \boldsymbol{K}^{-1}\boldsymbol{F}$, then

$$\boldsymbol{U} = \boldsymbol{CP} + \boldsymbol{U}_{\mathrm{F}} \tag{3}$$

$$\boldsymbol{U}' = \boldsymbol{C}'\boldsymbol{P}' + \boldsymbol{U}'_{\mathrm{F}} \tag{4}$$

where $\boldsymbol{U}'$, $\boldsymbol{C}'$, $\boldsymbol{P}'$, and $\boldsymbol{U}'_{\mathrm{F}}$ are the column matrices in an adjacent sub-structure, corresponding to $\boldsymbol{U}$, $\boldsymbol{C}$, $\boldsymbol{P}$, and $\boldsymbol{U}_{\mathrm{F}}$, respectively. The displacement increment vector of the contact point is $\boldsymbol{U}^*$, which can be defined as

$$\boldsymbol{U}^*=\boldsymbol{U}'-\boldsymbol{U} \tag{5}$$

Eqs. (3) and (4) can be used to obtain the relative distance equation:

$$\boldsymbol{U}^* = \boldsymbol{C}_{\mathrm{a}}\boldsymbol{P} + \boldsymbol{\varDelta}_{\mathrm{F}} \tag{6}$$

where $\boldsymbol{C}_{\mathrm{a}} = \boldsymbol{C} - \boldsymbol{C}'$ and $\boldsymbol{\varDelta}_{\mathrm{F}} = \boldsymbol{U}_{\mathrm{F}} - \boldsymbol{U}'_{\mathrm{F}}$. When $\boldsymbol{U}^* = 0$, Eq. (6) can be used to obtain the flexibility equation:

$$\boldsymbol{C}_{\mathrm{a}}\boldsymbol{P} = -\boldsymbol{\varDelta}_{\mathrm{F}} \tag{7}$$

In practical calculation, there are many sub-structure problems with a number of independent contact boundaries. Sub-structure problems can be solved with the general sub-structure method. The solution format is to establish two general sub-structures. Then, the sub-structure problem can be solved using the same method as the contact problem. Randomly distributed sub-structures can be divided into two categories according to their geometric positions and characteristics of contact surfaces, which are represented by $\mathrm{A}_i\ \left(i=1,2,\cdots,n_1\right)$ and $\mathrm{B}_i\ \left(i=1,2,\cdots,n_2\right)$, respectively, where n_1 and n_2 are the numbers of the two sub-structures. Each sub-structure forms a sub-structure finite element equilibrium equation (Sun et al. 2001); for category A, for example, the contact boundary conditions can be written as

$$\boldsymbol{K}_{\mathrm{A}i}\boldsymbol{U}_{\mathrm{A}i}=\boldsymbol{P}_{\mathrm{A}i}+\boldsymbol{F}_{\mathrm{A}i} \tag{8}$$

Collecting the contact boundary equations of each sub-structure, we can obtain the dominated equation of contact boundary of two general sub-structures:

$$\begin{cases}\boldsymbol{K}_{\mathrm{A}}\boldsymbol{U}_{\mathrm{A}}=\boldsymbol{P}_{\mathrm{A}}+\boldsymbol{F}_{\mathrm{A}}\\ \boldsymbol{K}_{\mathrm{B}}\boldsymbol{U}_{\mathrm{B}}=\boldsymbol{P}_{\mathrm{B}}+\boldsymbol{F}_{\mathrm{B}}\end{cases} \tag{9}$$

where $\boldsymbol{U}_{\mathrm{A}}=\left[\boldsymbol{U}_{\mathrm{A1}}\quad \boldsymbol{U}_{\mathrm{A2}}\quad \cdots\quad \boldsymbol{U}_{\mathrm{A}n_1}\right]$, $\boldsymbol{U}_{\mathrm{B}}=\left[\boldsymbol{U}_{\mathrm{B1}}\quad \boldsymbol{U}_{\mathrm{B2}}\quad \cdots\quad \boldsymbol{U}_{\mathrm{B}n_2}\right]$,

