Three-Dimensional Finite-Element Analysis of Jointed Concrete Pavement with Discontinuities

WAHEED UDDIN, ROBERT M. HACKETT, AJITH JOSEPH, ZHOU PAN, AND ALFRED B. CRAWLEY

A research study was conducted using the finite-element code ABAQUS to investigate the effects of pavement discontinuities on the surface deflection response of a jointed plain concrete pavement-subgrade model subjected to a standard falling weight deflectometer load. A significant improvement over the multilayered linear static analysis that does not allow for any discontinuity is shown. A three-dimensional pavement-subgrade finite-element model with appropriate boundary conditions has been developed. Transverse joints with dowel bars are modeled using gap and beam elements for an uncracked section, a section with cracked concrete layer, and a section with cracked concrete and cracked cement-treated base layers.

Structural analysis of a pavement-subgrade system subjected to a falling weight deflectometer (FWD) or moving wheel load based on layered linear elastic theory provides reasonable results if the pavement-subgrade system behaves as a linearly elastic system (*1*). However, for a deteriorated pavement-subgrade system with cracking and other pavement distresses (Figure 1), and where nonlinear behavior is expected from unbound granular pavement layers and subgrade, predicted linearly elastic response may differ significantly from measured response.

This paper presents some results of finite-element analyses of a jointed plain concrete pavement having discontinuities that is subjected to dynamic loading. Specifically, moduli for uncracked pavement and for pavement having cracks in both the concrete and base layers are backcalculated by matching finite-element simulation results to field measurements.

MULTILAYERED STATIC ANALYSIS OF A PAVEMENT-SUBGRADE SYSTEM

Pavement deflection response is usually analyzed using a multilayered linear elastic model under static load (*1*) to calculate the in situ Young's modulus for each layer in the pavement-subgrade system. In the multilayered linear elastic model of a pavement, each layer is characterized by its Young's modulus and its Poisson's ratio. Assuming a semi-infinite subgrade and infinite lateral boundaries, unique values of surface deflections at specified distances from the load can be theoretically predicted. Pavement nondestructive evaluation is performed through the measurement of surface deflections under a known dynamic load. The backcalculation procedure involves an iterative application of the multilayered elastic theory to calculate the in situ Young's modulus of the pavement layers. Surface deflections are predicted using assumed values of the Young's modulus and the Poisson's ratio of the pavement layers. Calculated surface deflections are matched with measured deflections until the percentage of error is reduced to an acceptably low value (*1,2*). The test load is simulated by an equivalent static load.

With the foregoing assumptions, the linearly elastic response of a pavement-subgrade model is reasonable, in the absence of pavement discontinuities and nonlinearities. However, these assumptions are clearly violated when the pavement is deteriorated or when the granular layers and subgrade exhibit nonlinear behavior, or both. Moreover, the assumption of static loading conditions is inconsistent with dynamic load application that occurs in operational situations.

THREE-DIMENSIONAL FINITE-ELEMENT ANALYSIS

Traditional pavement-subgrade analysis based on static load and multilayered linear elastic formulation with infinite dimensions in the horizontal plane and with a semi-infinite subgrade does not allow for dynamic behavior and pavement discontinuities. On the other hand, the finite-element method allows for the dynamic analysis of pavements and the consideration of finite or infinite dimensions of the physical pavement structure. Concrete pavement joints and voids beneath the pavement have in the past been modeled by the SLAB49 discrete element program (*3*). More recently, finite-element programs have been developed exclusively for pavement analysis, for example, ILLIPAVE for flexible and ILLISLAB for rigid pavements (*4,5*). These programs are capable of performing only static analyses.

The finite-element code ABAQUS is available for comprehensive structural pavement response analysis procedures, such as static and dynamic analysis (impulsive, steady-state vibratory forces, and moving wheel loads), a variety of material models (linear elastic as well as nonlinear elastic and viscoelastic material constitutive models), and problems involving crack modeling and body-to-body contact (*6,7*). The ABAQUS code for dynamic analysis, but of uncracked pavements, has been used successfully (*8*).

