MATLAB eXtended Finite Element Method (MXFEM)

User's Guide

Matthew Jon Pais
Structural and Multidisciplinary Optimization Group
Mechanical and Aerospace Engineering Department
University of Florida
Gainesville, FL 32611

August 30, 2011

Table of Contents

Chapter 1: Introduction

Modeling crack growth in a traditional finite element framework is a challenging engineering task. Originally the finite element framework was modified to accommodate the discontinuities that are caused by phenomena such as cracks, inclusions and voids. The finite element framework is not well suited for modeling crack growth because the domain of interest is defined by the mesh. At each increment of crack growth, at least the domain surrounding the crack tip must be remeshed such that the updated crack geometry is accurately represented.

Here, a simple two-dimensional plane stress and plane strain XFEM implementation within MATLAB is presented. The extended finite element method (XFEM) along with the level set method can be used to alleviate many of the inconveniences of using the finite element method (FEM) to model the evolution of a crack. Special enrichment functions are added to the traditional finite element framework through the partition of unity framework. For modeling the strong discontinuity of a cracked body two enrichment functions are used. The Heaviside step function represents the discontinuity away from the crack tip, and the linear elastic asymptotic crack tip displacement fields are used to account for discontinuity at the crack tip. The crack is represented independent of the mesh by the enrichment functions which allows for the crack geometry to be updated without a need to create/update a new mesh on the domain. In addition, enrichment functions for a bimaterial crack are presented. For the case of a material interface, an enrichment function is used which combines distance from the weak discontinuity and the absolute value function. Finally, for a void, a step function is used such that the global displacement approximation equals zero within the void.

Crack growth was modeled by combining the maximum circumferential stress criterion or the critical plane approach for predicting the direction of crack growth. A constant crack growth increment, classical Paris model, or a modified Paris model is used to incrementally grow a crack. The stress intensity factors needed for these models were calculated using the domain form of the J-integral interaction integrals.

Chapter 2: Use of MXFEM

The MATLAB code has been prepared such that the user modifies the inputQuasiStatic.m file, then runs xfemQuasiStatic.m from the MATLAB Command Window to solve a quasi-static crack problem. For an optimization problem the user modifies both inputOptimization.m and xfemOptimization.m and runs inputOptimization.m to solve the given problem. A detailed description of the input variables follows as well as a brief summary of the functions which make up the complete code follows.

bimatctipNodes.m

This function calculates the values of the enrichment functions at the nodes enriched with the bimaterial crack tip enrichment function.

boundaryCond.m

This function applies boundary conditions on the domain. Options included in the current package include fixing the bottom of the rectangular domain in the y-direction and bottom left hand corner with respect to x and y, edge crack, half center crack, full center crack, rollers on bottom and left edge, and symmetric about midpoint. For the case of voids, the additional fixed degrees of freedom are also calculated and added to the system of equations.

calcDOF.m

This function calculates the total number of degrees of freedom in the system considering traditional, Heaviside, crack tip, bimaterial crack tip, and inclusion degrees of freedom.

connectivity.m

This function calculates the global XYZ coordinates of all the nodes, defines element connectivity and also begins to build the NODES matrix which keeps track of the numbering for the enriched degrees of freedom.

crackCoord2Length.m

This function creates a vector a which consists of the crack length at each iteration from CRACK. This function is independent of the main code.

crackLength2Cycles.m

This function creates a vector N which consists of the cycle numbers at each iteration from back-calculation from Paris model based on the forward Euler method.

ctipNodes.m

This function calculates the values of the enrichment functions at the nodes enriched with the crack tip enrichment function.

elemStress.m

This function calculates the nodal stress values for the given geometry. If an element by element stress plot is to be made, then the nodal stress values may or may not be averaged based on the user's specifications.

enrElem.m

This function identifies the enriched elements or the enriched elements which will be modified by the reanalysis algorithm.

eqSIFLiu.m

This function calculates the equivalent stress intensity factor according to the method of Liu[1].

eqSIFRhee.m

This function calculates the equivalent stress intensity factor according to the method of Rhee[2].

eqSIFTanaka.m

This function calculates the equivalent stress intensity factor according to the method of Tanaka[3].

