Implementation of metal ductile damage criteria in Abaqus FEA

Fukun Li1*, Hao Yuan2, Huijuan Liu3
Key Laboratory of Disaster Prevention and Structural Safety of China Ministry of Education; Guangxi Key Laboratory of Disaster Prevention and Engineering Safety; College of Civil Engineering and Architecture, Guangxi University, Nanning, 530004, China.
*Corresponding author: E-mail: 1151981242@qq.com; 20110057@gxu.edu.cn

Abstract. Ductile damages often happen on metal structures under large deformations. In the current paper, the ductile damage criteria are discussed from the microscopic mechanism to metal ductile fracture models. Abaqus FEA makes the metal ductile damage criteria easy to be applied for practical problems. Therefore, the paper finally presents the detailed implementation of the ductile damage criteria in Abaqus FEA to describe the metal damage and degradation, simulating the ductile failure process with higher fidelity.

1. Introduction
Since the 1950s, the finite element method (FEM), derived from using structural mechanics to solve elastic mechanics issues, has been widely used in engineering technology. FEM is used in numerical analysis and solving engineering issues, and is often used for simulation analysis of scientific research experiment process. Abaqus FEA, an internationally-known general finite element analysis software, is featured with powerful calculation capabilities and simulation accuracy.

Abaqus FEA can stimulate the performance of typical engineering materials, thanks to its unit library covering a great quantity of geometrical shapes, performances and accuracy, and its material library covering metals, rubbers, polymer materials, composite materials, and reinforced concretes. Moreover, it is also recognized as the most powerful finite element analysis software to solve large and complex problems and simulate highly nonlinear problems. It can also use different element models, material models, and analysis processes according to different simulation scenarios [1-2].

2. Metal ductile damage mechanism
Ductile fracture is a form of fracture that often occurs when steel structural members undergo large plastic deformation. Starting from the microstructure, steel is composed of metal particles and impurities such as carbides and sulfides. Under load, steel generally has two types of fracture, namely, cleavage fracture and microvoid growth aggregate fracture. Cleavage fracture refers to that under the action of tensile force, the boundary between the metal particles and impurities in the steel microstructure is broken, and then it continues to extend until a macroscopic crack occurs. This kind of damage is often accompanied by small deformation, reflecting the brittleness of the material, so it is considered to be the mechanism of brittle fracture. In the process of microvoid growth and aggregation fracture, metal particles and impurity particles disengage from each other under the action of flow plastic deformation caused by the slip of the metal lattice to form voids. The metal flow plastic deformation between these
voids causes the voids to expand and aggregate continuously; eventually, cracks form. This type of fracture often presents large strain characteristics, so it is generally considered the mechanism of ductile fracture.

In recent decades, the prediction of ductile fracture of metal materials has been a hot spot of attention and research. Studies have shown that the growth of voids is mainly affected by two factors, equivalent plastic strain and stress triaxiality [3]. When the steel structure is under tension, the inclusions in the metal peel off from the metal matrix and form voids, causing metal failure. The internal damage of the material gradually evolves and accumulates before the material cracks. When the damage state variables are accumulated to a critical value, the material undergoes ductile fracture. In the physical world, ductile fracture under high-stress triaxiality can be attributed to microvoid growth and subsequent ligament necking between voids. In contrast, under low-stress triaxiality, the ductile fracture is attributed to microvoid extension and inter-ligament shearing [4].

Existing metal ductile fracture constitutive models often have high accuracy in engineering applications and are widely used in large-scale structural simulations. Rice and Tracey theoretically deduced the exponential correlation formula between void growth rate and triaxiality [5]. Trattnig et al. conducted a series of different triaxial tests on Austenitic steel and derived the same index correlation formula based on the experimental results [6].

2.1. Microscopic mechanism of ductile fracture
The ductile fracture mechanism of metals mainly includes three stages: void nucleation, growth, and coalescence.