The finite-element method enables the evaluation of the state of stress and strain in a continuum by transforming the continuum into an assemblage of finite elements. The elements are interconnected

---

W. Uddin, R. M. Hackett, A. Joseph, and Z. Pan, Department of Civil Engineering, University of Mississippi, University, Miss. 38677. A. B. Crawley, Mississippi Department of Transportation, P.O. Box 1850, Jackson, Miss. 39215.

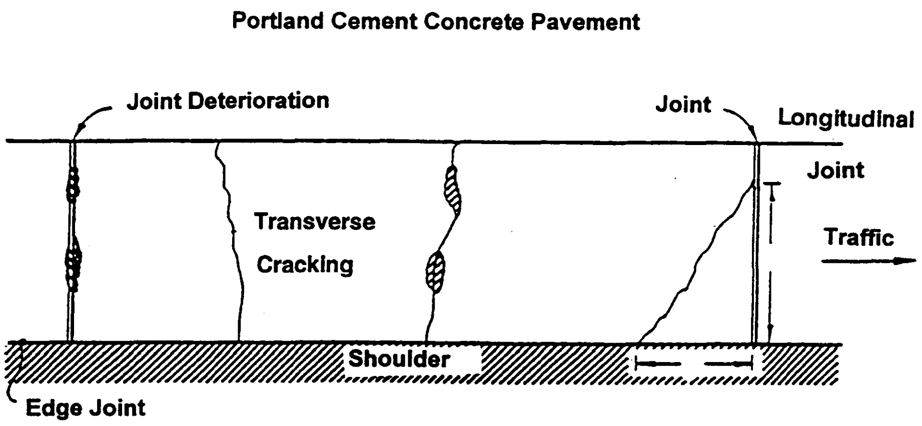

FIGURE 1 **Typical concrete pavement discontinuities.**

at their common nodes. Dynamic finite-element analysis involves the solution of the differential equation of motion:

$$M \ddot{U} + C \dot{U} + K U = F$$

where

M = mass matrix,
C = damping matrix,
K = stiffness matrix,
$\ddot{U}, \dot{U}, U$ = vectors of acceleration, velocity, and displacement, respectively, and
F = vector of nodal forces.

Integration of this equation yields displacements at the nodal points, for any given instant in time. The stresses and strains are computed through backsubstitution, using the kinematic and constitutive relationships.

THREE-DIMENSIONAL FINITE-ELEMENT MODELING

Finite-element modeling using any software package involves three stages: (a) preprocessing, where the finite element mesh is generated, loads and boundary conditions are assigned, and material properties are defined; (b) analysis, where displacements, stresses, and strains are computed; and (c) postprocessing, where the results are graphically presented. In the current study, the CAD/FEM program PATRAN is used as the pre- and postprocessor, and ABAQUS is used for the analysis. A brief description of these programs follows.

ABAQUS AND PATRAN SOFTWARE

The ABAQUS software (*6,7*) is a comprehensive finite-element code developed to solve two- and three-dimensional problems with static, harmonic, and transient dynamic loading and thermal gradient conditions. Material can be modeled as linear elastic, nonlinear elastic, viscoelastic, plastic, and modified elastic. Effects from cracks, voids, and material degradation can be analyzed. Computer simulation of the actual behavior can lead to a better understanding of pavement performance and to a reliable estimated prediction of loss of support over the pavement life.

PATRAN, a finite-element modeling computer program, supports powerful graphics capabilities for interactive mesh generation and output visualization (*9*). A finite-element model generated through the use of PATRAN is translated into ABAQUS input data, using a forward translation program. Subsequent to the analysis, the results from ABAQUS are translated back into PATRAN for graphical visualization and the plotting of results.