eqSIFYan.m

This function calculates the equivalent stress intensity factor according to the method of Yan[4].

fatigueClassicalParis.m

This function calculates the crack growth increment according to the classical Paris model[5].

fatigueModifiedParis.m

This function calculates the crack growth increment according to a modified Paris model[6].

forceVector.m

This function creates the global force vector.

gauss.m

This function contains the values of integration points and weights needed for Gauss quadrature in quadrilaterals and triangles.

growCrack.m

This function determines the angle and magnitude of the next crack growth increment from all crack tips. The direction of future crack growth is determined based on the maximum circumferential stress criterion. If the Paris model constants are assigned in inputQuasiStatic.m then the Paris model with the Tanaka mixed-mode correction are used to determine the increment of crack growth. If the effective stress intensity factor does not exceed the critical stress intensity factor no crack growth occurs and the iterations of growth exit.

growCrackVariable.m

The equivalent of growCrack.m for variable amplitude loading.

heaviNodes.m

This function calculates the values of the enrichment functions at the nodes enriched with the Heaviside enrichment function.

inputLoadHistory.m

Allows for variable amplitude, uniaxial tension loading in the bi-axial stress state. The value of amplitude at each iteration is input into loadHistory vector within inputLoadHistory.m. For the use of variable amplitude loading, FORCE in inputQuasiStatic should be empty.

inputOptimization.m

Follows the same layout as inputQuasiStatic.m however, the optimization problem also requires the modification of the xfemOptimization.m file in order for the crack to be moved at each iteration of the optimization. See benchmark for example. Note that based on the optimization being performed, the use of this file may be drastically different.

inputQuasiStatic.m

Text input file for definition of problem statement. If GUI is preferred then run mxfemGUI.m instead of directly editing this file. The following input variables are used to define the problem of interest: DOMAIN, MAT, CRACK, INC, VOID, GROW, FORCE, BC, PLOT. In order for an analysis to successfully run, the minimum required variables to be defined are DOMAIN, MAT, GROW, FORCE and BC. A listing of the variables, their size and their meaning follows.

DOMAIN
This variable defines the rectangular domain and mesh density for the given analysis.
DOMAIN(1) = number of elements in the x-direction
DOMAIN(2) = number of elements in the y-direction
DOMAIN(3) = element length in x-direction
DOMAIN(4) = element length in y-direction (currently DOMAIN(3) must equal DOMAIN(4))

MAT
This variable defines the material properties for the given analysis.
MAT(1) = Young's modulus for domain
MAT(2) = Poisson's ratio for domain
MAT(3) = Young's modulus for inclusion
MAT(4) = Poisson's ratio for inclusion
MAT(5) = plane stress (1) or strain (2)
MAT(6) = plane stress thickness
MAT(7) = critical stress intensity factor for domain
MAT(8) = threshold Mode I stress intensity factor
MAT(9) = threshold Mode II stress intensity factor
MAT(10) = yield stress for domain

CRACK
This variable defines the points which define the linear segments of the crack.
CRACK(i,1) = x-coordinate of the 1st point of the ith segment
CRACK(i,2) = y-coordinate of the 1st point of the ith segment
CRACK(i+1,1) = x-coordinate of the 2nd point of the ith segment, 1st point of the (i+1)th segment
CRACK(i+1,2) = y-coordinate of the 2nd point of the ith segment, 1st point of the (i+1)th segment

INC
This variable defines the location of the inclusion. Two options are provided, either a linear section cutting the domain into two pieces or circular inclusion(s).
INC(i,1) = x-coordinate of center of inclusion i
INC(i,2) = y-coordinate of center of inclusion i
INC(i,3) = radius of inclusion i

INC(i,1) = x-coordinate of first point defining linear inclusion
INC(i,2) = y-coordinate of first point defining linear inclusion
INC(i,3) = x-coordinate of second point defining linear inclusion
INC(i,4) = y-coordinate of second point defining linear inclusion

VOID
This variable defines the location of the voids. Circular void(s) are available.
VOID(i,1) = x-coordinate of center of void i
VOID(i,2) = y-coordinate of center of void i
VOID(i,3) = radius of void i

