SIMULATION OF ASPHALT MATERIALS USING A FINITE ELEMENT MICROMECHANICAL MODEL WITH DAMAGE MECHANICS

Martin H. Sadd, Ph.D.
Professor
Department of Mechanical Engineering & Applied Mechanics
University of Rhode Island
Kingston, RI 02881
Tel: 401-874-5548
Fax: 401-874-2355
E-mail: sadd@egr.uri.edu

Qingli Dai
Graduate Research Assistant
Department of Mechanical Engineering & Applied Mechanics
University of Rhode Island
Kingston, RI 02881
Tel: 401-874-2524
Fax: 401-874-2355
E-mail: daiq@egr.uri.edu

Venkit Parameswaran, Ph.D.
Visiting Research Faculty
Department of Mechanical Engineering & Applied Mechanics
University of Rhode Island
Kingston, RI 02881
Tel: 401-874-2227
Fax: 401-874-2355
E-mail: paramev@egr.uri.edu

Arun Shukla, Ph.D.
Simon Ostrach Professor
Department of Mechanical Engineering & Applied Mechanics
University of Rhode Island
Kingston, RI 02881
Tel: 401-874-2283
Fax: 401-874-2355
E-mail: shukla@egr.uri.edu

Submitted for Presentation at the 2003 TRB Annual Meeting and Publication in the Transportation Research Record: Journal of the Transportation Research Board

Word Count: (4698 text + 10 figures @ 250 + 0 tables @ 250 = 7198 words)

July 31, 2002
ABSTRACT

This work presents a theoretical/numerical study of the micromechanical behavior of asphalt concrete. Asphalt is a heterogeneous material composed of aggregates, binder cement and air voids. The load carrying behavior of such a material is strongly related to the local load transfer between aggregate particles, and this is taken as the microstructural response. Numerical simulation of this material behavior was accomplished by developing a special finite element model which incorporated the mechanical load-carrying response between the aggregates. The finite element scheme incorporated a network of special frame elements each with a stiffness matrix developed from an approximate elasticity solution of the stress and displacement field in a cementation layer between particle pairs. A damage mechanics approach was then incorporated within this solution, and this led to the construction of a softening model capable of predicting typical global inelastic behavior found in asphalt materials. This theory was then implemented within the ABAQUS FEA code to conduct simulations of particular laboratory specimens. A series of model simulations of indirect tension tests (IDT) were conducted to investigate the effect of variation of specimen microstructure on the sample response. Simulation results of the overall sample behavior compared favorably with experimental results. Additional comparisons were made of the evolving damage behavior within the IDT samples, and numerical results gave reasonable predictions.
1. INTRODUCTION

Asphalt is a complex heterogeneous material composed of aggregate, binder/cement, additives and void space. The load carrying behavior and resulting failure of such materials depends on many phenomena that occur at the aggregate/binder level. Thus the overall macro behavior is determined by the micromechanics within the cemented particulate system. Special additives and recycled asphalt product are also commonly used in pavement mixes, and this further complicates the material behavior by introducing several ageing effects such as hardening, chemical oxidation and binder microcracking. Because of the heterogeneous multiphase nature of asphalt material, it does not appear that traditional continuum mechanics theory will be able to predict important micro-behavior at the aggregate/binder level, and thus micromechanics modeling is needed. An understanding of the micromechanics offers the possibility to more accurately predict asphalt failure behavior and to relate such behavior to particular mix parameters such as binder properties and aggregate size, shape and gradation. The purpose of the present work is to develop a micromechanical model for asphalt concrete using a special finite element scheme incorporating damage mechanics concepts.

Over the past two decades, many studies have been investigating the micromechanical behavior of particulate, porous and heterogeneous materials. For example, studies on cemented particulate materials by Dvorkin et al. (1) and Zhu et al. (2) provide information on the basic load transfer between particles that are cemented together. Such studies provide details on the normal and tangential interparticle load transfer, and would be fundamental in developing a micromechanical theory for load distribution and failure of such materials. Several recent applications of such contact-based micromechanical analysis for asphalt behavior have been reported by Chang and Gao (3), Cheung, et al. (4) and Zhu et al. (5). In a related study, Krishnan and Rao (6) used mixture theory and presented a multi-phase approach to explain air void reduction in asphalt materials under load.

