

MIDAS-VT

MICROSTRUCTURE INELASTIC DAMAGE ANALYSIS SOFTWARE-VIRTUAL TESTER

University of Nebraska-Lincoln

MIDAS-VT User's Guide

Microstructure Inelastic Damage Analysis Software-Virtual Tester

Prepared by:

Dr. Yong-Rak Kim (UNL)

Dr. David Allen (TTI)

Dr. Dallas Little (TAMU)

Dr. Francisco Aragao (UFRJ-COPPE)

Keyvan Zare-Rami (UNL)

October 2018

Contents

1. Overview	4
2. Starting MIDAS.....	6
3. Preprocessor	7
3.1 Preprocessor-Case 1	9
Specimen Geometry	10
Microstructure	11
Mesh Generation	12
Export Output.....	13
3.2 Preprocessor-Case 2	15
Specimen Geometry	15
Input Mesh Data.....	15
Mesh Generation	16
Export Output.....	16
4. Processor	18
Model Data	19
Analysis Information	19
Test Information	19
Material Information	19
5. Post processor.....	23
Constitutive behavior	25
A. 1 Boundary value problem.....	25
A. 2 Linear elasticity	25
A. 3 Linear viscoelasticity.....	26
Fracture modeling.....	27
Power function damage model	28
Gaussian function damage model.....	28
6. Reference	29

1. Overview

MIDAS-VT software is a 2D finite element (FE) solver capable of analyzing homogenous and heterogenous media with handling the crack propagation. The software is developed as a standalone package with an intuitive graphical interface. This package is designed in three modules: *Preprocessor*, *Processor*, *Postprocessor* which work internally with the FE solver through input and output files (see dashed box in Figure 1.1). *Preprocessor* part generates FE model which contains mesh and boundary condition data. *Preprocessor* module is tailored to FE model of six common test configurations in infrastructure materials' field: Simple tension test, Simple shear test, Three-point bending beam test, Four-point bending beam test, Semi-circular bending beam test and Indirect shear test. These FE models can be used later in *Processor* where material properties and other test conditions are defined. Also, *Postprocessor* feature is provided within this package to display and visualize the result and export required graphs. The descriptions and implementation details of each modules are presented in the following chapters.

Comments and Questions:

If there is any question about the package or this guide, or run into problems, please contact to

yong-rak.kim@unl.edu

keyvan.zare@gmail.com

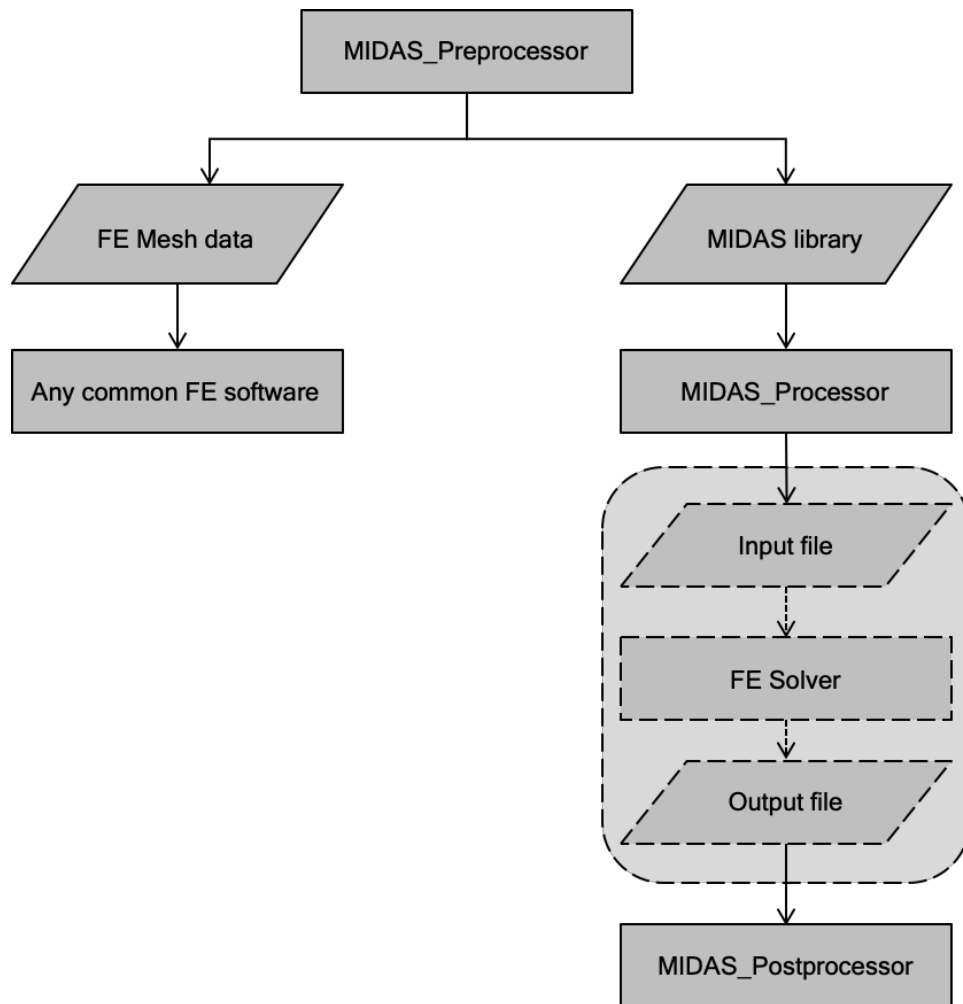


Figure 1.1. Flowchart of MIDAS-VT software

2. Starting MIDAS

1. File in the package:

Gallery folder, FESolver.exe, MIDAS_VT_Pre.exe, MIDAS_VT_Pro.exe, splash.PNG

2. Prerequisites:

Verify the Windows 64-bit version of MATLAB Runtime version 9.5 (R2018b) is installed on your computer. Or download it from following link:

<http://www.mathworks.com/products/compiler/mcr/index.html>

3. Run target software:

MIDAS_VT_Pre.exe, MIDAS_VT_Pro.exe

Note: all the output messages will be stored in STATUS.txt for future reference.

Note: The execution may take several minutes.