GROW
This variable defines how many iterations of crack growth will occur as well as how the growth is modeled.
GROW(1) = Total number of cycles
GROW(2) = Crack growth increment

GROW(1) = Total number of cycles
GROW(2) = Number of cycles per growth increment
GROW(3) = Paris model constant
GROW(4) = Paris model exponent

GROW(1) = Total number of cycles
GROW(2) = Number of cycles per growth increment
GROW(3) = modified Paris model constant
GROW(4) = modified Paris model exponent
GROW(5) = beta
GROW(6) = beta1
GROW(7) = n

FORCE

This variable determines the applied loading to the rectangular domain.
FORCE(1) = uniaxial tension in global x (1) or y (2) direction
FORCE(2) = magnitude of force in the x-direction along applied direction
FORCE(3) = magnitude of force in the y-direction along applied direction

FORCE = [] for variable amplitude loading, triggers reading loads from inputLoadHistory

BC

This variable determines which boundary conditions should be applied to the rectangular domain.
BC(1) = 1, bottom edge roller, bottom left corner fixed
BC(1) = 2, edge crack
BC(1) = 3, half center crack
BC(1) = 4, full center crack
BC(1) = 5, rollers bottom and left edges
BC(1) = 6, symmetric about midpoint

PLOT

This variable controls which plots are output by the MATLAB code. For all cases 1 = Yes, 0 = No.
PLOT(1,1) = plot level set functions
PLOT(1,2) = plot phi level set functions
PLOT(1,3) = plot psi level set function
PLOT(1,4) = plot zeta level set function (inclusions)
PLOT(1,5) = plot chi level set function (voids)
PLOT(1,6) = plot the discontinuity on the level set plot
PLOT(1,7) = define the narrow band level set radius for crack as integer for number of elements
PLOT(2,1) = plot mesh
PLOT(2,2) = plot node numbers
PLOT(2,3) = plot element numbers
PLOT(2,4) = plot enriched nodes (blue circle - Heaviside, blue square - crack tip, black circle - inclusion)
PLOT(2,5) = plot discontinuities (crack, inclusion, voids)
PLOT(2,6) = plot J-domain search radius
PLOT(3,1) = plot the deformed mesh
PLOT(3,2) = plot node numbers
PLOT(3,3) = plot element numbers
PLOT(3,4) = enriched nodes (blue circle - Heaviside, blue square - crack tip, black circle - inclusion)
PLOT(3,5) = plot deformed crack (inner element only)
PLOT(3,6) = deformation scaling factor (if zero, calculated by code)
PLOT(4,1) = plot the stress on an element-by-element basis
PLOT(4,2) = plot Sxx
PLOT(4,3) = plot Sxy
PLOT(4,4) = plot Syy
PLOT(4,5) = plot Svm

PLOT(4,6) = average nodal stress values
PLOT(5,1) = plot the stress contours, default is contour lines
PLOT(5,2) = plot Sxx
PLOT(5,3) = plot Sxy
PLOT(5,4) = plot Syy
PLOT(5,5) = plot Svm
PLOT(5,5) = plot the filled stress contours

Examples of PLOT for Various Outputs
Plot the phi and psi level sets with default narrow band radius:
PLOT(1,:) = [1 1 0 0 0 0 0];

Plot the phi, psi, zeta and chi level sets with default narrow band radius:
PLOT(1,:) = [1 1 1 1 1 0 0];

Plot the phi, psi, zeta and chi level sets with 10 element narrow band radius:
PLOT(1,:) = [1 1 1 1 1 0 10];

Plot the mesh with node numbers:
PLOT(2,:) = [1 1 0 0 0 0 0];

Plot the mesh with enriched nodes and discontinuities:
PLOT(2,:) = [1 0 0 1 1 0 0];

Plot the σ_{xx}, σ_{yy}, and σ_{xy} stress contours:
PLOT(5,:) = [1 1 1 1 0 0 0];

Plot only the Von Mises filled stress contours:
PLOT(5,:) = [1 0 0 0 1 1 0];

JIntegral.m

This function calculates the mixed-mode stress intensity factors for the traditional or bimaterial crack tip enrichment functions. The default J-domain search radius is 4 elements around the crack tip. The stress intensity factors are retuned such that the last tip in CRACK is first and the first tip in CRACK is second.