Numerical modeling of cemented particulate materials has generally used both finite and discrete element methods. The discrete element method (DEM) analyzes particulate systems by modeling the translational and rotational behaviors of each particle using Newton’s second law with appropriate inter-particle contact forces. Normally the scheme establishes an explicit, time-stepping procedure to determine each of the particle motions. DEM studies on cemented particulate materials include the work by Rothenburg, et al. (7), Chang and Meegoda (8), Trent and Margolin (9), Buttlar and You (10), Ullidtz (11) and Sadd et al. (12,13).

In regard to finite element modeling (FEM), Stankowski (14) applied standard FEM techniques to cemented particulate composites. Sepehr et al. (15) used an idealized finite element microstructural model to analyze the behavior of an asphalt pavement layer. A common finite element approach to simulate particulate and heterogeneous materials has used an equivalent lattice network system to represent the interparticle load transfer behavior. This type of microstructural modeling has been used previously: Bazant, et al. (16), Mora (17), Sadd et al. (18) and Budhu, et al. (19). Along similar lines, Guddati, et al. (20) recently presented a random truss lattice model to simulate microdamage in asphalt concrete and demonstrated some interesting failure patterns in an indirect tension test geometry. Birgisson et al. (21) used a displacement discontinuity boundary element approach to model asphalt mixtures. Bahia et al. (22) have also used finite elements to model the aggregate-binder response of asphalt materials, and Papagiannakis, et al. (23) have conducted similar studies for the viscoelastic response. Mustoe and Griffiths (24) developed a finite element model, which was equivalent to a particular discrete element approach. They pointed out that the FEM model has an advantage over the discrete element scheme for static problems.

This paper presents a numerical modeling scheme for asphalt concrete based on micromechanical simulation using the finite element method. The model first incorporates an equivalent lattice network approach whereby the local interaction between neighboring particles is modeled with a special frame-type finite element. The element stiffness matrix is constructed by considering the normal, tangential and rotation behaviors between cemented particles, and this is accomplished using an approximate elasticity solution within the cementation interface. The inelastic softening behavior exhibited by these materials is developed by incorporating a damage mechanics theory within the FEM model. Although this network approach is similar to other reported models, it is the element stiffness equation which makes this work...
different from previous research. This theoretical formulation was then implemented into the commercial ABAQUS FEA code using user-defined elements. A series of model simulations of indirect tension tests (IDT) were conducted to investigate the effect of variation of specimen microstructure on the sample response. IDT samples were generated using a MATLAB material generating code specially developed for this modeling effort. Simulation results of the overall sample behavior compared favorably with experimental results of actual asphalt samples. Additional comparisons were made of the evolving damage behavior within the IDT specimen, and numerical results gave reasonable predictions. Such a micromechanical model can provide important connections between asphalt material performance and details on the mix design.

2. MICROMECHANICAL FINITE ELEMENT MODEL

As mentioned, bituminous asphalt can be described as a multi-phase material containing aggregate, binder cement (including mastic and fine particles) and air voids (see Figure 1). The load transfer between the aggregates plays a primary role in determining the load carrying capacity and failure of such complex materials. In order to develop a micromechanical model of this behavior, proper simulation of the load transfer between the aggregates must be accomplished. The aggregate material is normally much stiffer than the binder, and thus aggregates are taken as rigid particles. On the other hand, the binder cement is a compliant material with elastic, inelastic, and time-dependent behaviors. Additionally, binder behavior can also include hardening, debonding and microcracking, and these lead to many complicated failure mechanisms. In order to properly account for the load transfer between aggregates, it is assumed that there is an effective binder zone between neighboring particles. It is through this zone that the micromechanical load transfer occurs between each aggregate pair. This loading can be reduced to resultant normal and tangential forces and a moment as shown in Figure 1.

In order to model the inter-particle load transfer behavior, some simplifying assumptions must be made about allowable aggregate shape and the binder geometry. Studies on aggregate geometry have commonly quantified particle size, shape, angularity and texture. However, for the present modeling only size and shape will be considered. In general, asphalt concrete contains aggregate of very irregular geometry as shown in Figure 2(a). Our approach is to allow variable size and shape using an elliptical aggregate model as represented in Figure 2(b). The finite element model uses an equivalent lattice network approach, whereby the interparticle load transfer is simulated by a network of specially created frame-type finite elements connected at particle centers as shown in Figure 2(c). From granular materials research, the material microstructure or fabric can be characterized to some extent by the branch vector distribution which are the line segments from adjacent particle mass centers. Note that the proposed finite element network coincides with the branch vector distribution.