3. Preprocessor

The *preprocessor* module is designed to generate the FE model which includes mesh information and boundary condition data. this module is customized according to the following test configurations (see Figure 3.4):

- Simple tension test
- Simple shear test
- Three-point bending beam test
- Four-point bending beam test
- Semi-circular bending test
- Indirect tension test

Overall flow of *preprocessor* is illustrated in Figure 3.1. To generate a model, the user needs to run *Preprocessor*. *Preprocessor* starts with a pop-up window, Figure 3.2, offering two options. The first option should be used when the user is generating the FE model directly from the sample image or sample geometry. This option corresponds to Case I in Figure 3.1 and Figure 3.2. The second option is only able to add cohesive elements into regular FE mesh which is generated in advance. This option corresponds to Case II in Figure 3.1 and Figure 3.2.

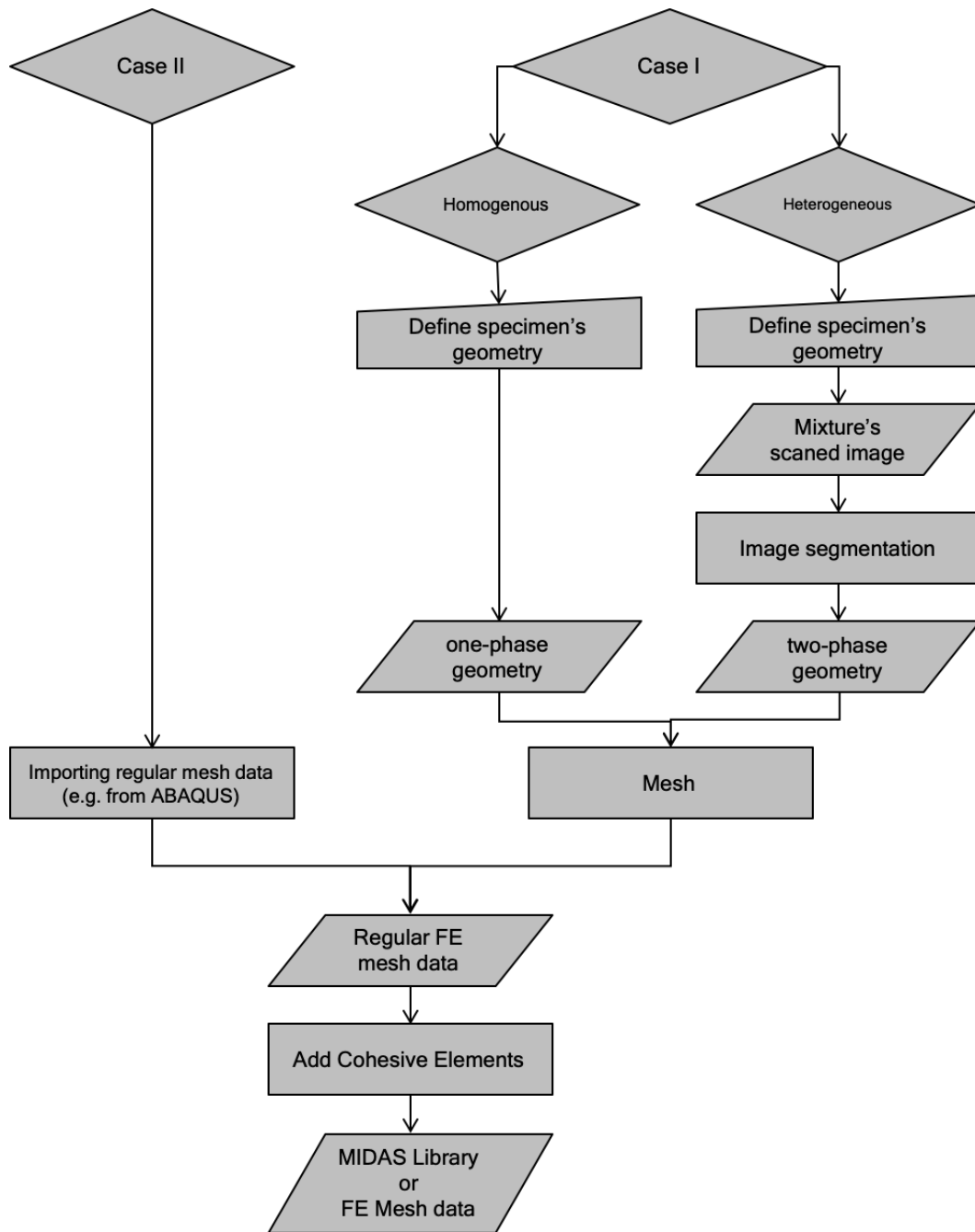


Figure 3.1. Flowchart of MIDAS preprocessor

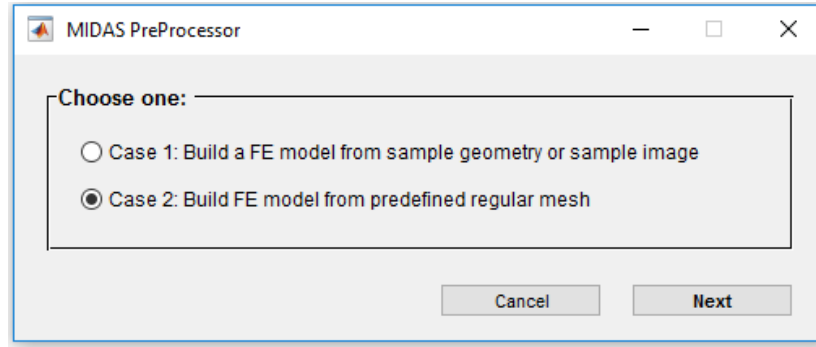


Figure 3.2. Starting preprocessor interface

3.1 Preprocessor-Case I

Case I directs the user to the window shown in Figure 3.3. There are four steps to generated the model: 1. Specimen Geometry; 2. Microstructure; 3. Mesh; 4. Export Output. For each section there are number of inputs and actions required which are described bellow.

