An ABAQUS User Subroutine for a Modified Cohesive Zone Model that Regularizes Convergence and Instability Problems

Y.F. Gao and A.F. Bower
Division of Engineering, Brown University, Providence, RI 02912
Email: Yanfei_Gao@brown.edu, Allan_Bower@brown.edu

METHODOLOGY

For the information of the cohesive zone model (CZM) and our modified version, please refer to "*A simple technique for avoiding convergence problems in finite element simulations of crack nucleation and growth on cohesive interfaces*," by Y.F. Gao and A.F. Bower, Modelling Simul. Mater. Sci. Eng. 12, 453-463, 2004.

ABAQUS allows the addition of a variety of user subroutines. We define a user element by UEL. Information of element and nodal displacements can be obtained from ABAQUS, and we update the stiffness matrix and the residual vector following the cohesive zone law. More coding details can be found in ABAQUS manual.

FILES

- CZM_for_instability.for — User element subroutine written in Fortran 90
- CZM_for_instability_test.inp — An example input file (Fig. 1 of our paper)
- CZM_for_instability_help.pdf — This help file
- MSMSE_YFG_AFB.pdf — Our paper that discusses the methodology

USAGE

- You must have ABAQUS and Fortran complier installed on your computer.
- Type: abaqus job=CZM_for_instability_test user=CZM_for_instability

REMARK

The example input file (CZM_for_instability_test.inp) corresponds to Fig. 1 in our paper. In the input file, you can specify the parameters that are used in the original Xu-Needleman model. Our new cohesive zone needs one additional parameter, i.e. fictitious viscosity, which is written in the SEPLAW function in the Fortran code. The result of this example should give you the load-displacement curve as shown in Fig. 4(a).

Practically, in the beginning, you'd better turn off the fictitious viscosity for your own special problem. If you encounter numerical troubles, then use our scheme with fictitious viscosity. Our paper discusses how to choose it for many practical cases.

December 9, 2004