2.1.1. Void nucleation.
Inclusions are formed around the second phase particles or at the interface between the particles and the matrix. For a given material, void nucleation is considered to occur under critical stress or critical strain. Under high-stress triaxiality, crack initiates from void nucleation and generally occurs under the normal stress state. Once the voids nucleate, they continue to expand, aggregate, and produce rough fracture surfaces.

2.1.2. Void nucleation.
In the early studies of ductile fracture, void expansion was the primary way of void growth [7]. The void expansion driven by the stress triaxiality characterizes the volume growth of the void. In various void expansion models, Rice and Tracey, based on the analysis of void growth, the exponential form of the ductile fracture criteria is derived, and its calculation formula is as follows:

\[ \bar{\varepsilon}_f = a_1 \exp(-a_2 \eta) \]  

(1)

Where, \( a_1 \) and \( a_2 \) are two positive material parameters to be calibrated, while \( \bar{\varepsilon}_f \) is the equivalent plastic strain at the onset of fracture. The original criteria put the constant \( a_2 \) at 1.5, which was found to apply to many engineering materials. This equation is abbreviated as the R-T model, also known as the void growth model (VGM), and its physical basis is that each void fracture when it grows to a critical size, and the plastic strain at this time represents the fracture strain. The initial focus of their derivation was the fracture at high triaxiality, that is, \( \gamma > 2 \). The following applications of the simple exponential relationship between triaxiality and fracture strain subsequently verified fracture with triaxiality between 1 and 2. This function describes the vital effect of stress triaxiality on the growth rate of voids and damage evolution. This triaxial correlation model has a good predictive effect on high triaxial ductile fractures.

3. Implementation of metal ductile damage model in Abaqus FEA
In Abaqus FEA, the failure mechanism specification consists of four different parts: 1) the definition of effective (or undamaged) material response (as shown in the curve a-b-c-d’ in Figure 1); 2) the failure initiation criteria (Point c in Figure 1); 3) the law of damage evolution (curve c-d section in Figure 1);
4) an option for element deletion, once the material stiffness is completely degraded, elements can be deleted from the calculation (Point d in Figure 1).

Figure 1. Uniaxial tensile stress-strain curve of standard material properties

Abaqus FEA provides various damage initiation standard options for ductile metals, each with different type of material failure related to the damage initiation criteria of metal fracture, including ductile criteria and shearing criteria.

Each damage initiation criteria have a related output variable to indicate whether the criteria are met during the analysis process. A value of 1.0 or higher indicates that the initialization criterion is met. For a given material, more than one damage initiation criteria can be specified. If multiple damage initiation criteria are specified for the same material, Abaqus FEA treats them independently. Once the specific initial criterion is met, the material’s stiffness degrades according to the specific damage evolution rule of the criteria. Without the damage evolution rule, the material hardness will not decrease. Only the failure mechanism that specifies the damage evolution response is the active state. Abaqus FEA only evaluates the starting criteria of the inactive mechanism for output purposes, but the mechanism does not affect the substance response.

The law of damage evolution describes the rate of decrease in the material’s stiffness to reach the corresponding initial standard. For the damage of ductile metals, Abaqus FEA hypothesis can use scalar damage variables $d_i (i \in N_{\text{act}})$ in modeling the stiffness reduction associated with each active failure mechanism, where $N_{\text{act}}$ refer to the active mechanism. At any time in the analysis process, the stress tensor of the material is given by the scalar damage equation.

$$\sigma = (1 - D) \bar{\sigma} \quad (2)$$

Therein, $D$ is the overall damage variable, $\bar{\sigma}$ is the effective (or undamaged) stress tensor calculated in the current increment. $\bar{\sigma}$ is the stress that exists in the material without damage. When $D=1$, the material loses its bearing capacity. By default, if all cross-section points at any integration location lose their load-bearing capacity, an element is deleted from the grid. According to user-specified criteria, the total damage variable captures the combined effects of all active mechanisms and is calculated based on a single damage variable $d_i$. Abaqus FEA supports different ductile metal damage evolution models and provides control related to element deletion caused by material failure. All available models use the same formula to reduce the strong grid dependence of strain localization effects on the results during progressive damage.