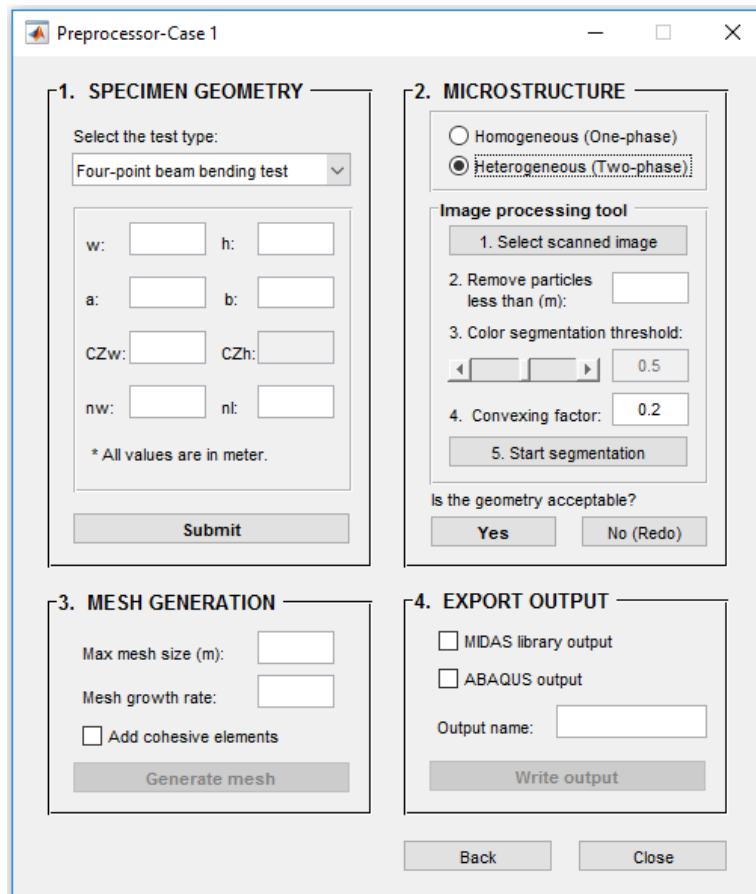


Figure 3.3. Preprocessor- Case I interface

Specimen Geometry

At this panel, the user needs to specify the test type and provide the necessary dimensions related to the specimen. The required dimensions are illustrated in Figure 3.4.

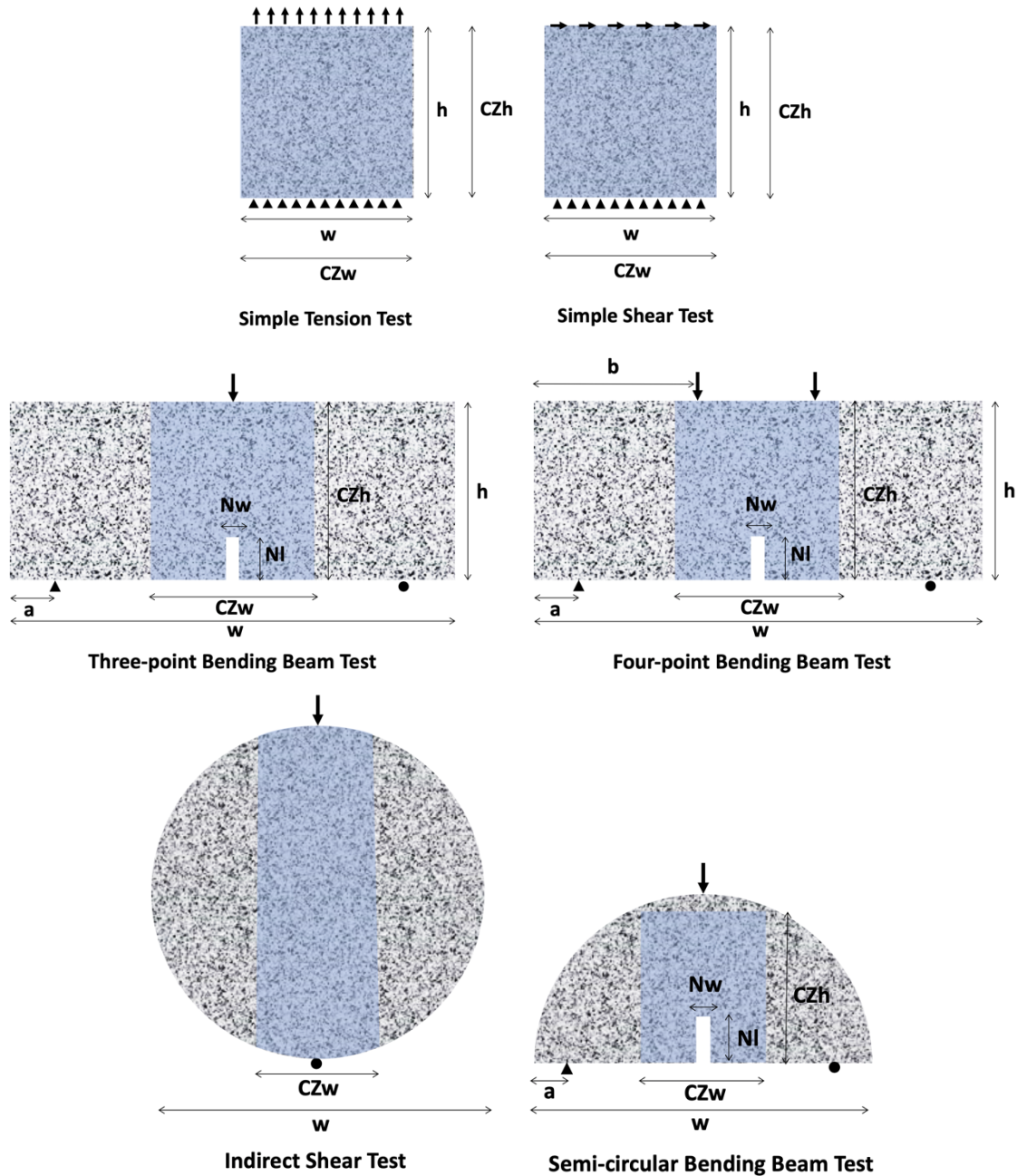


Figure 3.4. Tests' configurations (highlighted area represents user defined potential cracking region).

Microstructure

MIDAS *Preprocessor* is capable of creating either homogenous (one-phase) or heterogenous (two-phase) models. In homogenous case, the software generates model geometry using specimen dimensions provided in the first section. Heterogenous media, in current version of MIDAS, is defined as a two-phase media which is defined as non-contact random particles being scattered within a matrix phase. An image processing tool is provided in MIDAS *Preprocessor* to obtain mixture's microstructure from the actual image of a media. Thus, the user needs to provide the software with the cross-section image of the sample and follow the steps one to five shown in Figure 3.5.