The network model uses a specially developed, two-dimensional frame-type finite element to simulate the interparticle load transfer. These two-noded elements have the usual three degrees-of-freedom (two displacements and a rotation) at each node and thus require a 6x6 stiffness matrix to relate nodal (aggregate) motions to the applied forces and moments. Thus the element equation is written as

\[
\begin{bmatrix}
K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\
K_{12} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\
K_{13} & K_{23} & K_{33} & K_{34} & K_{35} & K_{36} \\
K_{14} & K_{24} & K_{34} & K_{44} & K_{45} & K_{46} \\
K_{15} & K_{25} & K_{35} & K_{45} & K_{55} & K_{56} \\
K_{16} & K_{26} & K_{36} & K_{46} & K_{56} & K_{66}
\end{bmatrix}
\begin{bmatrix}
U_1 \\
V_1 \\
\theta_1 \\
U_2 \\
V_2 \\
\theta_2
\end{bmatrix}
= 
\begin{bmatrix}
F_{n1} \\
F_{n2} \\
M_1 \\
F_{n1} \\
F_{n2} \\
M_2
\end{bmatrix}
\]

where \(U_i\), \(V_i\) and \(\theta_i\) are the nodal displacements and rotations, and \(F_n\) and \(M\) are the nodal forces. The usual scheme of using bar and/or beam elements to determine the stiffness terms is not appropriate for the current modeling, and therefore these terms were determined using an approximate elasticity solution from Dvorkin et al. (1) for the stress distribution in a cement layer between two particles. We use the
special case where the particle material stiffness is much greater than that of the cement layer, and thus the particles are assumed to be rigid. Dvorkin has shown that effects of non-uniform cement thickness for each binder element are generally small, and so the analytical solution with average cement thickness for each binder element was used. The two-dimensional model geometry is shown in Figure 3. Note that we are allowing arbitrary non-symmetric cementation, and thus the finite element line will not necessarily pass through the center of the binder material. Thus in general, \( w = w_1 + w_2 \), but \( w_1 \neq w_2 \neq w/2 \), and an eccentricity variable may be defined by \( e = (w_2 - w_1)/2 \).

The stresses \( \sigma_x, \sigma_z \) and \( \tau_{xz} \) within the cementation layer can be calculated for particular relative particle motion cases involving normal, tangential and rotational deformation. These stresses can then be integrated to determine the total load transfer within the cement binder, thus leading to the calculation of the various stiffness terms needed in the element equation. The details of this process have been previously reported by Sadd and Dai (25), and the final result is given by

\[
[K] = \begin{bmatrix}
K_{nn} & 0 & K_{ne} & -K_{nn} & 0 & -K_{nn}e \\
0 & K_{tt} & 0 & -K_{tt} & 0 & -K_{tt}r_2 \\
K_{ne} & 0 & K_{nn} & -K_{tt} & 0 & -K_{tt}r_2 \\
-K_{nn} & 0 & K_{nn} & 0 & K_{nn}e & K_{nn}e \\
0 & -K_{tt} & 0 & K_{nn} & 0 & K_{nn} \\
-K_{nn}e & -K_{tt}r_2 & -K_{tt}r_2 & 0 & K_{nn} & 0 \\
\end{bmatrix}
\]

where \( K_{nn} = (\lambda + 2\mu)w/h_o \), \( K_{tt} = \mu w/h_o \), \( \lambda \) and \( \mu \) are the usual elastic moduli, \( h_o \) is the average cementation thickness, \( r_1 \) and \( r_2 \) are the radial dimensions from each aggregate center to the cementation boundary, \( w_1 \) and \( w_2 \) are cementation widths, and \( e = (w_2 - w_1)/2 \). Each binder element stiffness matrix is significantly different depending on two-particle layout and size and binder geometry. This procedure establishes the elastic stiffness matrix, and it is clearly a function of the micromechanical material variables including particle size and shape and cementation geometry and moduli.