$$\boldsymbol{K}_{\mathrm{A}}=\begin{bmatrix}\boldsymbol{K}_{\mathrm{A1}} & & & \\ & \boldsymbol{K}_{\mathrm{A2}} & & \\ & & \ddots & \\ & & & \boldsymbol{K}_{\mathrm{A}n_1}\end{bmatrix},\quad \boldsymbol{K}_{\mathrm{B}}=\begin{bmatrix}\boldsymbol{K}_{\mathrm{B1}} & & & \\ & \boldsymbol{K}_{\mathrm{B2}} & & \\ & & \ddots & \\ & & & \boldsymbol{K}_{\mathrm{B}n_2}\end{bmatrix},$$

$\boldsymbol{P}_{\mathrm{A}}=\left[P_{\mathrm{A1}}\quad P_{\mathrm{A2}}\quad \cdots\quad P_{\mathrm{A}n_1}\right]$, $\boldsymbol{P}_{\mathrm{B}}=\left[P_{\mathrm{B1}}\quad P_{\mathrm{B2}}\quad \cdots\quad P_{\mathrm{B}n_2}\right]$,

$\boldsymbol{F}_{\mathrm{A}}=\left[F_{\mathrm{A1}}\quad F_{\mathrm{A2}}\quad \cdots\quad F_{\mathrm{A}n_1}\right]$, and $\boldsymbol{F}_{\mathrm{B}}=\left[F_{\mathrm{B1}}\quad F_{\mathrm{B2}}\quad \cdots\quad F_{\mathrm{B}n_2}\right]$.

3 Multi-body finite element programming

3.1 Coefficient matrix

During the solution process, the unit load of contact points can be input to form a coefficient matrix. If the tangential load is imposed on a point at the contact surface, we write 1 and 0 in the input document. If the normal load is imposed, we write 0 and 1 in the input document. After the content is read from the input document, a finite element subroutine is called for the unit load on each contact point. If the tangential force is imposed, it will form the odd series of the coefficient matrix. If the normal force is imposed, it will form the even series. The matrix $\boldsymbol{D}$ is used to store the coefficient matrix.

3.2 Right side item

The right side item is a displacement matrix associated with an external load. Each row of the matrix represents the displacement of a certain contact point caused by the external force in one direction. The matrix $\boldsymbol{P}$ is used to store the right side item.

3.3 Additional matrix of rigid body displacement

At the beginning of the procedure we can define a parameter to determine whether to calculate the rigid body displacement or not. If we need to consider it, the procedure will add the corresponding content in the coefficient matrix and the right side item. Otherwise, we will skip this step and proceed with the computation.

3.4 Storage matrix of contact surface status

For the status of contact points on the contact surface, the procedure is to define a matrix $\boldsymbol{N}$ to store the information, so that the status of contact points can be determined according to the status of the contact surface: If the contact point i is in the continuum status, $N_i = 0$. If the contact point i is in the disconnected status, $N_i = 1$. If the contact point i is in the bedding-slip status, $N_i = 2$. If the contact point i is in the inverse-slip status, $N_i = 3$.

4 Case study

During construction of a project, there may exist differences between deformations in the upstream and downstream side walls, and these may be different from section to section. That is to say, the deformation is large in some sections and small in other sections. This study applied the theory described above to study some typical cases and provided a reasonable explanation.

The basic data used in this case were the median values of field data. The main plant was 30 m in width and 70 m in height, and jointed rocks were considered to have no thickness. The rock mechanics parameters were as follows: the deformation modulus was 25 GPa, the Poisson ratio was 0.25, the coefficient of friction between layers was 0.3 without considering the adhesive force between layers, the horizontal geotropism stress was 8.0 MPa, and the vertical geotropism stress was 5.0 MPa. The area with both the width and the height of 110 m surrounding the cavern were considered the calculation region. Since the region is axisymmetric, only half of the region was considered the research object as shown in Fig 1. Fig. 2 shows the node coding map of contact surface.