In the cross-section image, particles must be lighter in color than the matrix background. MIDAS *Preprocessor* uses color segmentation method to distinguish particles from matrix phase. *Color segmentation threshold* is a value between 0 to 1, which defines the limit between particles (light color areas) and matrix phase (dark color areas). This value will be calculated automatically, however the user can adjust the value, if necessary. Also, the user is allowed to adjust how to bound the particles' peripheral using *shrink factor*. Setting *shrink factor* to 0 gives a convex hull around the particles while setting it to higher values gives a compact boundary around the particles (the default value is 0.2). The proper values of *color segmentation threshold* and *shrink factor* depends on the image and can be obtained by trial and error. When the image segmentation process is done, the user can compare the result to the actual microstructure (Figure 3.6). If the microstructure is similar enough to the original image, the user can approve the image segmentation result and go to the meshing step by clicking "Yes" button. If there are still mismatches that cannot be improved by *changing color segmentation threshold* and *shrink factor*, the user needs to modify the image manually. The manual image treatment is basically repainting the indistinct areas or separating the connected particles in the image. Figure 3.6 shows an image example after treatment. To modify particles or matrix areas, the selected color's lightness/darkness should approximately match the average intensity value of the corresponding area (particles or matrix). To facilitate the manual treatment procedure, Microsoft Paint software is integrated with MIDAS *Preprocessor*. By clicking "No (redo)" button (see Figure 3.5) the microstructure's image will be opened in Microsoft Paint for additional adjustment. Then, after the image treatment is finished, the user needs to do the image processing, step one to five (see Figure 3.5), once more. This procedure can be continued until the desired accuracy is obtained.

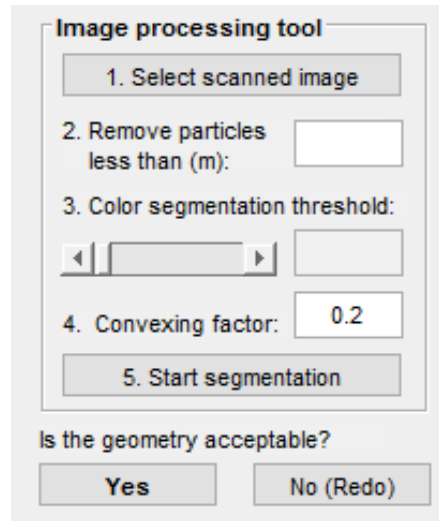


Figure 3.5. Image processing tool interface

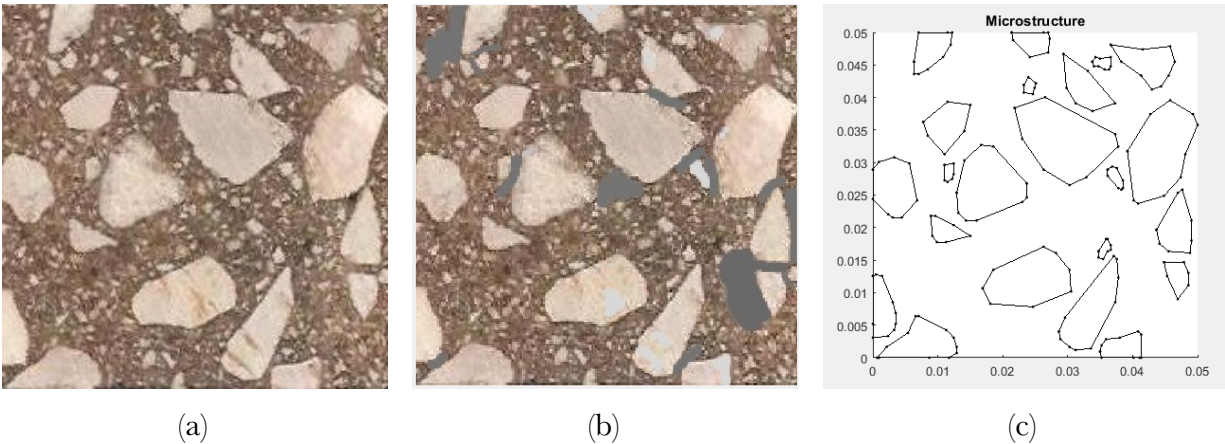


Figure 3.6. (a) Original image, (b) treated image and (c) detected microstructure

Mesh Generation

The meshing module is designed to mesh the geometry using three-nodes elements. The user can adjust the *maximum mesh size* and *mesh growth rate*. The later one is specified as a scalar strictly between 1 and 2 and defines how the mesh size increases away from small parts of the geometry. The meshing module comes with an automated Cohesive Element insertion feature. Cohesive Element allows to simulate crack initiation and propagation within FE framework. Generally, cohesive elements are zero thickness elements that links adjacent nodes of regular neighbor elements to each other (Zare Rami Keyvan et al 2017). The *Add Cohesive Element* checkbox permits the user to add cohesive element between regular FE elements within the cracking region which is defined in

Specimen geometry. The Cohesive Elements will be added within matrix phase, which is called cohesive element, and particle-matrix interface, which is called adhesive element. Cohesive Elements will be labeled according to their types, i.e. adhesive and cohesive, which allow the user to assign corresponding properties to them later in *Processor*.

Export Output

The output produced by Preprocessor includes mesh data and boundary condition data associated with the test type. The mesh data contains node coordinates and element connectivity matrix. The output data can be exported in two formats, one format is compatible with MIDAS *Processor*, which is saved as a *.mat* file, another one is compatible with common FE software such as ABAQUS, which is saved as a *.inp* file. The format of the later one is as follows:

- Nodes coordinate matrix

```
*Node
Entry 1      Entry 2      Entry 3
... (continued)
```

Entry 1: node ID

Entry 2: x-coordinate of node

Entry 3: y-coordinate of node

- Bulk elements connectivity matrix

```
*Element, type= Entry 0
Entry 1      Entry 2      Entry 3      Entry 4
... (continued)
```

Entry 0: bulk element type, which is 3-node

Entry 1: element ID

Entry 2: 1st node number in global numbering system

Entry 3: 2nd node number in global numbering system

Entry 4: 3rd node number in global numbering system

- Cohesive element connectivity matrix

```
*Element, type= Entry 0
Entry 1      Entry 2      Entry 3      Entry 4      Entry 5
... (continued)
```

Entry 0: cohesive element type, which is 4-node cohesive

Entry 1: element ID

Entry 2: 1st node number in global numbering system

Entry 3: 2nd node number in global numbering system

Entry 4: 3rd node number in global numbering system

Entry 5: 4th node number in global numbering system

- Element sets which list the elements ID within each set

```
*Elset, elset= Entry 0  
Entry 1      [Entry 2]      ...  
... (continued)
```

Entry 0: Set ID, which are: Phase1, which represents elements in matrix phase, Phase2, which represents elements in particles, Cohesive Elements, and Adhesive Elements