levelSet.m

This function creates the ϕ, ψ, ζ and χ level set functions used to track the crack tips, crack body, voids and inclusions. In addition this file defines the locations of the enriched degrees of freedom and assign these enriched nodes tracking values in the NODES matrix.

numIterations.m

This function calculates the number of increments based upon the input file.

plotContour.m

This function plots the stress contours and discontinuities. The option for a filled contour is provided.

plotDeformation.m

This function plots the deformed mesh. Options are available for plotting node and element numbers, the enriched nodes are available.

plotLevelSet.m

This function plots the ϕ, ψ, ζ and χ level set functions. The discontinuities may be plotted.

plotMain.m

This function controls which plots are created based on inputQuasiStatic.m

plotMesh.m

This function plots the finite element mesh. Options are available for plotting node and element numbers, the enriched nodes, the discontinuities and the J-domain radius.

plotStress.m

This function plots the stress distribution within each element and discontinuities. The option to average nodal stress values is available.

stiffnessMatrix.m

This function calculates the global stiffness matrix for the system of equations.

subDomain.m

This function subdivides elements containing discontinuities into triangles so that accurate integration can be performed in these elements.

updateStiffness.m

This function performs the reanalysis algorithm on the stiffness matrix.

xfemOptimization.m

This function is optimized by the desired function in order to optimized the user defined function.

xfemQuasiStatic.m

This function controls the calling of the various functions such that the desired analysis functions well. This is the function which is run in the command window such that the analysis runs.

xfemQuasiStaticVariable.m

This function controls the calling of the various functions such that the desired analysis functions well. This is the function which is run in the command window such that the analysis runs for variable amplitude loading.

Chapter 3: Examples

The following examples are given to show that the provided code provides accurate results for a variety of problems which have well known theoretical values. Please refer to the Examples folder in the provided download file which contains the input files for these Example problems.

1. Center Crack in a Finite Plate Under Uniaxial Tension

The theoretical Mode I stress intensity factor K_I for this configuration is given as

$$K_I = F\left(\lambda\right)\sigma\sqrt{\pi a} \tag{3.1}$$

where σ is the applied nominal stress, $2a$ is the crack length and $F\left(\lambda\right)$ is a factor associated with the finite effect of the plate[7] given by

$$F(\lambda) = \sqrt{\sec\left(\frac{\pi\lambda}{2}\right)}\left(1 - 0.025\lambda^2 + 0.06\lambda^4\right) \tag{3.2}$$

and λ is the ratio between the crack length and the width of the plate given as

$$\lambda = \frac{a}{W} \tag{3.3}$$

where $2W$ is the width of the plate.

Here the following values are used, a = 1, W = 3, σ = 1. This corresponds to a theoretical Mode I stress intensity factor of 1.90. Two models are used and for each case the elemental length is 1/20 which corresponds to a mesh of size 60 x 200. When half of the center crack is modeled the calculated stress intensity factors are K_I = 1.86 and K_{II} = 0.003, which are in good agreement with theoretical values. When the full center crack is modeled the calculated stress intensity factors are K_I = 1.88 and K_{II} = -2.02E-4 at the right tip and K_I = 1.88 and K_{II} = -0.61E-4 at the left tip, which are also in good agreement with theoretical values.

2. Edge Crack in a Finite Plate Under Uniaxial Tension

The theoretical Mode I stress intensity factor K_I for this configuration is given as

$$K_I = F\left(\lambda\right)\sigma\sqrt{\pi a} \tag{3.4}$$

where σ is the applied nominal stress, a is the crack length and $F\left(\lambda\right)$ is a factor associated with the finite effect of the plate[8] given by

$$F\left(\lambda\right) = 1.12 - 0.231\lambda + 10.55\lambda^2 - 21.72\lambda^3 + 30.39\lambda^4 \tag{3.5}$$

and λ is the ratio between the crack length and the width of the plate given as

$$\lambda = \frac{a}{W} \tag{3.6}$$

where W is the width of the plate.