PAVEMENT-SUBGRADE MODEL PARAMETERS

The following dimensions and boundary conditions for a three-dimensional finite-element pavement model were investigated in a previous study (*10*):

- Rollers on the lateral sides of the model.
- Subgrade width of 13.3 m (43.65 ft) for the quarter-symmetric model.
- Pavement length of 9.14 m (30 ft). This is equal to one and one-half times the length of a typical concrete slab on U.S. Highway 78 in Marshall County, Mississippi.
- Discontinuous shoulders along the pavement edges, modeled using gap elements. The outside shoulder is 3.04 m (10 ft) wide.

FINITE-ELEMENT MODEL FOR PAVEMENT WITH NO DISTRESS

Mesh Configuration and Material Properties

The pavement is modeled as a three-layered linear elastic system that consists of a portland cement concrete layer, a cement-treated base (CTB) layer, and a subgrade. The thicknesses and material properties for a jointed plain concrete pavement structure of U.S. Highway 78 are shown on Table 1. FWD deflection data were collected from nine test sections on US-78 in May 1994.

FWD loading was used in the study. The center of the load was located 1.5 m (5 ft) from the edge of the outside shoulder and 3 m (10 ft) from the transverse joint.

Mesh size and configuration are an important part of finite-element modeling; precise mesh refinement is necessary in regions of high stress intensity. A refined mesh was developed for the vicinity of the loaded area and the transverse joint. The finite-element model was generated using PATRAN. The model has a subgrade depth of

TABLE 1 U.S. Highway 78 Pavement Structure and Moduli Backcalculated from Static Analyses

Layer	Thickness mm (inches)	Backcalculated Modulus, Mpa (ksi) 1994 FWD Deflection Data	
Concrete	254 (10.0)	36,855	(5,349)
Cement Treated Base (CTB)	152 (6.0)	4,272	(620)
Subgrade	Semi-infinite	176	(25.6)

12 m (40 ft) and has 7,546 finite elements. A three-dimensional view of the model is shown in Figure 2. The central processing unit time for running the model was approximately 200 sec on a Cray Y-MP supercomputer. This model was used for all of the simulations.

The overall dimensions of the model and the boundary conditions applied on the edges are based on the actual dimensions of the U.S. Highway 78 jointed plain concrete pavement under study and on the results of previous studies (*11*). During the current study, a detailed investigation of the U.S. Highway 78 pavement in Marshall County was conducted; it involved

- Detailed visual distress survey and mapping based on Strategic Highway Research Program procedures to identify good and deteriorated pavement sections,
- Ground-penetrating radar survey to establish pavement thickness nondestructively and to identify possible weak areas,
- Noncontact thermographic survey to establish locations having possible voids or moisture damage, or both,
- Coring to verify pavement layer thickness and to identify areas with loss of support, and
- Nondestructive deflection testing with FWD on transverse joints and midslab locations, and side-by-side Dynaflect testing on selected locations.

Transverse Joint Modeling

In Figure 3, the details of a typical transverse joint with dowel bars are shown. The dowel bars are modeled using beam elements.