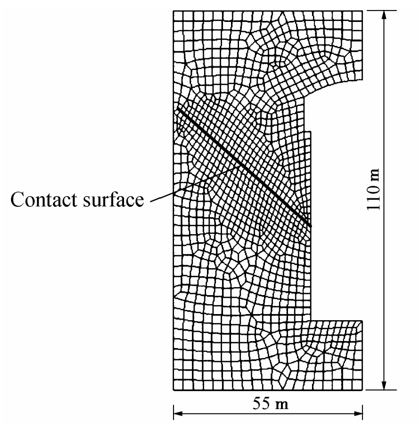

Fig. 1 Sketch of model grid

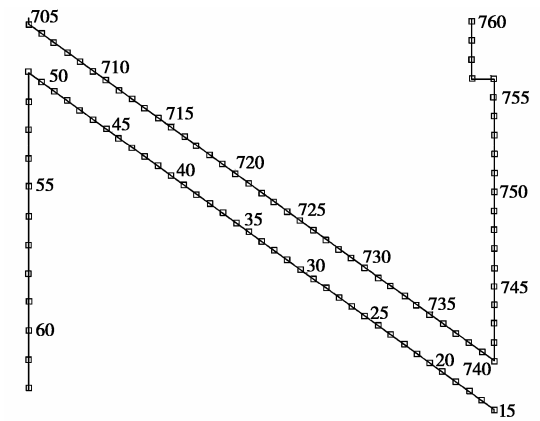

Fig. 2 Node coding map of contact surface

As seen from Table 1, the maximum deformation occurs at node 741 of the upstream side wall of the main plant where the maximum horizontal displacement occurs and its value is about 102 mm. The maximum vertical displacement that occurs at node 20 reaches 16.388 0 mm.

Table 1 Node displacement values of upper side wall of main plant mm

Node	Horizontal displacement	Vertical displacement	Node	Horizontal displacement	Vertical displacement
21	17.542 7	−16.376 9	748	70.718 5	10.125 4
20	18.013 5	−16.388 0	747	74.952 7	10.327 0
19	18.443 1	−16.387 9	746	79.274 9	10.499 3
18	18.825 4	−16.376 1	745	83.686 6	10.639 8
17	19.153 2	−16.349 4	744	88.188 7	10.745 8
16	19.412 6	−16.301 0	743	92.769 2	10.820 1
15	19.591 7	−16.190 7	742	97.433 2	10.850 5
749	66.572 0	9.897 2	741	102.139 0	10.846 3

From Fig. 3, we can see that the rock around the cavern cracks at the joints, and the maximum deformation of the side wall appears near the joints. The joints cut the upstream side wall of the main plant into blocks. The deformation of the rock block above the joints is relatively large, which is the main reason for the large deformation of the upstream side wall. Comparison of the simulation results with the measured results in the actual test shows that the horizontal and vertical displacements of the nodes around the caverns were simulated fairly accurately, particularly the maximum deformation value and the site at which the maximum deformation occurs on the upstream side wall of the main plant. This case study therefore indicates that the multi-body FEM is a good way to solve discontinuous deformation problems in underground caverns of stratified rock.

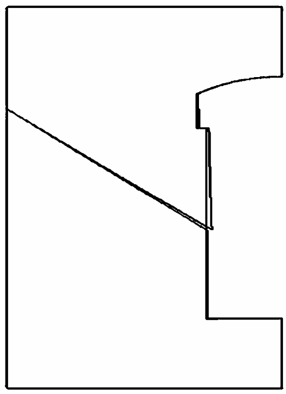

Fig. 3 Sketch of deformation of main plant

5 Conclusions

The multi-body FEM was developed and used in deformation analysis of underground caverns in this study. The results indicate that the multi-body FEM can solve discontinuous deformation problems in underground caverns very well, and it has the following advantages:

(1) Considering the displacement coordination equation the control equation helps to avoid rock embedding problems.