3. DAMAGE MECHANICS MODEL

In order to simulate the inelastic and softening behaviors observed in asphalt materials, a damage mechanics approach was applied within the inter-particle cementation model. Although work on such a damage model has been previously reported by Zhong and Chang (26), the approach by Ishikawa, Yoshikawa and Tanabe (27) was found to be more useful for our finite element model. The theory was originally developed for concrete materials whereby the internal micro-cracks within the matrix cement and around the aggregates are modeled as a continuous defect field. Inelastic behavior is thus developed by the growth of damage within the material with increasing loading. A damage tensor \([\Omega]\) is defined by considering the reduction of the effective area of load transfer within the continuum. The total strain field is defined as the sum of the elastic and damage strains

\[
\{\varepsilon\} = \{\varepsilon_e\} + \{\varepsilon_f\}
\]

and thus the elastic constitutive relationship can be expressed as

\[
\{\sigma\} = [D_o]\{\varepsilon_e\} = [D_o]\{\varepsilon - \varepsilon_f\}
\]

where \([D_o]\) is initial elastic stiffness matrix.

The damage strain represents the difference between the total and elastic strains and can be written as

\[
\{\varepsilon - \varepsilon_f\} = \{\varepsilon\} - \{\varepsilon_e\} = \{\varepsilon\} - \{\varepsilon_e\}
\]
This leads to the development of a damage stiffness matrix \([D_s]\) defined by

\[
\{\sigma\} = ([I] - [\Omega]) [D_o] [\varepsilon] = [D_s] [\varepsilon]
\]

The damage stiffness matrix can be obtained from the initial elastic stiffness matrix

\[
[D_s] = ([I] - [\Omega]) [D_0]
\]

In order to characterize the nonlinear damage behavior of asphalt concrete, a particular Weibull distribution function is chosen to describe the evolution of the defect field within the binder cement. Such forms have been used before and for the uniaxial hardening response, the constitutive relation is taken as

\[
\sigma = \sigma_0 (1 - e^{-b(\varepsilon/\varepsilon_0)}) \Rightarrow \frac{\partial \sigma}{\partial \varepsilon} = D_0 e^{-b(\varepsilon/\varepsilon_0)}
\]

where the material parameters \(\varepsilon_0\) and \(b\) are related to the softening strain and damage evolution rate respectively, \(\sigma_0\) is the material strength, and \(D_0 = \sigma_0 b / \varepsilon_0\) is the initial elastic stiffness. Using the damage stiffness definition from relationship (7), the uniaxial damage stiffness \(D_s\) and the damage scalar \(\Omega\) become

\[
D_s = (1 - \Omega) D_0 = D_0 e^{-b(\varepsilon/\varepsilon_0)}, \text{ where } \Omega = 1 - e^{-b(\varepsilon/\varepsilon_0)}
\]

In this damage model, the critical damage scalar \(\Omega_c\) and critical strength \(\sigma_c\) are expressed as

\[
\Omega_c = 1 - e^{-b}, \text{ and } \sigma_c = \sigma_0 (1 - e^{-b})
\]

Note that the critical value of the damage scalar \(\Omega_c\) could be less than 1 for asphalt concrete. Once the damage scalar reaches \(\Omega_c\), the material point will gradually lose its stiffness.

After critical strength the softening behavior is taken as

\[
\sigma = \sigma_0 (1 - e^{-b}) e^{m(1-\varepsilon/\varepsilon_0)} \Rightarrow \frac{\partial \sigma}{\partial \varepsilon} = -\frac{D_0 m}{b} (1 - e^{-b}) e^{m(1-\varepsilon/\varepsilon_0)}
\]

where \(m\) is a material parameter related to the softening rate. And the corresponding damage softening stiffness \(D_s\) and the virtual damage scalar \(\Omega\) become

\[
D_s = (1 - \Omega) D_0 = -\frac{D_0 m}{b} (1 - e^{-b}) e^{m(1-\varepsilon/\varepsilon_0)}, \text{ where } \Omega = 1 + \frac{m}{b} (1 - e^{-b}) e^{m(1-\varepsilon/\varepsilon_0)}
\]