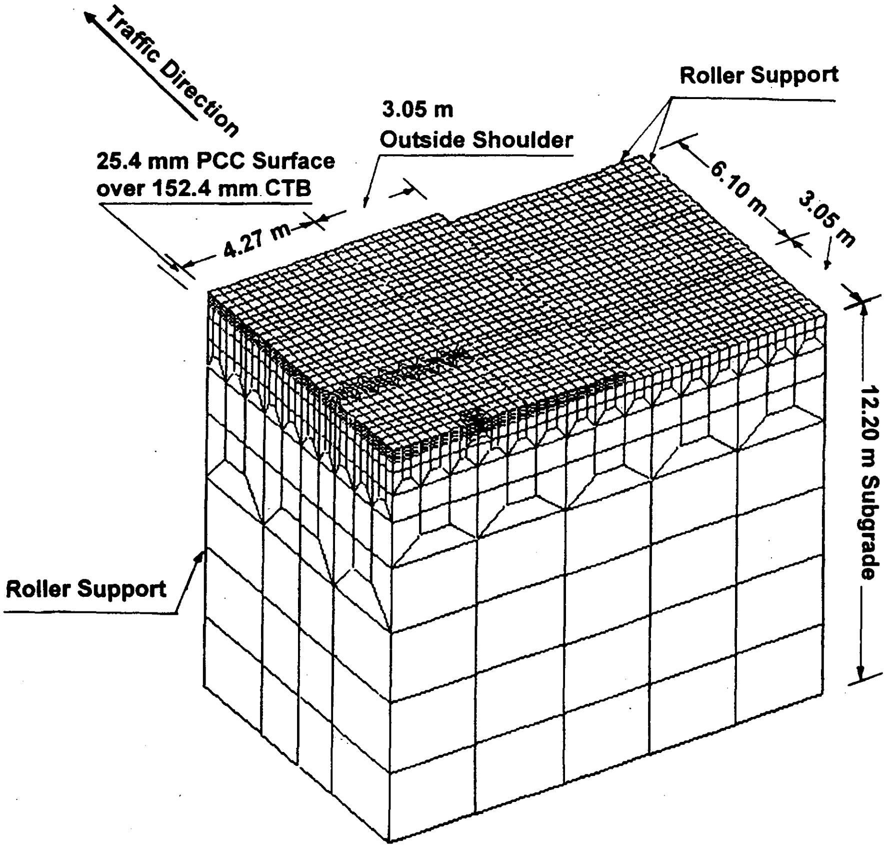

FIGURE 2 Three-dimensional view of the finite-element model.

