Widely Employed Constitutive Material Models in Abaqus FEA Software Suite for Simulations of Structures and Their Materials: A Brief Review

S. M. Anas*, Rafat Tahzeeb, Rayeh Nasr Al-Dal'aieen, Mehtab Alam, and Mohd Shariq

1 Department of Civil Engineering, Jamia Millia Islamia (A Central University), 110025 New Delhi, India
2 College of Graduate Studies, Universiti Tenaga Nasional, Jalan Ikram-UNITEN, 43000 Kajang, Selangor, Malaysia
3 Civil Engineering Department, College of Engineering, Al-Balqa Applied University (BAU), 19117 Salt, Jordan
4 Department of Civil Engineering, Netaji Subhas University of Technology, 110073 New Delhi, India

Abstract

The structural response of masonry/concrete structures depends upon the load-carrying mechanism and subsequently deformations produced by loads carried. In masonry/concrete structures, identification of the stress/strain including their non-linear relationship under imposing stress conditions and strain hardening/softening makes the structural response more complicated. Elastic damage models or elastic-plastic constitutive laws are inadequate to simulate masonry/concrete response under high strain-rate loadings. Further, irreversible or plastic strain cannot be realized using the elastic damage model. Several constitutive damage models are available in the literature. In this article, a concise explanation of the functioning of different material models in the Abaqus software package has been provided. These models include concrete damage plasticity for concrete and masonry, traction separation constitutive laws for brick-mortar interface, Hashin's criteria for CFRP, Johnson-Cook plasticity for steel, and crushable foam plasticity hardening for metallic foams. Researchers frequently utilize these models for numerical simulations and modeling of infrastructural elements and their respective materials when subjected to various structural loads. Besides, this paper presents a discourse on problem-solving methods and a comparison between explicit and implicit analysis. The research provides valuable input to researchers and practitioners in the field of structural engineering for an in-depth understanding of the functioning of Abaqus' pre-existing material models.

1. Concrete Damage Plasticity (CDP)

CDP often called as Barcelona Model, is a tweaked version of the Drucker-Prager (D-P) strength hypothesis\[2\]. It was first put up by\[3\] for single unvarying load conditions associated with a failure of brittle materials like concrete, masonry, and ceramics under low confining-pressure levels. For cyclic and dynamic loading conditions, CDP was upgraded by\[4\] upon incorporating damage factors used by various scholars (e.g.,\[5-7\]). The Barcelona model implies that the degradation of elastic strength/modulus of the above materials is isotropic and managed by a damage parameter, \(d\) associated with two different evolution models or modes of failure: tensile cracking \((d_t)\) and compressive crushing \((d_c)\)\[1\]. The stress-strain connection in CDP is described by the following equation.

\[
\sigma = (1 - d)D_0 \varepsilon^e_t, (\varepsilon - \varepsilon^p) = D^e_t, (\varepsilon - \varepsilon^p) \quad 0 < d < 1
\]

*Corresponding author: s1910521@st.jmi.ac.in
response of the material gets weakened because of the development of cracks in the material’s elastic stiffness resulting in

\[
F = \frac{1}{1-\alpha} [\tilde{q} - 3\alpha \tilde{p} + \beta (\varepsilon_{\text{pl}} e_{\text{pl}}) (\tilde{\sigma}_{\text{max}}) - \gamma (-\tilde{\sigma}_{\text{max}})] - \tilde{\sigma}_c (\varepsilon_{\text{pl}}) = 0
\]
\[
\alpha = \left( \frac{\sigma_{\text{bb}}}{\sigma_{\text{co}}} \right)^{-1} 0 \leq \sigma \leq 0.50
\]

\[
\beta(e^{-p^2}) = \frac{\bar{s}_c(e^{-p^2})}{\bar{s}_c(e^{-p^2})}(1 - \alpha) - (1 + \alpha)
\]

\[
\gamma = \frac{3(1-K_c)}{2K_c - 1}
\]

\[
\bar{p} = -\frac{1}{3} : 1
\]

\[
\bar{q} = \frac{1}{\sqrt{2}} S : S
\]

\[
\bar{S} = \bar{p} + \bar{q}
\]

\[
G = \sqrt{(\epsilon \sigma_{\text{to}} \tan \Psi)^2 + \bar{q}^2 - \bar{p} \tan \Psi}
\]

(a) Yield surface in 2D plane stress

(b) Deviatoric plane

Fig. 3. CDP’s yield surface criteria
\[ \epsilon \to \text{results in additional curvature to the potential flow} \] 

A tighter curvature of the potential flow crossing the \( p \)-axis occurs for \( \epsilon \) less than 0.10, causing convergence problems.