The uniaxial stress-strain response corresponding to this particular constitutive model is shown in Fig. 4 for the case of \(\varepsilon_0 = 0.3\), \(b = 5\) and \(m = 1\).
This damage modeling scheme was incorporated into the finite element network model by modifying the micro-frame element stiffness matrix given in equation (2). Using the uniaxial relation (9), the normal and tangential damage stiffness terms for the hardening behavior can be separately written as

\[
\left(K_{nn}\right)_s = K_{nn} e^{-b(\Delta u_n / \Delta U_n)}, \quad \left(K_{nn}\right)_s = K_{nn} e^{-b(\Delta u_n / \Delta U_n)}
\]

and using equation (12) the corresponding normal and tangential damage softening stiffnesses are given as

\[
\left(K_{nn}\right)_s = -(K_{nn} m / b)(1 - e^{-b}) e^{m(1-\Delta u_n / \Delta U_n)}
\]

\[
\left(K_{nn}\right)_s = -(K_{nn} m / b)(1 - e^{-b}) e^{m(1-\Delta u_n / \Delta U_n)}
\]

where \(\Delta u_n\) and \(\Delta u_t\) are the normal and tangential accumulated relative displacements and \(\Delta U_n\) and \(\Delta U_t\) are the normal and tangential displacement softening criteria. Thus the micro-frame element damage stiffness matrix \([K_s]\) is constructed from equation (2) by replacing \(K_{nn}\) and \(K_{tt}\) with \((K_{nn})_s\) and \((K_{tt})_s\).

The initiation of binder softening behavior for tension, compression and shear is governed by softening criteria based on accumulated relative displacements between particle pairs. A simple and convenient scheme to determine the softening criteria is based on using the dimensions of the inter-particle binder geometry in the form

\[
\Delta U_n^{(t)} = c_n h_n
\]

\[
\Delta U_n^{(c)} = c_n h_n
\]

\[
\Delta U_t = c_n w
\]

where \(c_n, c_{nc}, c_u\) represent tension, compression and shear softening factors. These material constants correspond to the critical strength and can be determined from experimental data. Since the cementation geometry \(h_n\) and \(w\) will in general be different for each particle pair, it is expected that each element will have different softening criteria related to its local microstructure.

This model binder damage behavior is a result of the material’s defects including microcracks, voids and joints. Additional defects in the multiphase asphalt system could include interface cracks between the aggregates and binder, commonly caused by settlement of aggregate during compaction. As external loading is increased, existing microcracks coalesce to form finite cracks thus leading to interior failure of the binder or interfacial debonding with the aggregates. When the two-particle binder element is subjected to tension and/or shear force, the element was allowed to soften (interfacial damage) after the loading exceeded an appropriate bond strength. Since tension and shear behaviors are coupled, the binder element would lose all its tension and shear stiffness after the element has interfacially softened. Compressional damage softening is viewed as the Mode-II fracture behavior of microcracks inclined to the loading direction.

For compressional behavior between particle pairs, the cementation spacing will decrease with load increment. Eventually, softening behavior will be initiated and the total element stiffness will significantly decrease. This will lead to the closing of the cementation gap when \(\Delta U_n^{(c)} = h_n\), thus creating contact between the aggregates. At this point the element normal stiffness must be modified to account for this change of physics. The aggregate-to-aggregate stiffness would be significantly higher than the cementation elastic stiffness and currently the model uses a contact stiffness three orders of magnitude larger than the elastic stiffness.

This softening modeling scheme was incorporated into the ABAQUS finite element code using the nonlinear User Defined Element (UEL) subroutine. UEL subroutine would determine the
compression and tension force in each element and perform the required damage calculations in the normal and tangential directions. In the ABAQUS analysis, displacement control boundary conditions were employed and the Modified Riks method was used in order to provide a more stable solution scheme. Also, because aggregate (nodal) displacements became sizeable, the mesh geometry was updated during each load increment.

4. INDIRECT TENSION TEST SIMULATIONS
The indirect tension test (IDT) involves the compressional loading of a cylindrical specimen along its diameter, and is commonly used to determine the tensile or splitting strength of bituminous materials. Since this test encompasses softening failure and fracture damage, it appears to be well suited for micromechanical simulation using our damage model. Under the assumption of uniform loading through the thickness, a two-dimensional circular cross-section may be used for numerical simulation. Several numerical IDT samples have been created using a MATLAB material generating code specially developed for this modeling effort. The generating code provides a convenient numerical routine to spatially distribute particles of general elliptical size and to distribute binder cement between adjacent aggregate pairs. The code allows user control over the details of the generated microstructure or fabric, and this establishes the nature and connectivity of the finite element mesh. Finally the code generates the required input files needed for the finite element simulation program. Details on this material generating code have been previously given (25). Figure 5 illustrates three particular models that have been developed for simulation. All models have approximately the same overall diameter of 101mm (4in) and thickness of 63mm (2.5in), and this was chosen to allow comparison with previously collected experimental test data on standard 4in samples. Model 1 had 65 particles (in four particle size groupings) resulting in 201 finite elements. Model 2 used a variable aggregate size distribution of 71 circular particles in groupings of 14, 11, 7 and 4mm to approximate an actual sample gradation curve. This resulted in 232 model elements. Model 3 had 96 particles from groups of 14, 11, 7, 4 and 2mm and this gave 286 elements. Thus each of the three generic models had somewhat different internal microstructure, and further differences were created through variation in the binder moduli and damage parameters. Model boundary conditions constrain both horizontal and vertical displacements of the bottom pair of aggregates, while the top particle pair accept the applied vertical displacement loading.