Here the following values are used, a = 1, W = 3, σ = 1. This corresponds to a theoretical Mode I stress intensity factor of 3.17. For an elemental length of 1/20 which corresponds to a mesh of size 60 x 120, the calculated stress intensity factors are K_I = 3.17 and K_{II} = 0.0011, which are in good agreement with theoretical values.

3. Circular Inclusion in a Finite Plate

Here an inclusion with E = 70 GPa and v = 0.3 and radius 0.5 is placed in the center of a plate of size 6 x 10 with E = 50 GPa and v = 0.3 which is subjected to a unit tension in the y-direction. The stress values were calculated using ANSYS and the resulting stress plots were compared to the MATLAB XFEM implementation.

4. Circular Void in a Finite Plate

Here a void of radius 0.3 is placed in the center of a plate of size 3 x 3 which is subjected to a unit stress in the y-direction. The stress plot for σ_{yy} is in excellent agreement with the expected 3σ stress concentration at the edges of the hole.

5. Paris Crack Growth Model

Here the XFEM code is used to grow a crack until the Mode I stress intensity factor reaches the critical value. The example used for comparison is one in which a center crack of length 0.02 is on the wall of a pressure vessel with applied pressure 0.06 MPa, radius of 3.25 m and thickness of 0.00248 m. The material properties are those of an aluminum alloy with E = 70 GPa and v = 0.33. The XFEM simulates growth in increments of 100 cycles up to 4500 for crack growth in an infinite plate. At each increment, the XFEM values are compared to those of the theoretical current crack size from Paris model[5], which are given as

$$a_N = \left[NC\left(1-\frac{m}{2}\right)\left(\frac{pr}{t}\sqrt{\frac{\pi}{2}}\right)^m + a_o^{\,1-\frac{m}{2}} \right]^{\frac{1}{1-\frac{m}{2}}} \qquad (3.7)$$

where N is the number of cycles, C is the Paris model constant, m is the Paris model exponent, and a_o is the initial crack length. A comparison of the XFEM and Paris model values are given in Figure 1.

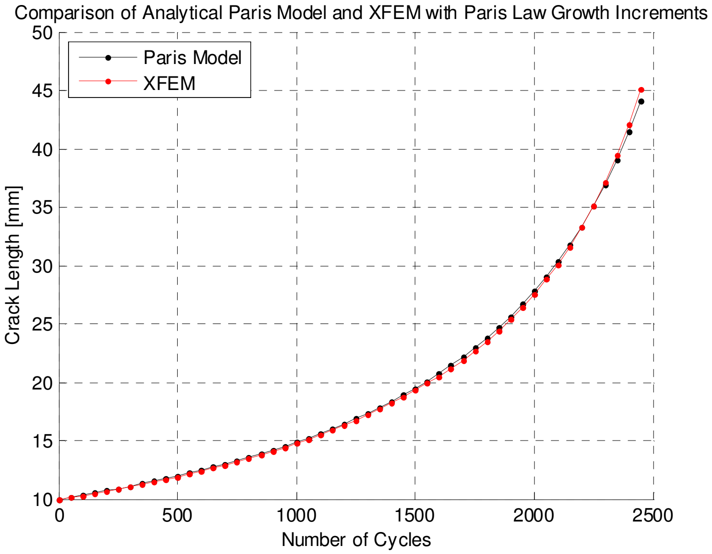

Figure 1. Comparison of XFEM and Paris model predictions of crack growth.

6. Crack Growth in Presence of a Hard/Soft Inclusion

Here an example problem used by Bordas[9] is repeated. In this problem, a domain of width 4 and height 8 contains a circular inclusion with radius 1 and center at width/2 and height/4. An initial edge crack of 0.5 is located along the left edge of the domain at height/2. The inclusion is either harder or softer than the main material with a ratio of 10 or 0.1. The crack is grown until it passes the inclusion. Results correspond very well to published results. Note that the boundary conditions are not specified in the paper and that the resulting difference between the assumed and published solution are most likely a result of different boundary conditions.

7. Crack Initiation Angle for Crack Initiating from a Hole in a Plate

Here an example optimization problem is solved of the form

$$\begin{aligned} &\min && -G(\theta) \\ &\text{s.t.} && -\frac{\pi}{2} \leq \theta \leq \frac{\pi}{2} \end{aligned} \tag{3.8}$$

for a plate with a hole under uniaxial tension. The theoretical solution is 0 degrees, while the optimization result is an initial angle of -1.7287e-005 degrees.