**Fig. 4.** Flow rule in CDP [1].

During FEM simulations, constitutive models with material strength degradation and softening mechanism usually produce convergence challenges. CDP made use of the visco-inelastic strain rate tensor \( \dot{\epsilon}^{\text{pl}}_{\text{pl}} \) established [15], given in Eq. (10).

\[
\dot{\epsilon}^{\text{pl}}_{\text{pl}} = \frac{1}{\mu} (\epsilon^{\text{pl}} - \dot{\epsilon}^{\text{pl}}) \]  

A visco-inelastic system's temporal retardation is represented by the viscosity parameter, \( \mu \). A lower value (\( \approx 0.001 \)) of this parameter might help to enhance model convergence in the softening without affecting the outcomes [16-17].

**Salient features of CDP:**

- Abaqus’ MAT_CDP considers the linear elastic response of concrete/masonry before the cracking; “MAT” refers to the material model. After the cracking, the concrete/masonry shows a plastic response with concrete/masonry yield that is described by a 3D yield or failure surface. This model follows various inbuilt constitutive laws and has the feature of eradicating the damaged element (finite element) when the peak principal strain in the element surpasses the predefined value by the user resulting in a total loss of element strength/stiffness [1].

- MAT_CDP is an excellent tool for simulating crack propagation in concrete or masonry materials [16]. Simulation of the damage also helps to prevent the concrete/masonry from acting as an ideal plastic material.

- MAT_CDP is capable of predicting damage and other related responses under high strain rate loadings, therefore taking the edge of increased strength or dynamic strength effects under explosion loading in this study. A material model consisting of multiple CDP descriptions, at different strain rates, can be linked either by an optional parameter DEPENDENCIES (depends on the strain rate) in Abaqus or by writing FORTRAN material routine and joining it with Abaqus [1]. This creates something like a discrete description of the changing characteristics of the material/s (similar to interpolation).

- MAT_CDP can quantify the total damage to the numerical model and presents it in terms of damage dissipation energy: a mechanical strain-based parameter, defined as the amount of energy dissipated by damage due to the applied loading [1]. In this work, this parameter is abbreviated as DDE. The CDP damage parameter, \( d \), as discussed above can also be related to DDE and total absorbed energy (\( Q \)) as follows:

\[
Q - D_{DE} = (1 - d)Q \\
\]  

\[
d = \frac{D_{DE}}{Q} \]
2. Drucker-Prager (D-P)

\[ \alpha A_1 + \sqrt{B_2} - l = 0 \]

\[ A_1 = \sigma_1 + \sigma_2 + \sigma_3, \quad B_2 = t_{ij}B_{2} = \frac{t_{ij}t_{ij}}{2} \]

\[ \alpha = \frac{\sigma_c - 1}{\sqrt{3} \sigma_c + 1} \]

\[ l = \frac{2\sigma_c}{\sqrt{3}(\sigma_c + 1)} \]

\( \sigma_t, \sigma_c \)

3. Nonlinear Model for Brick-Mortar (B-M) Interface

\[ \sigma = K\delta \]
The elastic stiffness matrix, $K$, can be expressed as:

$$
\begin{align*}
\{\sigma_N\} &= \begin{bmatrix} K_{NN} & 0 & 0 \\ 0 & K_{SS} & 0 \\ 0 & 0 & K_{TT} \end{bmatrix} \{\delta_N\} \\
K_{NN} &= \frac{E_b}{\ell_m(\ell_b-\ell_m)} \\
K_{SS} &= \frac{G_b}{\ell_m(\ell_b-\ell_m)}
\end{align*}
$$

Elastic modulus, $E$, and fracture energy, $G$, are material properties; subscripts $b$ and $m$ refer to brick and mortar units; $\sigma_N$, $\sigma_S$, and $\sigma_T$ are normal and shear stresses, respectively. The traction-separation law for defining cohesive contact between the brick units is shown in Figure 6.