A series of numerical IDT simulations were conducted for each of the three models using different values of binder softening factors. Figure 6 shows the IDT sample simulation response of vertical load versus displacement for Model 1 using three different compression softening factors. Since each case had identical elastic and hardening parameters, the initial hardening responses are essentially the same. However, as the compression softening factor is increased, less softening behavior is generated and these cases will produce a higher maximum load as shown. Similar results have been found for Models 2 and 3. It is noted that Model 3 with the higher percentage of fine material gives a slightly stiffer response in comparison to Model 2. Numerical simulation results for these three models have also been compared with experimental data for a particular asphalt mix containing 30% of recycled product. Figure 7 shows the model comparisons with data from three IDT tests. It is evident that the simulations compare favorably with the data, thus indicating that the softening damage model can be used to predict such behavior.

In order to investigate the nature of the microstructural softening/damage processes within an IDT sample, a special series of numerical and experimental tests were conducted. The simulations involved the three models previously shown in Figure 5. The model assumes that there exists a continuous distribution of defects in the binder material. This defect field is taken to grow with the material deformation, and this results in a softening response of particular elements in the FEM network. As per equations (13) and (14), this softening behavior can affect the compression, tension and shear behavior of the element. As previously mentioned, in asphalt material this evolving damage process involves the growth of microcracks to form macrocracking and/or aggregate debonding. Each of the three models was subjected to incremental loading, and during this process all elements within the model were monitored for softening behavior. The element softening evolution and load-displacement response for each model are shown in Figures 8, 9 and 10. For each model, the initial onset of sample loading is
shown in Figure (a) and this also indicates the initial tension and compression behavior within the finite element network. It was found that the largest compressional behavior occurred in vertical elements near the loaded centerline, while horizontal elements in this region perpendicular to the loading direction had the greatest tension loads. Later loading steps are shown in Figures (b), (c) and (d), and the location of these loading steps for each model are illustrated on the overall sample load-displacement plot. Early softening elements at loading step (b) did not significantly affect the stiffness, and models had nonlinear hardening behavior. Step (c) just past the model critical strength, generated small softening behavior, while step (d) had extensive softening as reflected in the large number of softening elements. It is observed that the evolution of damage occurs in the central portions of the model sample where the element loadings are the greatest. Preliminary photographic data from an actual IDT test was collected on the behavior of the surface aggregates and binder damage patterns. One particular photograph is shown in Figure 11 which illustrates the total softening (fracture) behavior along an irregular path through the binder material. The damage simulation results in Figures 8-10 qualitatively agree with the results shown in the photograph. Further work is needed to develop a relation between the damage growth and how it will specifically lead to binder fracture.

5. SUMMARY AND CONCLUSIONS
A two-dimensional micromechanical model has been developed to simulate the behavior of asphalt concrete. The material microstructure composed of aggregates, binder cement and air voids was simulated with an equivalent finite element network that represents the load-carrying behavior between aggregates in the multiphase material. These network elements were specially developed from an elasticity solution for cemented particulates. Incorporating a damage mechanics approach with this solution allowed the development of a softening model capable of predicting typical global inelastic behavior found in asphalt materials. This theory was then implemented within the ABAQUS FEA code using the User Defined Element subroutine. In order to create material models, a numerical material generating code was developed that constructs aggregate-binder systems with varying degrees of microstructure.