Appendix: Auxiliary Stress Fields

The auxiliary stresses derived by Westergaard and Williams are

$$\sigma_{11} = \frac{1}{\sqrt{2\pi r}}\left\{K_I \cos\frac{\theta}{2}\left[1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right] - K_{II}\sin\frac{\theta}{2}\left[2 + \cos\frac{\theta}{2}\cos\frac{3\theta}{2}\right]\right\} \tag{A1}$$

$$\sigma_{22} = \frac{1}{\sqrt{2\pi r}}\left\{K_I \cos\frac{\theta}{2}\left[1 + \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right] + K_{II}\sin\frac{\theta}{2}\cos\frac{\theta}{2}\cos\frac{3\theta}{2}\right\} \tag{A2}$$

$$\sigma_{33} = v(\sigma_{11} + \sigma_{22}) \tag{A3}$$

$$\sigma_{23} = \frac{1}{\sqrt{2\pi r}} K_{III}\cos\frac{\theta}{2} \tag{A4}$$

$$\sigma_{31} = \frac{-1}{\sqrt{2\pi r}}\sin\frac{\theta}{2} \tag{A5}$$

$$\sigma_{12} = \frac{1}{\sqrt{2\pi r}}\left\{K_I \sin\frac{\theta}{2}\cos\frac{\theta}{2}\cos\frac{3\theta}{2} + K_{II}\cos\frac{\theta}{2}\left[1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right]\right\} \tag{A6}$$

and the auxiliary displacements are

$$u_1 = \frac{1}{2\mu}\sqrt{\frac{r}{2\pi}}\left\{K_I \cos\frac{\theta}{2}(\kappa - \cos\theta) + K_{II}\sin\frac{\theta}{2}(\kappa + 2 + \cos\theta)\right\} \tag{A7}$$

$$u_2 = \frac{1}{2\mu}\sqrt{\frac{r}{2\pi}}\left\{K_I \sin\frac{\theta}{2}(\kappa - \sin\theta) + K_{II}\cos\frac{\theta}{2}(\kappa - 2 + \cos\theta)\right\} \tag{A8}$$

$$u_3 = \frac{2}{\mu}\sqrt{\frac{r}{2\pi}}K_{III}\sin\frac{\theta}{2} \tag{A9}$$

where μ is the shear modulus and κ is the Kosolov constant.

References

[1] Liu Y., Mahadevan S., "Threshold stress intensity factor and crack growth rate prediction under mixed-mode loading." *Engineering Fracture Mechanics*, Vol. 74, 2007, pp. 332-345.

[2] Rhee H., Salama M., "Mixed-mode stress intensity factors solutions for a warped surface flaw by three-dimensional finite element analysis." *Engineering Fracture Mechanics*, Vol. 28, 1987, pp. 203-209.

[3] Tanaka K., "Fatigue crack propagation from a crack inclined to the cyclic tension axis." *Engineering Fracture Mechanics*, Vol. 6, 1974, pp. 493-507.

[4] Yan X., Zhang Z., "Mixed mode criteria for the materials with different yield strengths in tension and compression." *Engineering Fracture Mechanics*, Vol. 42, 1992, pp. 109-116.

[5] Paris P., Gomez M., Anderson W., "A rational analytic theory of fatigue." *The Trend in Engineering*, Vol. 13, 1961, pp. 9-14.

[6] Xiaoping H., Moan T., Weicheng C., "An engineering model of fatigue crack growth under variable amplitude loading." *International Journal of Fatigue*, Vol. 30, 2008, pp. 2-10.

[7] *Stress Intensity Factors Handbook*. Pergamon Press, 1987.

[8] Brown W., Srawley J., "Plain strain crack toughness testing of high-strength metallic materials." *ASME STP 410*, Vol. 1966, pp. 12.

[9] Bordas S., Vinh Nguyen P., Dunany C., Nguyen-Dang H., Guidoum A., "An extended finite element library." *International Journal for Numerical Methods in Engineering*, Vol. 2, 2006, pp. 1-33.