Entry <i>: element ID

- Node sets which list the nodes related to each boundary condition

```
*Nset, nset= Entry 0  
Entry 1      [Entry 2]      ...  
... (continued)
```

Entry 0: boundary condition nodes which is different for each test:

- Simple tension test
 - TT_T which corresponds to the top boundary nodes
 - TT_B which corresponds to the bottom boundary nodes
 - TT_L which corresponds to the left boundary nodes
- Simple shear test
 - ST_T which corresponds to the top boundary nodes
 - ST_B which corresponds to the bottom boundary nodes
- Three-point bending beam test
 - TPBT_LS which corresponds to the left support nodes
 - TPBT_RS which corresponds to the right support nodes
 - TPBT_LP which corresponds to the loading point nodes
- Four-point bending beam test
 - FPBT_LS which corresponds to the left support nodes

- FPBT_RS which corresponds to the right support nodes
- FPBT_LLP which corresponds to the left loading point nodes
- FPBT_RLP which corresponds to the right loading point nodes
- Semi-circular bending beam test
 - SCBT_LS which corresponds to the left support nodes
 - SCBT_RS which corresponds to the right support nodes
 - SCBT_LP which corresponds to the loading point nodes
- Indirect tension test
 - ITT_BS which corresponds to the bottom support nodes
 - ITT_TLP which corresponds to the top loading point nodes

Entry <i>: node ID

3.2 Preprocessor-Case II

This feature is provided for a case in which the model is meshed in advance by regular 3-nodes FE elements. Choosing case II in Figure 3.2, directs the user to *Preprocessor-Case II* window, as shown in Figure 3.7. To generate the model the user need to finish steps one to four.

Specimen Geometry

Similar to the *preprocessor-case I* the user must select test type which is followed by inputting specimen dimensions as shown in Figure 3.4. There is no need to enter cracking region dimensions in this section. The cracking region is specified by introducing elements within the cracking region as part of a separate set (see next section, input mesh data).

Input Mesh Data

The mesh data must include the following information and be written in a single *.txt* or *.inp* file using the following format:

- Nodes coordinate matrix

```
*Node
Entry 1      Entry 2      Entry 3
... (continued)
```

Entry 1: node ID

Entry 2: x-coordinate of node

Entry 3: y-coordinate of node

- Bulk elements connectivity matrix

```
*Element, type= Entry 0
Entry 1      Entry 2      Entry 3      Entry 4
... (continued)
```

Entry 0: bulk element type, which is 3-node

Entry 1: element ID

Entry 2: 1st node number in global numbering system

Entry 3: 2nd node number in global numbering system

Entry 4: 3rd node number in global numbering system

- Elements' sets which list the elements ID within each set

```
*Elset, elset= Entry 0
Entry 1      [Entry 2]      ...
... (continued)
```

Entry 0: Set ID, which are: Phase1, which represents elements of matrix phase, Phase2, which represents elements of particles, CZ, which represents elements in cracking region

Entry <i>: element ID

Comment: in common FE softwares such as ABAQUS, to group elements in different sets and assign associated tag to each group, *partitioning* and *element set* tools is used.

Comment: the user doesn't need to provide boundary condition nodes, they will be obtained automatically using specimen geometry.

Mesh Generation

This module adds zero thickness cohesive elements between bulk elements which are within cracking region (Zare Rami Keyvan et al 2017).

Export Output

This step is identical to *Export output* in *Preprocessor-case I* (see section 3.1).

Preprocessor-Case 2

1. SPECIMEN GEOMETRY

Select the test type:

Four-point beam bending test

w: h:

a: b:

* All values are in meter.

Submit

2. INPUT MESH DATA

To read the regular mesh data (without cohesive element) which is generated beforehand.

Import mesh data: Select

Select delimiter: Space

Submit

3. MESH GENERATION

Adding zero thickness cohesive elements between regular FE elements.

Add cohesive elements

4. EXPORT OUTPUT

☐ MIDAS library output

☐ ABAQUS output

Output name:

Write output

Back Close

Figure 3.7. Preprocessor- Case II interface

4. Processor

The FE model generated by MIDAS *Preprocessor* which is already stored as MIDAS *Library* file is imported in MIDAS processor to define the FE model. The MIDAS *Library* file only includes mesh data and boundary condition data. *Processor* module allows the user to specify test loading condition, assign material properties and run the simulation (Figure 4.2). This process is categorized in 4 steps which are described as follows.

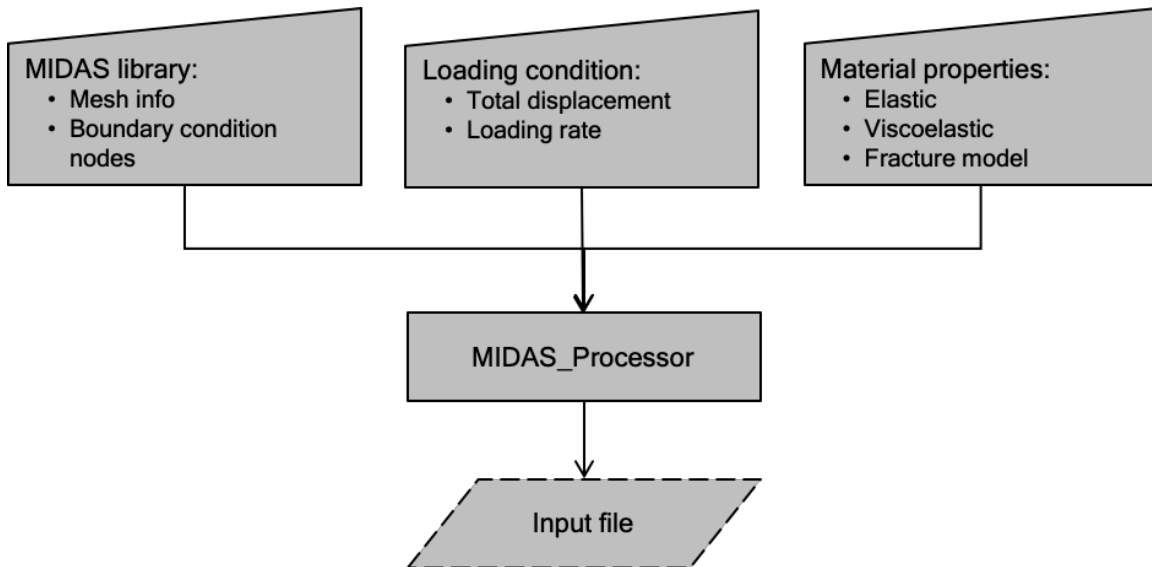


Figure 4.1. Flowchart of MIDAS-Processor

The screenshot shows the MIDAS Processor interface with the following sections:

- 1. MODEL DATA**: Contains a "Select mesh data:" label and a "Select" button.
- 2. ANALYSIS INFORMATION**: Contains an "Analysis Type" section with two radio buttons: "Plane stress" (selected) and "Plane strain".
- 3. TEST INFORMATION**: Contains a "Displacement Control" section with three input fields: "Total displacement (m):", "Total time (s):", and "Time increment (s):".
- 4. MATERIALS INFORMATION**: Contains two sub-sections:
 - Constitutive Behavior**: Includes "No. of materials:" (input field), "1. Material:" (dropdown), "2. Const. behavior:" (dropdown), and "3. Input properties:" (button with ">>>").
 - Fracture Properties**: Includes "No. of interfaces:" (input field), "1. Interface:" (dropdown), "2. Fracture model:" (dropdown), and "3. Input properties:" (button with ">>>").

At the bottom of the interface are two large buttons: "1. Run" and "2. Post Processing".

Figure 4.2. Processor interface

Model Data

In the first step, the user needs to load the general information of the model including test type, mesh information and boundary condition data. This can be done directly by importing *MIDAS Library* file, which is in .mat format, generated using *Preprocessor* (see section 3).

Analysis Information

Either plane stress or plane strain can be chosen.

Test Information

At this stage, MIDAS is only able to simulate displacement controlled tests. To define loading condition, *Total displacement*, which acts on loading boundary points (see Figure 3.4) and *Total loading time* is needed. As it is clear the loading rate will be calculated internally using the following formula:

$$\text{Loading rate} = \text{Total displacement} / \text{Total time}$$

MIDAS uses a fixed time incremental formulation to solve the nonlinear problem. This value must be specified here.

Material Information

There are two separate panels designed for inputting material models, one is for bulk elements' constitutive models, and the other is for cohesive element traction separation models. Number of

material types and number of interfaces type will be automatically filled from the MIDAS *library* that is loaded in *Model Data* section. Number of material types can be either 1 or 2 which corresponds to homogenous model or heterogenous model (at this version of MIDAS-VT, only two-phase heterogeneity is included) respectively. Number of interphases can be either 0 or 2. Zero corresponds to a model without cohesive element, and the number 2 corresponds to a model with cohesive element. The number 2 implies the presence of cohesive and adhesive elements.

Figure 4.3 shows two panels of the Material information panel. Panel (a) is titled 'Constitutive Behavior' and contains four input fields: 'No. of materials:', '1. Material:', '2. Const. behavior:', and '3. Input properties:'. Panel (b) is titled 'Fracture Properties' and contains four input fields: 'No. of interfaces:', '1. Interface:', '2. Fracture model:', and '3. Input properties:'. Both panels have a light gray background and a thin black border.

Figure 4.3. Material information panel.

For each material, steps one to three must be completed (Figure 4.3-a). Firstly, selecting material type, secondly selecting constitutive behavior type, and finally input associated properties. In MIDAS, two constitutive behavior is provided for bulk elements: isotropic linear elastic and isotropic linear viscoelastic.

- Isotropic linear elastic

When the constitutive behavior is isotropic linear elastic, the window shown in Figure 4.4 pops up to enter the elastic modulus and Poisson's ratio (see Appendix A).

Figure 4.4 shows a dialog box titled 'LinearElastic'. It has a standard Windows window title bar with a minimize button, a maximize button, and a close button. The main content area is titled 'ISOTROPIC LINEAR ELASTIC' and contains two input fields: 'Elastic Modulus (Pa):' with the value '5200000000' and 'Poisson's Ratio:' with the value '0.15'. At the bottom of the dialog box, there are two buttons: 'Submit' and 'Close'.

Figure 4.4. Inputting linear elastic properties.

- Isotropic linear viscoelastic

When the constitutive behavior is isotropic linear viscoelastic, the window shown in Figure 4.5 pops up to input the number of Prony series, Prony series coefficients and Poisson's ratio (see Appendix A).

	CL (Pa)	Rho
Inf	7020000	-
1	245648430	0.00003
2	422264070	0.0003
3	399318930	0.003
4	251827650	0.03
5	69096874	0.3
6	22585797	3
7	7816581	30
8	3459600	300
9		
10		

Figure 4.5. Inputting linear viscoelastic properties.

Similarly, for each interface steps 1 to 3 must be completed (Figure 4.3-b). Firstly, selecting interface type, secondly selecting the fracture model, and finally input associated properties. At this stage, MIDAS provides only one type of fracture model for both cohesive and adhesive elements which is called viscoelastic fracture model (see Appendix B).

- Viscoelastic fracture model

Viscoelastic fracture model parameters can be entered through the window shown in Figure 4.6. Alongside with the Prony series parameters, critical displacement and initial stress, the user needs to define damage evolution function parameters. Regarding to the material response two damage evolution function, *Power function* and *Gaussian function* are provided (see Appendix B).

The screenshot shows the VECohesive software window with the following parameters:

VISCOELASTIC FRACTURE MODEL

Cohesive Zone Parameters

No. of prony series: 5

	EC (Pa)	Rho
Inf	1400	-
1	56058696	0.0014
2	11814777	0.014
3	2314771	0.14
4	269841	1.4
5	33929	14
6		
7		
8		
9		

Sigma N: Delta N:

Sigma T: Delta T:

Damage Evolution

Damage evolution law: Power functio... (dropdown)

C: m: n:

A: m: L:

D:

Submit Close

Figure 4.6. Inputting viscoelastic fracture model properties

Start simulation

When the parameters are inputted correctly, the user can start the simulation by pushing *Run* button in Figure 4.2. The *Processor* will generate an input file and run the *FESolver*. The simulation status will be updated in *status.txt* within the working directory after each solution step. This file also can be checked to see whether running is over or not.

5. Post processor

When the simulation is done, the user can visualize the result through *Post Processing* button in Figure 4.2. The user can select the target graph from the popup menu and plot it (Figure 5.1). Also, the user can generate the stress contour video using *Contour* panel. After selecting desire stress, there is need to specify contour levels, then wait until *Postprocessor* calculates color-bar's Min. and Max. values. These values can be edited by user's choice. The video will be created by plotting each frame on the screen (Figure 5.2). Avoid closing the frames until the “operation completed” appears).