Fig. 5. Traction-separation rule for brick-mortar connection.

Fig. 6. Traction-separation response for defining contact between masonry units.


\[
\left( \frac{\sigma_N}{\sigma_{N,\text{max}}} \right)^2 + \left( \frac{\sigma_T}{\sigma_{T,\text{max}}} \right)^2 + \left( \frac{\tau}{\tau_{\text{crit}}} \right)^2 = 1
\]

\[
\tau_{\text{critical}} = c + \mu \sigma
\]

4. Hashin Damage (H-D) Criteria

- C-FRP laminate modeling

\[
\sigma = C: \varepsilon
\]

\[
C = \begin{bmatrix}
C_{11} & C_{12} & C_{13} \\
C_{21} & C_{22} & C_{23} \\
C_{31} & C_{32} & C_{33} \\
C_{44} & & \\
& C_{55} & \\
& & C_{66}
\end{bmatrix}
\]

\[
C_d = \begin{bmatrix}
b_{11}C_{11} & b_{12}b_{22}C_{12} & b_{11}b_{33}C_{13} & 0 & 0 & 0 \\
b_{21}b_{22}C_{22} & b_{11}b_{33}C_{23} & 0 & 0 & 0 \\
& & b_{11}b_{33}C_{33} & 0 & 0 & 0 \\
b_{12}b_{44} & & & & & 0 \\
& & b_{44}C_{44} & 0 & 0 & 0 \\
& & & & b_{45}C_{55} & 0 \\
& & & & & b_{46}C_{66}
\end{bmatrix}
\]
\[ \sigma = C_d \varepsilon \]

\[ \sigma = \left( A + B \varepsilon_{eq}^n \right) \cdot \left[ 1 + C \ln \left( \frac{\varepsilon}{\varepsilon_0} \right) \right] \cdot \left[ 1 - \left( \frac{T - T_0}{T_{melt} - T_0} \right)^m \right] \]

\[ \sigma_{eq} \]

\[ A, B, C, n, m \]

\[ T_0, T_{melt}, T \]

\[ \varepsilon, \varepsilon_0, \varepsilon_{eq}^p, \varepsilon_{eq}^h \]

**Modeling of interface between C-FRP and exposed surface**

5. **Johnson-Cook Plasticity (JCP)**

6. **Crushable Foam Plasticity Hardening (CFPH)**
damping and energy absorption capacity; where “pores” refer to “gas-filled pores” made during the manufacturing process. The former has a very light composition while the latter has a very dense composition. Open-cell foams are generally employed for fluid-based applications while closed-cell ones are used for load-bearing applications. Manufacturing companies utilize metallic foams in various military vehicles to protect them from landmine blasts. 

CFPH available in Abaqus for modeling crushable or crashworthy foams is regulated by the Von Mises shear stress ($\sigma$) and hydrostatic pressure ($p$). The yield surface or failure envelope ($F$) which is governed by an equivalent inelastic strain is an ellipse in the stress plane $p$-$q$, and circle in the deviatoric plane (Figure 7), and is defined as:

$$F = \sqrt{q^2 + \alpha^2 p^2} - B = 0$$

$$B = \alpha p_c = \sigma_c \sqrt{1 + \left(\frac{\alpha}{\beta}\right)^2}$$

$$p = \frac{1}{3} \text{trace } \sigma$$

$$q = \sqrt{\frac{2}{3} S : S - \frac{1}{2} B^2}$$

$$\alpha = \frac{3k}{\sqrt{3-k^2}}$$

$$\beta = \frac{3}{\sqrt{2}} \sqrt{\frac{1-2v_p}{1+v_p}}$$

$1.0 \leq v_p \leq 0.50.$

Additional details related to CFPH can be found in [1].
7. Solving Technique and Strain-rate Effects

Abaqus, a multi-physics commercial program for the finite element analysis and computer-aided engineering, developed by Dassault Systèmes Simulia Corp. (Germany) in the year 1978, consists of two core products or modules—Implicit (often called Abaqus/Standard) and Explicit (Abaqus/Explicit).