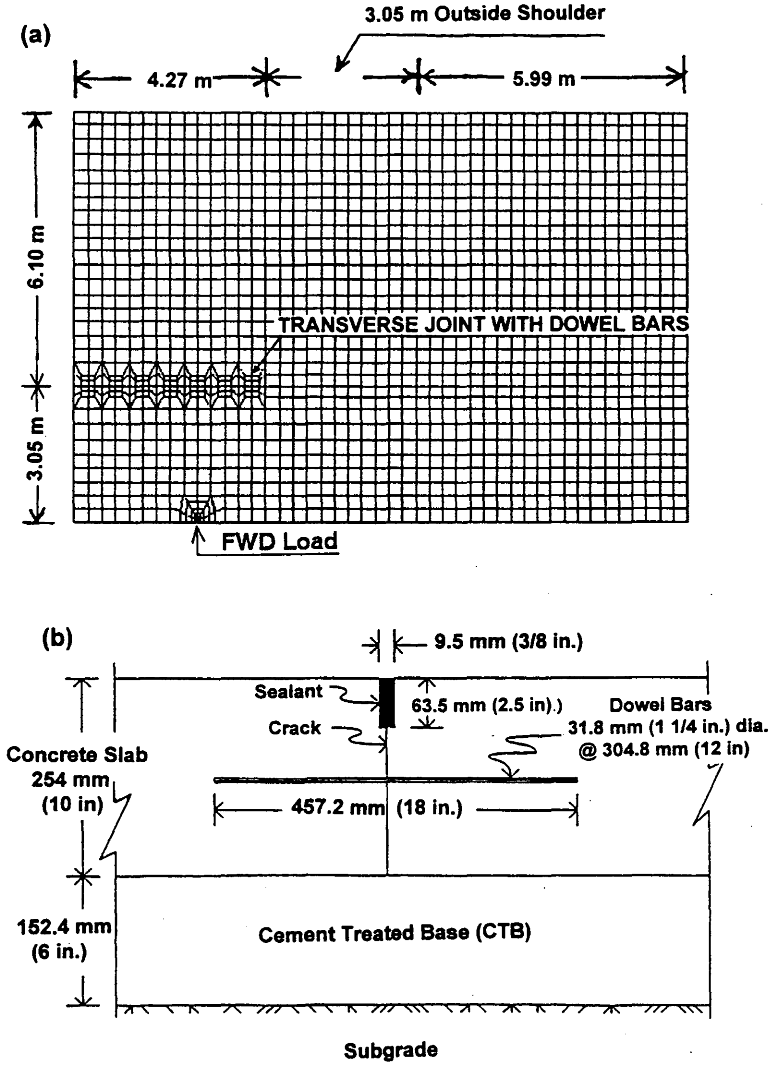

FIGURE 3 (a) Plan view of finite-element mesh and (b) cross section of transverse joint.

While one end of the dowel is fixed into one concrete slab, the other end is free to move back and forth in the adjacent concrete slab, depending on the thermal expansion or contraction of the slabs. Thus, the interaction between the dowel bar and the concrete involves body-to-body contact. Gap elements in ABAQUS are used in this regard. These elements are used to specify the interaction between the dowel and the surrounding concrete medium. The steel dowel bars are 457 mm (18 in.) long, with a circular cross section having a diameter of 31.75 mm (1.25 in.). The dowel bars are placed in the middle of the slab at 305-mm (12-in.) spacing. A Young's modulus of 206,700 MPa (30,000 ksi) and a Poisson's ratio of 0.3 are used to characterize the dowel bars. Also, a crack through the thickness of the concrete slab develops at the transverse joint immediately after construction. Gap elements are used to model the contact between the two faces of the crack. The use of these elements for modeling the crack is explained in detail in a later section.

Static Analysis Results

The FWD deflection data were analyzed using the static layered elastic analysis incorporated in the PEDD1 backcalculation software. The average backcalculated modulus values for uncracked pavement sections are presented in Table 1. These backcalculated modulus values provide an initial estimate of layer material properties for dynamic backcalculation using a simulated FWD load.

Pavement Modulus Backcalculation Using Dynamic Analysis Results

Dynamic analysis was performed using the ABAQUS implicit approach. This is different from the explicit procedure (*7*) in that the implicit method computes the deflections at any time *t* by knowing the deflections at time *t* − 1 by solving a set of nonlinear equations, whereas the explicit method computes the deflections at any time *t* by adding the deflection increments between time *t* and time *t* − 1, computed by double integration of the acceleration obtained from the dynamic equations at that degree of freedom, to the deflection at time *t* − 1. The FWD load time history based on a typical FWD pulse of 33-msec duration, measured on the test pavement, is used to simulate the FWD impact load. The ABAQUS dynamic deflections were compared with the FWD measured deflections, and modulus

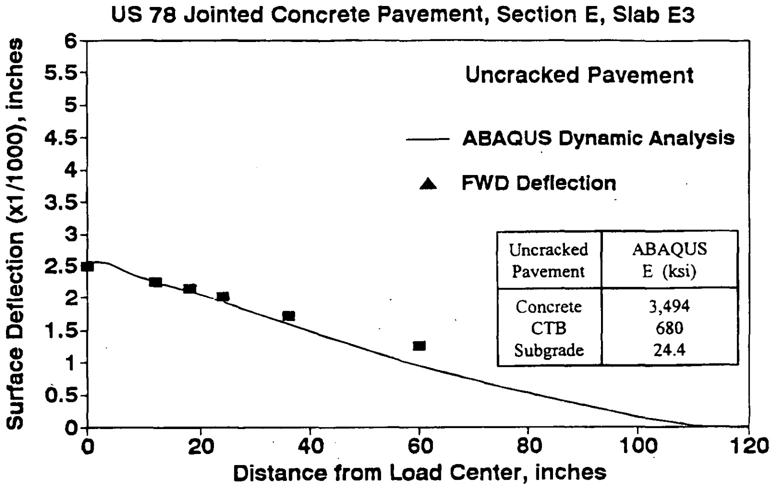

FIGURE 4 Matching of ABAQUS-computed surface deflections with measured FWD deflections for uncracked pavement.

values were adjusted until a close agreement was achieved between computed and measured dynamic deflections. The computed and measured deflections for the uncracked test Slab E3 are shown in Figure 4. The backcalculated modulus values for the uncracked concrete, the uncracked base, and the subgrade are listed in Table 2.