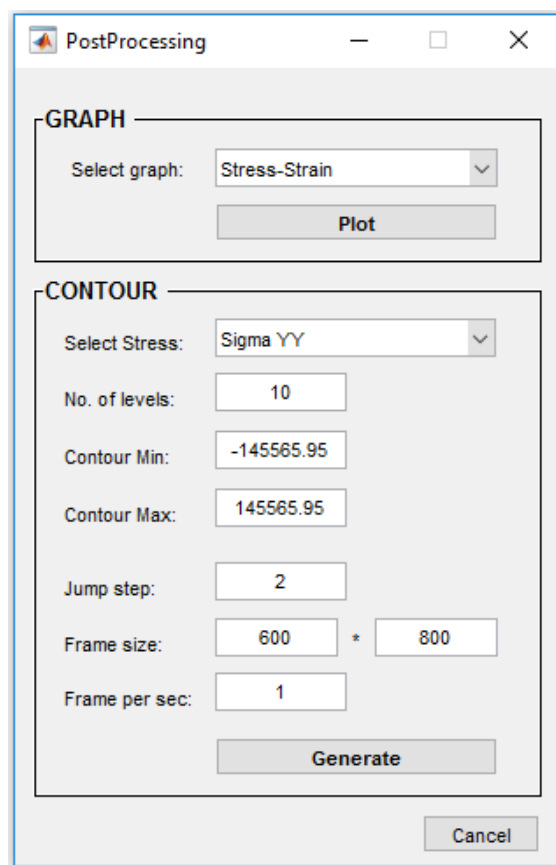


Figure 5.1. Post Processor

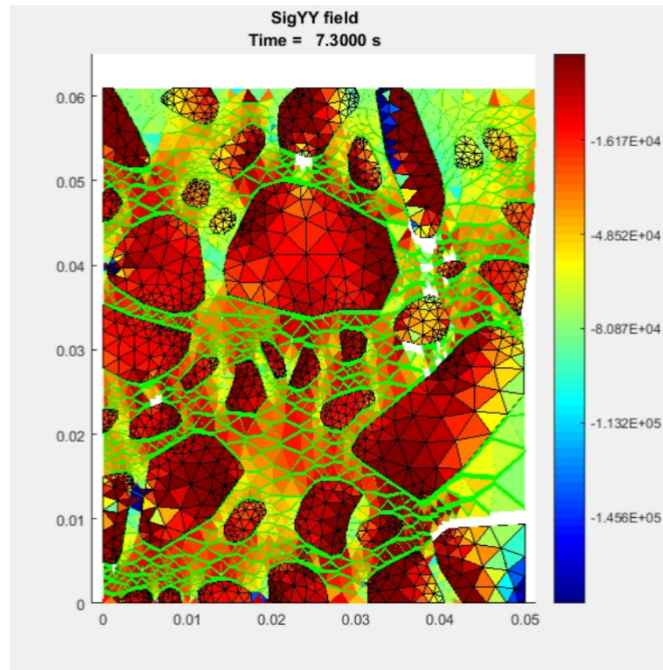


Figure 5.2. Stress contour

Appendix A. Constitutive behavior

A. 1 Boundary value problem

In the absence of body forces, inertial effects, and large deformations, an initial boundary value problem of the domain can be well-posed by an appropriate initial-boundary conditions and a set of governing equations: conservation of linear momentum (Eq. [1]), conservation of angular momentum (Eq. [2]), infinitesimal strain-displacement relationships (Eq. [3]), constitutive equations (Eq. [4]), and fracture criteria (Eq. [5]) (see references Zocher, Groves et al. 1997; Arago, Kim et al. 2010; Kim, Arago 2013).

$$\sigma_{ji,i} = 0 \quad \text{in } V \quad \text{Equation 1}$$

$$\sigma_{ji} = \sigma_{ij} \quad \text{in } V \quad \text{Equation 2}$$

$$\varepsilon_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \quad \text{in } V \quad \text{Equation 3}$$

$$\sigma_{ij}(x_m, t) = \Omega_{\tau=-\infty}^{\tau=t} \{\varepsilon_{kl}(x_m, \tau)\} \quad \text{in } V \quad \text{Equation 4}$$

$$\Gamma \geq \Gamma_c \Rightarrow \frac{\partial}{\partial t}(\partial V_I) > 0 \quad \text{in } V \quad \text{Equation 5}$$

where σ_{ji} , stress tensor; ε_{ij} , strain tensor; u_i , displacement vector; Ω is a functional mapping that describes the constitutive behavior at each position in the mixture; Γ , fracture energy release rate at a particular position in the mixture; Γ_c , critical energy release rate; V , volume of the domain, ∂V_I , internal boundary (such as cracks) in the mixture; x_m , spatial coordinates; and t time of interest.

A. 2 Linear elasticity

The isothermal-isotropic linear elastic constitutive behavior that is provided in MODAS-VT can be expressed as:

$$\sigma_{ij} = \frac{E\nu}{(1+\nu)(1-2\nu)} \varepsilon_{kk} \delta_{ij} + \frac{E}{1+\nu} \varepsilon_{ij} \quad \text{Equation 6}$$

where σ_{ij} is the stress tensor, ε_{ij} is the strain tensor, E is the elastic Young's modulus, ν is Poisson's ratio, and δ_{ij} is Kronecker delta.

A. 3 Linear viscoelasticity

Isothermal-isotropic linear viscoelasticity with a time-independent Poisson's is implemented in MIDAS, which is given by

$$\sigma_{ij} = \frac{\nu}{(1+\nu)(1-2\nu)} \int_0^t E(t-\tau) \delta_{ij} \frac{\partial \varepsilon_{kk}}{\partial \tau} d\tau + \frac{1}{1+\nu} \int_0^t E(t-\tau) \delta_{ij} \frac{\partial \varepsilon_{ij}}{\partial \tau} d\tau \quad \text{Equation 7}$$

where $E(t)$ is the viscoelastic stress relaxation modulus which is expressed as:

$$E(t) = E_\infty + \sum_{n=1}^N E_n \exp\left(-\frac{t}{\rho_n}\right) \quad \text{Equation 8}$$

where E_∞ and E_n are spring constants in the generalized Maxwell model, ρ_n is the relaxation time, and N is the number of Maxwell units in the generalized Maxwell model.