(2) Using the FEM for the internal blocks, the multi-body FEM attains the same usage scope as the FEM, and improves the internal deformation solution accuracy.

(3) Compared with existing calculation methods, this method can better reflect the main mechanical characteristics when dealing with discontinuity problems in rock engineering, such as the contact problems. Thus, it can obtain better results. Moreover, the model parameters needed for the calculation are easy to obtain.

References

Kim, Y., Amade, B., and Pan, E. 1999. Modeling the effect of water, excavation sequence and rock reinforcement with discontinuous deformation analysis. *International Journal of Rock Mechanics and Mining Sciences*, 36(7), 949-970. [doi:10.1016/S0148-9062(99)00046-7]

Koo, C. Y., and Chern, J. C. 1998. Modification of the DDA method for rigid block problems. *International Journal of Rock Mechanics and Mining Sciences*, 35(6), 683-693. [doi:10.1016/S0148-9062(97)00319-7]

Krstulovic, O. L., Wriggers, P., and Korelc, J. 2002. A C^1-continuous formulation for 3D finite deformation frictional contact. *Computational Mechanics*, 29(1), 27-42. [doi:10.1007/s00466-002-0317-z]

Li, X. W., Ai, K. S., and Chen, W. J. 2000. A new non-smooth model for three dimensional frictional contact problems. *Computational Mechanics*, 26(6), 528-535. [doi:10.1007/s004660000202]

Luan, M. T., Li, Y., and Yang, Q. 2000. Discontinuous deformation computational mechanics model and its application in stability analysis of rock slope. *Chinese Journal of Rock Mechanics and Engineering*, 19(3), 289-294. (in Chinese)

Panda, S., and Natarajan, R. 1981. Analysis of laminated composite shell structures by finite element method. *Computers and Structures*, 14(3-4), 225-230. [doi:10.1016/0045-7949(81)90008-0]

Ren, Q. W., and Yu, T. T. 1999. Theory and calculating model of block element method. *Engineering Mechanics*, 16(1), 67-77. (in Chinese)

Shi, G. H., and Goodman, R. E. 1988. Discontinuous deformation analysis: A new method for computing stress strain and sliding of blocks systems. *Proceedings of the 29th U.S. Symposium on Rock Mechanics, Key Questions in Rock Mechanics*, 381-393. Minneapolis: Taylor and Francis Group.

Shi, G. H. 1993. *Block System Modeling by Discontinuous Deformation Analysis*. Southampton: Computational Mechanics Publications.

Sun, L. S., Wang, D. X., and Xie, N. G. 2001. A summary of finite element analysis for contact problems. *Advances in Science and Technology of Water Resources*, 21(3), 18-20. [doi:1006-7647(2001)03-0018-03]

Wang, Z. H., Gao, L. S., and Song, W. J. 2002. Finite element analysis of 3-D friction-contact problems. *Journal of Tsinghua University (Science and Technology)*, 42(S1), 25-28. (in Chinese)

Zhang, L. J. 2005. *Hydraulic Structures of the Mechanical Model of Contact Problems at the Three Gorges Project and its Application*. Ph. D. Dissertation. Nanjing: Hohai University. (in Chinese)

Zhong, W. X. 1981. On the application of generalized displacement in finite element method and its implementation. *Chinese Journal of Mechanical Engineering*, 17(3), 15-30. (in Chinese)

Zhong, W. X., and Zhu, J. P. 1985. Generalized displacement method in finite element analysis of thin-walled beams. *Journal of Dalian Institute of Technology*, 24(4), 25-37. (in Chinese)

Zhou, J. S., and Zhao, N. 1993. Static and dynamic analysis to the rigid body of discontinuous medium: Spring element method. *Journal of Hohai University (Natural Sciences)*, 12(2), 51-57. (in Chinese)