3.1. Ductile damage initiation

The ductile criterion is a phenomenon model used to predict the damage caused by nucleation, growth, and aggregation of voids. The model assumes that the equivalent plastic strain at the beginning of the damage is $\varepsilon_{pl}^0$, being a function of stress triaxiality and strain rate:
Where \( \eta = -p / q \) is the stress triaxiality, \( p \) is the pressure stress, \( q \) is the Mises equivalent stress, \( \dot{\varepsilon}^{\text{pl}} \) are the equivalent plastic strain rates. The criteria for causing damage can be met when the following conditions are met:

\[
\int \frac{d\varepsilon^{\text{pl}}}{\bar{\varepsilon}_D^{\text{pl}}(\eta, \dot{\varepsilon}^{\text{pl}})} = 1
\]

(4)

Where \( \omega_D \) is a state variable that monotonously increases with plastic deformation. In the analysis process, each increment \( \omega_D \) is calculated as:

\[
\Delta \omega_D = \frac{\Delta \varepsilon^{\text{pl}}}{\bar{\varepsilon}_D^{\text{pl}}(\eta, \dot{\varepsilon}^{\text{pl}})} \geq 0
\]

(5)

The exponential dependence of the equivalent fracture strain on the stress triaxiality is obtained through the material property test, as shown in the following formula:

\[
\varepsilon_f^{\text{pl}} = \alpha \cdot \exp(-\beta \cdot D)
\]

(6)

And enter the parameters of Abaqus FEA ductile damage as the initial criteria of damage.

3.2. Ductile damage evolution

The damage evolution ability of ductile metals hypothesizes that the characteristic of damage is the gradual degradation of material hardness, leading to material failure. Abaqus FEA has setting options that specify how each damage mechanism promotes the overall degradation of the material.

3.2.1. Evolution principles. Figure 2 illustrates the characteristic stress-strain behavior of damaged materials.

The overall damage variable \( D \) captures the combined effects of all active agencies and is calculated as the single damage variable \( d_t \) multiplying the intermediate variables \( d_{\text{mult}} \), as follows:

\[
d_{\text{mult}} = 1 - \prod_{k \in N_{\text{act}}} (1 - d_k)
\]

(7)

Then, the total damage variable is taken as the largest one among \( d_{\text{mult}} \) and other remaining variables:
\[ D = \max \left\{ d_{\text{mult}}, \max_{j \in N_{\text{max}}} d_j \right\} \] \tag{8}

In the above expression, \( N_{\text{mult}} \) and \( N_{\text{max}} \) express the set of active mechanisms that contribute to the overall damage in the form of multiplication and maximum value, where \( N_{\text{act}} = N_{\text{mult}} \cup N_{\text{max}} \).

3.2.2. Evolution parameters.

Once the initial failure criterion is reached, the effective plastic failure displacement \( \vec{u}_{\text{pl}} \) [8], according to the evolution equation, is defined as

\[ \vec{u}_{\text{pl}} = L \hat{\vec{e}}_{\text{pl}} \] \tag{9}

Where \( L \) is the characteristic length of the element.

The equivalent plastic displacement is determined by considering the grid dependence and characteristic length. The calculation formula is as follows:

\[ \vec{u}_{\text{pl}} = \lambda_s L_{\text{char}} \left( \vec{e}_{\text{pl}} - \vec{e}_{\text{u}} \right) \] \tag{10}

\[ \lambda_s = \frac{L_{\text{E}}}{L_{\text{E}}} \] \tag{11}

The coefficients are:

Characteristic element length \( L_{\text{char}} = \lambda_s L_{\text{E}} \) is defined as component size \( L_{\text{E}} \) multiplying by the component type factor \( \lambda_{\text{E}} \).

There are two kinds of damage variables. One is Linear form

\[ d = \frac{L \hat{\vec{e}}_{\text{pl}}}{\vec{u}_{\text{pl}}} = \frac{\vec{u}_{\text{pl}}}{\vec{u}_{\text{pl}}} \] \tag{12}

This definition can ensure that when the elastic-plastic displacement value is effectively reached \( \vec{u}_{\text{pl}} = \vec{u}_{\text{pl}} \), the material stiffness completely degrades \( (d=1) \).