Appendix B. Fracture modeling

Crack propagation in the matrix phase is modeled by a cohesive zone model (Figure 3). The cohesive zone can consider gradual damage in the material by employing a traction-separation response.

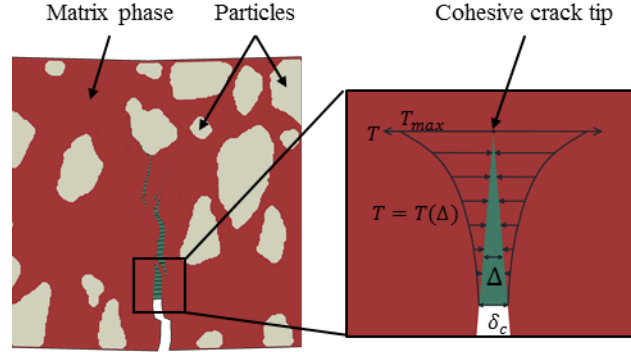


Figure 3. Cohesive zone model

The constitutive relation governing mechanical behavior of the cohesive zone is as follows (Allen, Searcy 2000; Allen, Searcy 2001; Kim, Allen et al. 2007; Kim, Allen et al. 2005; Kim, Allen et al. 2006):

$$T_i(t) = \frac{1}{\lambda(t)} \frac{u_i(t)}{\delta_i} [1 - \alpha(t)] \left[\sigma_i^f + \int_0^t E^c(t - \xi) \frac{\partial \lambda(\xi)}{\partial \xi} d\xi \right] \quad \text{Equation 9}$$

where $i = n$ (normal direction), t (tangential direction); $T_i(t)$, is the cohesive zone area-averaged traction; $\lambda(t)$, the Euclidean norm of the cohesive zone displacements; $u_i(t)$, the cohesive zone displacement; $\delta_i(t)$, the cohesive zone material length parameter; $\alpha(t)$, the internal state variable representing damage evolution characteristics; σ_i^f , the requisite stress level to initiate cohesive zone.

$E^c(t)$, the linear viscoelastic relaxation modulus of the cohesive zone.

$$E(t) = E_\infty + \sum_{n=1}^N E_n \exp\left(-\frac{t}{\rho_n}\right) \quad \text{Equation 10}$$

where E_∞ and E_n are spring constants in the generalized Maxwell model, ρ_n is the relaxation time, and N is the number of Maxwell units in the generalized Maxwell model.

For the 2D case, λ can be separated into normal (opening) and tangential (shear sliding) components, as the following:

$$\lambda(t) = \sqrt{\left(\frac{u_n(t)}{\delta_n}\right)^2 + \left(\frac{u_t(t)}{\delta_t}\right)^2} \quad \text{Equation 11}$$

Power function damage model

There are two phenomenological model is implemented in MIDAS to model damage evolution I material. First one is called *Power function model* which is in a form of power relationship as a function of the strainlike term $\lambda(t)$ and internal state variable $\alpha(t)$ (Yoon, Allen 1999; Allen, Searcy 2001):

$$\dot{\alpha}(\lambda) = C[1 - \alpha]^n[\lambda(t)]^m, \quad \text{when } \frac{d\lambda}{dt} > 0 \text{ and } \alpha < 1 \quad \text{Equation 12}$$

$$\alpha = 0, \quad \text{when } \frac{d\lambda}{dt} \leq 0 \text{ and } \alpha = 1 \quad \text{Equation 13}$$

where C and m are microscale phenomenological material constants governing damage evolution behavior. When $\alpha(t)$ reaches the value of unity, the crack face traction decays to zero, thus resulting in crack extension.

Gaussian function damage model

The second model is in form of a Gaussian function that relates $\dot{\alpha}(t)$ with $\lambda(t)$ and $\dot{\lambda}(t)$:

$$\dot{\alpha}(\lambda, \dot{\lambda}) = A(\dot{\lambda})^m \exp\left[-\frac{(\lambda - \bar{\lambda})^2}{2\Delta^2}\right] \quad \text{Equation 14}$$

where A , m , $\bar{\lambda}$ and Δ are microscale phenomenological material constants governing damage evolution behavior.

6. Reference

- ALLEN, D.H. and SEARCY, C.R., 2000. Numerical aspects of a micromechanical model of a cohesive zone. *Journal of Reinforced Plastics and Composites*, 19(3), pp. 240-248.
- ALLEN, D.H. and SEARCY, C.R., 2001. A micromechanical model for a viscoelastic cohesive zone. *International Journal of Fracture*, 107(2), pp. 159-176.
- ARAGO, F.T.S., KIM, Y., LEE, J. and ALLEN, D.H., 2010. Micromechanical model for heterogeneous asphalt concrete mixtures subjected to fracture failure. *Journal of Materials in Civil Engineering*, 23(1), pp. 30-38.
- KIM, Y., ALLEN, D.H. and LITTLE, D.N., 2007. Computational constitutive model for predicting nonlinear viscoelastic damage and fracture failure of asphalt concrete mixtures. *International Journal of Geomechanics*, 7(2), pp. 102-110.
- KIM, Y., ALLEN, D.H. and LITTLE, D.N., 2005. Damage-induced modeling of asphalt mixtures through computational micromechanics and cohesive zone fracture. *Journal of Materials in Civil Engineering*, 17(5), pp. 477-484.
- KIM, Y., ALLEN, D.H. and SEIDEL, G.D., 2006. Damage-induced modeling of elastic-viscoelastic randomly oriented particulate composites. *Journal of Engineering Materials and Technology*, 128(1), pp. 18-27.
- KIM, Y. and ARAGO, F.T.S., 2013. Microstructure modeling of rate-dependent fracture behavior in bituminous paving mixtures. *Finite Elements in Analysis and Design*, 63, pp. 23-32.
- YOON, C. and ALLEN, D.H., 1999. Damage dependent constitutive behavior and energy release rate for a cohesive zone in a thermoviscoelastic solid. *International Journal of Fracture*, 96(1), pp. 55-74.
- ZOCHER, M.A., GROVES, S.E. and ALLEN, D.H., 1997. A three-dimensional finite element formulation for thermoviscoelastic orthotropic media. *International Journal for Numerical Methods in Engineering*, 40(12), pp. 2267-2288.
- ZARE RAMI KEYVAN, AMELIAN SOROOSH, KIM YONG-RAK, YOU TAESUN, LITTLE DALLAS, Modeling the 3D Fracture-Associated Behavior of Viscoelastic Asphalt Mixtures. *Engineering Fracture Mechanics (Submitted)*.