PAVEMENT CRACKING SIMULATION

Longitudinal and transverse cracks, joints, and voids beneath the concrete surface layer are the most critical discontinuities in concrete pavements, as illustrated in Figure 1. The structural response of a deteriorated pavement can differ significantly from that of an uncracked pavement having no distress. ABAQUS dynamic analyses were made to investigate the effect of cracks on the pavement properties.

The effect of cracks in the pavement can be modeled using special-purpose elements. A crack is modeled by having two independent nodes on two free faces of the crack linked by special-purpose unidirectional gap elements. The elements allow two continuous surfaces to be in contact, or not in contact, through contact pressure and friction between the contacting surfaces. ABAQUS monitors the relative displacement of the two nodes of the element in the given direction. This arrangement results in two contact surfaces that are separated by an initial selected gap width at the top. The gap element controls the interaction between the contact surfaces in such a way that these surfaces do not penetrate each other under any contact pressure. An appropriate value of the friction coefficient parameter between the contact surfaces should be assumed in the analysis to simulate aggregate interlock effects across the crack. A zero-friction coefficient means that no shear forces will develop and the contact surfaces will be free to slide. A very large friction coefficient implies that the surfaces will lock and no sliding will occur.

A sensitivity analysis of friction coefficient and gap width was conducted by varying the crack gap width from 5.1 mm (0.2 in.) to 0.25 mm (0.01 in.). It was concluded that the critical gap width was 0.25 mm (0.01 in.) at which the effect of friction coefficient on surface deflection is significant (*10*). This is expected for a low to

TABLE 2 Comparison of Pavement Moduli Values Backcalculated from ABAQUS Dynamic Analyses for Uncracked and Cracked Pavement Sections

Layer	Backcalculated Moduli, MPa (ksi)*		
	Uncracked Pavement	Cracked Concrete, Uncracked CTB	Cracked Concrete, Cracked CTB
Concrete	24,074 (3,494)	12,746 (1,850)	12,746 (1,850)
Cement Treated Base (CTB)	4,685 (680)	4,685 (680)	2,067 (300)
Subgrade	168 (24.4)	168 (24.4)	168 (24.4)
Test Section Location	E slab E3	F slab F3	G slab G1

* (Based on FWD deflection data measured in 1994 on US Highway 78 in Marshall County, Mississippi)

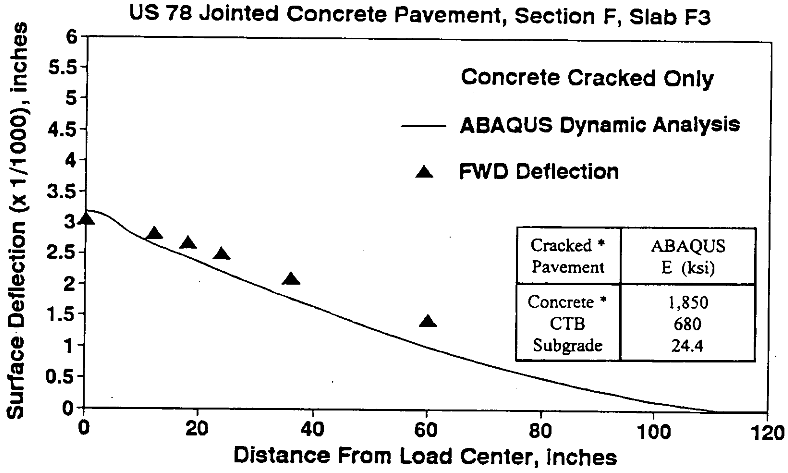

FIGURE 5 Matching of ABAQUS-computed surface deflections with measured FWD deflections for pavement with crack in concrete layer.

medium severity crack in the pavement concrete layer. Therefore, a gap width of 0.25 mm (0.01 in.) and a friction coefficient of 0.5 were used in this study.