A series of three models of indirect tension samples were numerically generated. Each model sample was numerically subjected to the loading and constraints of typical IDT tests. Simulation results were then compared with collected experimental data. The overall sample load-deformation simulations compared favorably with the data. The damage model was able to correctly predict the extensive softening behavior found in actual asphalt materials. Finally a brief investigation was made into the evolution of the internal micro-damage within the IDT models. During incremental loading of the sample, all elements within the model were monitored for softening and the evolution of this behavior was recorded. Photographic data from an actual IDT test was collected on the behavior of the surface aggregates and binder damage patterns. Comparisons of the model damage evolution with the experimental photographic data showed reasonable qualitative comparison. Future work will pursue in more detail these damage comparisons between the micromechanical model and data on real asphalt concrete. Simulations will also be conducted on models with more realistic aggregate microstructure including numerical generated models from surface scan data of actual asphalt samples. The current two-dimensional model is limited to assume uniform behavior through the thickness of the simulated sample. Clearly this assumption is not accurate and a three dimensional extension of the current model is planned.

ACKNOWLEDGEMENTS
The authors would like to acknowledge support from the Transportation Center at the University of Rhode Island under Grants 99-08 and 00-38. Additional support was also provided from the Northeast Asphalt Institute and Cardi Construction Corporation.

REFERENCES

LIST OF FIGURES

FIGURE 1. Multi-phase asphalt material.
FIGURE 2. Asphalt modeling concept.
FIGURE 3. Cement binder between two adjacent particles.
FIGURE 4. Uniaxial stress-strain response for damage model.
FIGURE 5. IDT microstructural models.
FIGURE 6. IDT simulations of Model 1 for different compression softening factors.
   Model parameters: $E = 71.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{tt} = 0.2$, $c_{nt} = 0.2$.
FIGURE 7. Comparison of IDT model simulations with experimental data.
   Model 1 parameters: $E = 71.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$.
   Model 2 parameters: $E = 63.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$.
   Model 3 parameters: $E = 63.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$.
FIGURE 8. Model 1 softening element evolution and load vs displacement curve in IDT simulation.
FIGURE 9. Model 2 softening element evolution and load vs displacement curve in IDT simulation.
FIGURE 10. Model 3 softening element evolution and load vs displacement curve in IDT simulation.
FIGURE 11. Typical failure geometry in an actual IDT sample.
FIGURE 1 Multi-phase asphalt material.
FIGURE 2 Asphalt modeling concept.
FIGURE 3 Cement binder between two adjacent particles.
FIGURE 4 Uniaxial stress-strain response for damage model.

\[ \frac{\sigma}{\sigma_0} = 0.3 \quad m = 1 \]

\[ b = 5 \]
FIGURE 5 IDT microstructural models.

Model 1
65 particles, 201 elements

Model 2
71 particles, 232 elements

Model 3
96 particles, 286 elements
FIGURE 6 IDT simulations of Model 1 for different compression softening factors. Model parameters: $E = 71.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nt} = 0.2$, $c_{nt} = 0.2$. 
FIGURE 7 Comparison of IDT model simulations with experimental data.

Model 1 parameters: $E = 71.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$.
Model 2 and 3 parameters: $E = 63.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$. 
FIGURE 8 Model 1 softening element evolution and load vs displacement curve in IDT simulation. Model 1 parameters: $E = 71.5$MPa, $\nu = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$. 

(a) $\Delta = 0.4mm$  
(b) $\Delta = 1.0mm$  
(c) $\Delta = 2.5mm$  
(d) $\Delta = 4.0mm$
FIGURE 9 Model 2 softening element evolution and load vs displacement curve in IDT simulation. Model 2 parameters: $E = 63.5$ MPa, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{nl} = 0.2$, $c_{nt} = 0.2$. 

(a) $\Delta = 0.4\text{mm}$ 
(b) $\Delta = 1.0\text{mm}$ 
(c) $\Delta = 2.5\text{mm}$ 
(d) $\Delta = 6.0\text{mm}$
FIGURE 10 Model 3 softening element evolution and load vs displacement curve in IDT simulation. Model 3 parameters: $E = 63.5\text{MPa}$, $v = 0.3$, $b = 2$, $m = 0.8$, $c_{nc} = 0.2$, $c_{tt} = 0.2$, $c_{nt} = 0.2$. 

(a) $\Delta = 0.4\text{mm}$  
(b) $\Delta = 1.0\text{mm}$  
(c) $\Delta = 2.5\text{mm}$  
(d) $\Delta = 6.0\text{mm}$
FIGURE 11 Typical failure geometry in an actual IDT sample.