Another one is exponential form. The corresponding damage variable calculation formula is:

\[ d = 1 - e^{-\alpha (\sigma^m/\sigma_p)} \] \tag{13}

3.2.3. Element deletion. By specifying an upper limit in Abaqus FEA \( D_{\text{max}} \) as the total damage variable \( D \), select the “deleting element” command to deal with severely damaged elements when the maximum degradation is reached. By default, elements are deleted from the grid when the material reaches its maximum degradation. In the definition of the maximum damage value \( D_{\text{max}} \), the default setting depends on whether the element is deleted when the maximum degradation is reached. For the default case of element deletion, \( D_{\text{max}} = 1.0 \); otherwise \( D_{\text{max}} = 0.99 \). The output variable SDEG contains the value of \( D \). When \( D \) reaches \( D_{\text{max}} \), no more damage accumulation will be performed at the integration point. Except for cohesive elements with traction separation response, Abaqus FEA damages all rigid elements for elements that may eventually be removed:

\[ \sigma = (1 - D) \bar{\sigma} \] \tag{14}
In Abaqus/Explicit, if $D$ of an element at all cross-section points at any integral position reaches $D_{\text{max}}$, then the element is deleted from the grid.

4. Conclusion
This paper introduces the large-scale finite element analysis software Abaqus FEA, including its applicable problems, calculation principles, and describes the microscopic mechanism of the metal ductile damage criteria. Through the finite element analysis, the material properties of the material can be set in Abaqus FEA, which can simulate the structure’s damage during the stress process and express the fracture failure of the material by applying the form of element deletion to the model. In summary, in Abaqus FEA, the application of ductile damage models has been more advanced. Ductile damage models built-in are comprehensive to facilitate multi-parameter theoretical models to predict the damage and failure of metal components, which are of very constructive significance to the analysis of ductile materials.

Acknowledgements
This work was supported by the National Natural Science Foundation of China [grant number 51708135, 52068003]; the Guangxi Natural Science Foundation Program [grant number 2019GXNSFAA185057]; and the Systematic Project of Guangxi Key Laboratory of Disaster Prevention and Engineering Safety [grant number 2019ZDK043, 2019ZDK044]. Opinions, findings, and conclusions expressed in this paper are those of the authors and not necessarily those of the sponsors.

References
[1] Liu Z, Qian Y. CAE analysis series: ABAQUS finite element analysis from entry to proficiency[M]. Beijing:People’s Post Publishing House, 2015. 6.
[2] Jiang B, Kong X, Luo Y. CAE analysis series: Detailed explanation of Abaqus FEA engineering examples[M]. Beijing: People’s Post Publishing House, 2014. 9.
[3] Zhu Y, MD Engelhardt. Prediction of Ductile Fracture for Metal Alloys Using a Shear Modified Void Growth Model [J]. Engineering Fracture Mechanics, 2018: 491-513.
[4] Barsoum I, Faleskog J. Rupture mechanisms in combined tension and shear-Experiments [J]. International Journal of Solids and Structures, 2007.
[5] Rice J R, Tracey D M. On the ductile enlargement of voids in triaxial stress fields[J]. Journal of the Mechanics & Physics of Solids, 1969, 17(3): 201-217.
[6] Trattnig G, Antretter T, Pippan R. Fracture of austenitic steel subject to a wide range of stress triaxiality ratios and crack deformation modes[J]. Engineering Fracture Mechanics, 2008, 75(2): 223-235.
[7] McClintock, F. A. A Criterion for Ductile Fracture by the Growth of Holes [J]. Journal of Applied Mechanics, 1968, 35(2): 363-371.
[8] Pavlovic M, Markovic Z, Veljkovic M, et al. Bolted shear connectors vs. headed studs’ behaviour in push-out tests [J]. Journal of Constructional Steel Research, 2013; 88(SEP.):134-149.