Longitudinal, diagonal, and transverse cracks were observed at some pavement sections of U.S. Highway 78. A crack of low severity observed on one of the test section slabs (Slab F3) initiates from the transverse joint at a distance of 1.5 m (5 ft) from the pavement edge. A core extracted from this test section indicates cracking in the concrete layer only. This distress was simulated in the finite-element model and the calculated ABAQUS dynamic deflections were substantially higher than the corresponding deflection calculated for the uncracked pavement. The effective concrete modulus for this cracked pavement slab is naturally expected to be smaller in magnitude than that for the uncracked pavement layer. After a few iterations, a lower concrete modulus value yielded ABAQUS dynamic deflections that agree reasonably well with the FWD deflections measured at this cracked pavement site, as shown in Figure 5. The backcalculated modulus values for the cracked concrete, the uncracked base, and the subgrade are listed in Table 2.

Using the FWD data, distress data, and core data collected in this study, Slab Gl, in test Section G, was selected as representative of cracking in concrete and cement-treated base layers. By applying a similar iterative approach, the modulus values were backcalculated to the point where the computed dynamic deflections agreed reasonably well with the measured deflections, as shown in Figure 6. The backcalculated modulus values for the cracked concrete, the cracked base, and the subgrade are listed in Table 2. These results are indicative of the importance of using dynamic analysis to backcalculate appropriate values of pavement modulus.

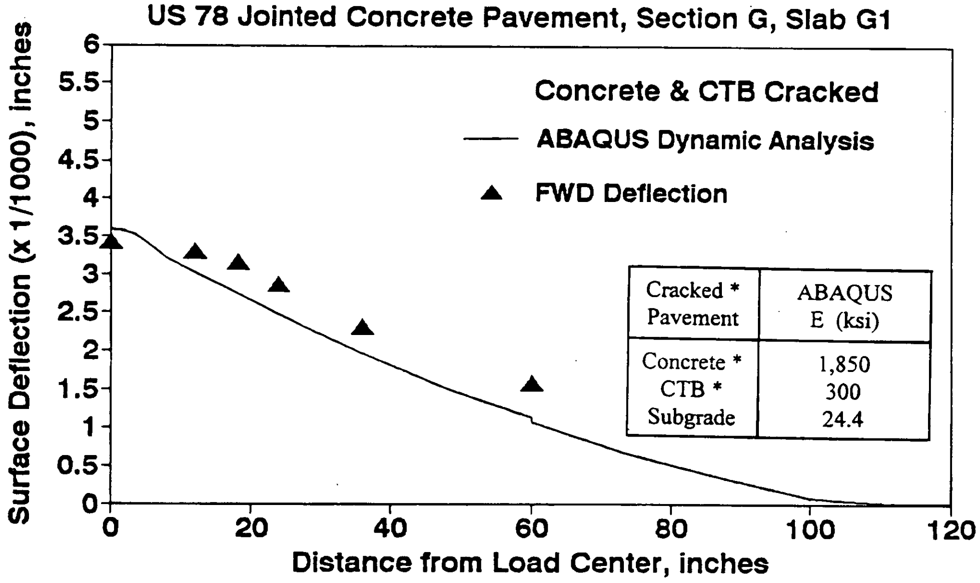

FIGURE 6 Matching of ABAQUS-computed surface deflections with measured FWD deflections for pavement with crack in concrete and cement treated base layers.

SUMMARY AND CONCLUSIONS

The effects of pavement cracking and FWD dynamic loading on the structural response of jointed plain concrete pavement-subgrade systems have been studied using the ABAQUS finite-element code. General observations and specific conclusions follow.