The explicit module is based on central-difference time integration algorithm/rule requires very high computer resolution and a small time-step (usually of order microsecond to the millisecond) to simulate the process of high strain-rate loadings such as blast and the corresponding dynamic responses and damages \[1, 52\]. In this module, formulations of dynamic equations express the node displacement at a particular time, \(t_{j+1}\) (\(j = \text{time increment}\)) in terms of displacements, velocities, and spatial accelerations at the preceding time step (or iteration). This method readily accommodates both material and geometric nonlinearities and captures the process of blast loading; nevertheless, all explicit formulations have a threshold time step over which the module becomes unstable \[52\]. The threshold time is influenced by the material’s wave velocity, which is predefined by Abaqus, and the smallest dimension of the element mesh \[1\].

On the other hand, formulations of dynamic equations in an implicit module define node displacement at a time \(t_{j+1}\) in terms of all the displacements, velocities, and spatial accelerations at that time only. Further, the response of linear-elastic systems is calculated by simultaneously solving dynamic equations for the entire system; this is often done using matrix techniques and is a very effective approach. However, for nonlinear dynamic systems subjected to high strain-rate loadings, the explicit module/code is employed to estimate the response at the end of each time increment, followed by one or more adjustments to enhance not only the model convergence but also the simulation results/outcomes \[1\]. Note that the explicit module (Abaqus/Explicit) is often employed for extreme loading conditions (e.g., blast, impact, etc.) while the implicit module (Abaqus/Standard) is usually adopted to simulate quasi-static, static, cyclic, and low-speed dynamic problems.

Figure 8[17]. Computational time as well as the possibility of errors can be better optimized using an explicit module which makes it the most efficient solver with regard to computing solutions for fine-scale large numerical models \[52\].

Fig. 9. Different modules for numerical modeling and simulation in Abaqus.
Hence, Abaqus/Explicit is used in the present work to minimize numerical convergence issues. The “Nlgeom” (accounts for nonlinear geometry) in Abaqus/Explicit is kept ON for all the simulations presented herein. In addition, the hourglass control feature is enabled for all the considered 3D finite elements. Quick performance of explicit module/analysis results in better numerical convergence and simulation of the highly complex response of masonry components when constructed using fine-scale or microscopic modeling techniques. Further, this module does not require a complete global stiffness matrix for solving dynamic equations rather, it makes use of the theory of wave propagation [1] to solve or provide solutions for internal variables [17, 52]. However, the implicit module/analysis requires a global stiffness matrix and inverted it to provide implicit solutions for the generated dynamic equations. Note that the solutions obtained using an explicit module are conditionally stable, time increments are very small, and the simulation hardly requires a large number of iterations which makes the explicit solver much more suitable and useful for complex or highly nonlinear problems [17, 70].

Under extreme loading conditions, the loading (strain) rate affects the static properties (e.g., strength, elastic modulus, etc.) of the materials and is generally represented in terms of stress-increase factors (SIFs) or dynamic-increase factors (DIFs) [17]. When a structure or its elements is subjected to extreme loadings (e.g., blast, impact, etc.), a significant quantity of energy is abruptly transmitted to the materials of the structure through stress wave propagation, and the materials experience continuously varying strength/stiffness, elastic modulus, energy absorption and dissipation effects. Under high strain rates (e.g., $10^{-1}$ to $1000$ s$^{-1}$), the apparent strength might rise dramatically depending on the material being utilized [55]. In the literature [72], increases in strength of more than 50% for steel, more than 100% for standard concrete in compression, and more than 600% for concrete in tension have all been mentioned.

The strain rate influence on the material properties is introduced as SIF (or DIF), which is variable with the strain rate [17]. However, the available strain rate dependent constitutive models are not developed for masonry.

Figure 9 shows a flowchart of different modules in Abaqus for modeling and simulation of the analysis; a detailed description of these modules/steps is given in [1].

