Technical University of Denmark

Implementing Advanced Materials Models in a Commercial Finite Element Code

Mikkelsen, Lars Pilgaard; Mishnaevsky, Leon

Publication date:
2013

Implementing Advanced Materials Models in a Commercial Finite Element Code

Lars P. Mikkelsen, Leon Mishnaevsky Jr

Composite and Materials Mechanics, Department of Wind Energy,
Risø Campus, Technical University of Denmark.

1 Introduction

Obviously, commercial finite element codes include only a finite number of material models. Material models which not always sufficiently mimic a given material behavior under some given loading condition. Fortunately, some commercial codes includes user interfaces making it possible to implement user specified material laws. In the commercial finite element code, Abaqus, such a user defined interface is available implementing user subroutines at different complexity levels. A complexity increasing going from the simpler user-defined fields subroutine (UField) where some material parameters in an already available material law can be made dependent on user-defined field variables, over the user material subroutine (UMAT) where a given constitutive relation between the increments of the strain and stress components can be defined, to the much more complex case implementing material relation which require new fundamental unknowns in the finite element procedure through a user element user subroutine (UEl). In the presented work, examples of each illustrate some of the possibilities.

2 User-defined materials

Three types of material laws with increasing complexity is used to illustrate the possibility implementing non-standard material laws in the commercial finite element code Abaqus. The first two models relates to a fiber reinforced composite material implementing non-linear material effects on the material response at the constituent level, while the third model incorporate a length scale effect using a complex strain gradient dependent material model. A material model implementing the scale effect of the underlying microstructure without actually modeling it.

2.1 Damage material model

In order to simulate the damage evolution in fibers, matrix or the interface of composites, a User Defined Field user subroutine is implemented based on a finite element weakening method [1]. The idea of this approach is that the stiffness of finite elements is reduced if a stress or a damage parameter in the element or a nodal point exceeds some critical level. In this subroutine, the phase to which a given finite element in the model is assigned is defined through the field variable of the element. Depending on the field variable, different failure conditions are assigned by the subroutine to each finite element of the model. The subroutine checks whether the element failed or not, according to the properties of the matrix, interphase and fibers. Another field variable characterizes the state of the element (“intact” versus “damaged”). If the value of the damage parameter or the principal stress in the element exceeds the corresponding critical level, the second field variable of the element is changed, and the Young modulus of this element is set to a very low value (50 Pa, i.e., about 0.00001% of the initial value). The numbers of failed elements are printed out in a file, which can be used to visualize the calculated damage distribution. Both Weibull distribution of the strengths of each finite elements in fibers and of whole fibers, as well as constant fiber strength, uniform and Gaussian distributions are included into the subroutine, and can be tested in the simulations.

2.2 Smeared-out non-linear composite law

Both in tension and compression, the stiff fibers in unidirectional polymer matrix composites carry the load. Nevertheless, while the composite strength in tension is given by the fiber strength, the composite strength in compression is given by the fiber supporting effects of the matrix material. Realistic predictions of compressive failure in composites require a non-linear matrix response. In [2], this was implemented using a User-defined Material user subroutine implementing independent non-linear material response of the two constituents in a polymer matrix composite. The two non-linear material responses, for simplicity assumed to follow a J2-flow theory, was implemented in a UMat routine taken into account finite straining and rotation of the material points. During this, it is possible to model the kinking phenomena often observed as the compressive failure mechanism in thick laminates. The user defined material law is working on the Gaussian integration point level and can therefore in principle be used together with all the available elements types in the finite element code. Nevertheless, in the present form the model is restricted to the plane strain case.

2.3 Strain gradient dependent plasticity model

A strain gradient dependent plasticity model is an enhanced plasticity model making it possible to account for scale effect during plastic yielding. In the model implemented [3], an advanced material law is used where not only the displacements but also the effective plastic strain is treated as fundamental unknowns. Thereby, it is possible to model the effects of the underlying microstructure of a metallic material without actually modeling it. Instead, the effect is taken into account through a incorporated length scale. Do to the structure of the material law; a completely new element definition must be taken into account. This is implemented using the User Element user subroutine interface. The element implemented is a combined four and eight node isoparametric element where the displacements is approximated by 8 node shape functions while the effective plastic strain is implemented using 4 node shape functions. The user defined element makes it possible for the user to control nearly everything regarding calculating the element stiffness matrix including updating the state variables. The implementation follows the procedure prescribed in [4] and can e.g. predict non-uniformed deformations introduced by higher order boundary condition restricting the plastic yielding of the material along some constrained layer. Implementing a user defined element is nearly as general as building up your own finite element code except controlling the numerical solution procedure.

3 Conclusions

Non-standard material laws will in many cases not require building up a new finite element code. Instead you can take advances of the user-defined subroutines build into some commercial finite element codes. Thereby, it is possible to take advance of many of the features already available in the codes including the graphically user interface for the pre- and post-processing process. In addition, it is only necessary to select a complexity level complying with your material law ranging from the simpler user-defined field definition to the very general but also more complex user-defined element definition.

4 References

[1] Mishnaevsky L, Brøndsted P. Micromechanisms of damage in unidirectional fiber reinforced composites: 3D computational analysis. Compos Sci Technol 2009;69:1036–44.

[2] Sørensen KD, Mikkelsen LP, Jensen HM. User subroutine for compressive failure of composites. 2009 Simulia Cust. Conf., London, UK: SIMULIA; 2009, p. 618–32.

[3] Mikkelsen LP. Implementing a gradient dependent plasticity model in ABAQUS, Paris, France: SIMULIA; 2007, p. 482–92.

[4] Niordson CF, Hutchinson JW. Non-uniform plastic deformation of micron scale objects. Int J Numer Methods Eng 2003;56:961–75.