Advances in high-speed, high-capacity computational simulation have provided a capability of modeling with extreme accuracy the response of physical systems that are distinguished by discontinuities and highly nonlinear behavior. Jointed concrete pavements certainly fall within this category of physical systems that require sophisticated analyses to achieve efficient and economical design and construction. It is important to recognize the applicability of advanced modeling procedures to these types of systems and to use them to gain much greater understanding of their behavior than could be gained from the use of simplified approximate methods that fail, in many cases, to accurately portray the actual system response.

Lower backcalculated modulus values are to be expected for cracked pavements, compared with those backcalculated for uncracked pavements. In this study, procedures for quantifying this knowledge are demonstrated. This study also demonstrates the extensive usefulness of three-dimensional finite-element simulation of the effect of cracks and dynamic loading for accurately calculating values of deflection and stresses and strains, as well as the values of reduced moduli, associated with deteriorated pavement systems, and resulting from the presence of cracks and voids. It is impossible to carry out these types of studies with traditional multilayered linear elastic analyses as well as other finite-element programs that do not allow crack modeling and dynamic analysis.

Further three-dimensional finite-element modeling is under way to investigate the effects of voids beneath the surface concrete layer and the effects of thermal gradients on jointed concrete pavement response, using the ABAQUS concrete material model.

ACKNOWLEDGMENTS

The work was conducted as part of a research study supported by the Mississippi Department of Transportation and FHWA. The finite-element simulations were conducted on the Cray Y-MP supercomputer at the University of Mississippi with the support of the Mississippi Center for Supercomputing Research staff.

REFERENCES

1. Uddin, W., A. H. Meyer, W. R. Hudson, and K. H. Stokoe II. Project-Level Structural Evaluation of Pavements Based on Dynamic Deflections. In *Transportation Research Record 1007*, TRB, National Research Council, Washington, D.C., 1985, pp. 37–45.
2. Uddin, W., A. H. Meyer, and W. R. Hudson. Rigid Bottom Considerations for Nondestructive Evaluation of Pavements. In *Transportation Research Record 1070*, TRB, National Research Council, Washington, D.C., 1986, pp. 21–29.
3. Hudson, W. R., and H. Matlock. Analysis of Discontinuous Orthotropic Pavement Slabs Subjected to Combined Loads. In *Highway Research Record 131*, HRB, National Research Council, Washington, D.C., 1966.
4. Elliott, R. P., and M. R. Thompson. *Mechanistic Design Concepts for Conventional Flexible Pavements.* Transportation Engineering Series 42. University of Illinois, Urbana, 1985.
5. Tabatabie, A. M., and E. J. Barenberg. Finite Element Analysis of Jointed or Cracked Concrete Pavements. In *Transportation Research Record 671*, TRB, National Research Council, Washington, D.C., 1978.
6. *ABAQUS, Theory Manual Version 5.3.* Hibbitt, Karlsson, & Sorenson, Inc., Pawtucket, R.I., 1993.
7. *ABAQUS, User's Manual.* Hibbitt, Karlsson, & Sorenson, Inc., Pawtucket, R.I., 1993.
8. Zaghloul, S., and T. White. Use of a Three-Dimensional, Dynamic Finite Element Program for Analysis of Flexible Pavement. In *Transportation Research Record 1388*, TRB, National Research Council, Washington, D.C., 1993, pp. 60–69.
9. *PATRAN User's Manual.* PDA Engineering, PATRAN Division, Costa Mesa, Calif., 1993.
10. Uddin, W., D. Zhang, and F. Fernandez. Finite Element Simulation of Pavement Discontinuities and Dynamic Load Response. In *Transportation Research Record 1448*, TRB, National Research Council, Washington, D.C., 1994, pp. 100–106.
11. *U. S. Highway 78, Marshall County Project Documents.* Mississippi Department of Transportation, 1990, 1991.

Publication of this paper sponsored by Committee on Rigid Pavement Design.