8. Summary

This paper provides a condensed overview of the different material models available in the Abaqus software suite that are utilized for the characterization of nonlinear behavior in various materials. It also includes a brief discussion on solving techniques, such as explicit versus implicit analysis, as well as dynamic or stress-increase factors. When it comes to modeling specific materials, different software packages like Abaqus, LS-DYNA, Air3D, and others have their own built-in material models, although their mechanisms are quite similar [55]. It is worth noting that the material models employed in this study have been widely used by researchers in recent years for modeling similar materials under extreme loading conditions, and they have proven to be highly accurate in simulating and predicting material response and damage. For more detailed information about the mechanisms behind these models, please refer to [1].

9. Parameters and Symbols

- $c$: Cohesion
- $C$: Undamaged Stiffness Matrix
- $C_d$: Damaged Stiffness Matrix
- $D_0^e_l$: Material Damaged Elastic Stiffness
- $d$: Scalar Damaged Stiffness Variable
- $D_0^u_l$: Material Undamaged Elastic Stiffness
- $E_0$: Undamaged Elastic Modulus of Material
- $F$: Failure Envelope or Yield Surface
- $f_{b0}$, $f_{c0}$: Stress ratio (ABAQUS User Guide, 2020)
- $f_w$: Bond Strength
- $K_c$: Shape Factor
- $K_{NN}, K_{SS}, K_{TT}$: Stiffness values for a traction (stress) in the normal direction (NN), and tangential directions (SS and TT)
- $p$: Hydrostatic Pressure
- $\overline{p}$: Effective Hydrostatic Pressure
- $p_c$: Hydrostatic Compression
- $p_0$: Initial Hydrostatic Compression
\( Q = \) Total Absorbed Energy
\( q = \) Von Mises Shear Stress
\( \overline{q} = \) Von Mises Equivalent Effective Stress
\( \overline{S} = \) Deviatoric Part of Von Mises Effective Stress Tensor
\( T_0 = \) Reference Temperature
\( T_{melt} = \) Melting Point
\( \alpha = \) Dimensionless Material Constant
\( \gamma = \) Dimensionless Material Constant
\( \varphi = \) Eccentricity
\( \varphi = \) Dilation Angle
\( \mu = \) Viscosity Parameter
\( \psi_{\text{critical}} = \) Critical Shear Stress
\( \mu_{\text{max}} = \) Maximum Effective Principal Stress
\( \sigma_{\text{c0}} = \) Initial Uniaxial Compression
\( \sigma_{\text{c}} = \) Compressive Stress
\( \sigma_{\overline{c}} = \) Effective Compressive Stress
\( \sigma_{\text{c0}} = \) Initial Compressive Stress
\( \sigma_{\text{cu}} = \) Ultimate Compressive Stress
\( \sigma_{\text{eq}} = \) Equivalent Von Mises Flow Stress
\( \sigma_{N_{\text{max}}} = \) Tensile Strength of the Joints
\( \sigma_{S_{\text{max}}} = \) Shear Strength of the Joints
\( \sigma_{T_{\text{max}}} = \) Shear Strength of the Joints
\( \sigma_{t_{0}} = \) Tensile Stress
\( \sigma_{t_{0}} = \) Initial Tensile Stress
\( \varepsilon_{\text{pl}} = \) Inelastic Strain
\( \varepsilon_{\text{eq}} = \) Equivalent Plastic Strain
\( \delta_{N}, \delta_{S}, \text{and} \delta_{T} = \) Traction Separations in normal (N), and tangential directions (S and T)

References

1. Abaqus CAE, “Damage Plasticity, explicit platform, material library, interactions, \( \ldots \) processing”, Help & Learning manual, ABAQUS DS \( \ldots \) Download.
2. Drucker, D. C., & Prager, W. Soil mechanics and plastic analysis or limit design. \( \ldots \) Quarterly of Applied Mathematics \( \ldots \) 1952.
8. Valente, M., & Milani, G. Nonlinear dynamic and static analyses on eight historical masonry towers in the north-east of Italy. \( \ldots \) Engineering Structures \( \ldots \) 2016.


