Advanced numerical simulations in concrete mechanics

Andrzej Winnicki¹, Adam Wosatko², Michał Szczecina³ and Jerzy Pamin⁴

¹, ², ⁴ Faculty of Civil Engineering, Cracow University of Technology, Cracow, Poland
³ Faculty of Civil Eng. & Architecture, Kielce University of Technology, Kielce, Poland

¹ andrzej@hypatia.L5.pk.edu.pl

ABSTRACT

The paper focuses on the following three issues: deterministic size effect (Wosatko et al., 2018), punching shear failure for flat slabs without shear reinforcement (Wosatko et al., 2019) and frame corners under opening bending moment (Szczecina, 2018). Two material models are used for the analyses: a gradient enhanced damage model developed by authors (GD) and Concrete Damaged Plasticity model (CDP) available in ABAQUS FEA.

1. INTRODUCTION

The paper presents the capabilities of two modern material models for concrete to render real behaviour of concrete elements under static loading. Three different cases are analysed and evaluated: deterministic size effect (Wosatko et al., 2018) using a gradient enhanced damage model developed by authors (GD), punching shear failure for flat slabs without shear reinforcement (Wosatko et al., 2019) using Concrete Damaged Plasticity model (CDP) available in ABAQUS FEA (SIMULIA, 2014) and failure of frame corners under opening bending moment (Szczecina, 2018), also using the CDP model.

The paper is intended as a brief overview of the obtained results. More details can be found in the sources quoted above. In general, the paper shows that the modern material models are mature enough to cope with the complicated real behaviour exhibited by concrete elements in real life problems.

¹Professor
²Assistant Professor
³Assistant Professor
⁴Professor
2. SIZE EFFECT

2.1 Summary of gradient damage model

In the simplest model of continuum damage mechanics one damage measure $\omega$, which grows from 0 (no damage) to 1 (complete loss of stiffness), is a function of damage history parameter $\kappa^d$ and depends on the deformation of a body. The concept of effective stress $\hat{\sigma}$, which acts on the undamaged material skeleton while actual stress $\sigma$ satisfies equilibrium, is used together with the assumption of strain equivalence in the real and effective (fictitious, undamaged) configuration. The stress tensors are related by scalar $\omega$:

$$\sigma = (1 - \omega) \hat{\sigma}$$
$$\hat{\sigma} = E : \epsilon^e$$  \hspace{1cm} (1)

where $E$ is the Hooke’s operator. The elastic strain tensor $\epsilon^e$ is equal to the strain tensor $\epsilon$ when the standard elasto-damage model is considered (this is the case explored in this paper), but the model can easily be coupled with a plastic behaviour of the undamaged material "skeleton" and then $\epsilon^e = \epsilon - \epsilon^p$.

The damage evolution in the gradient-enhanced model is governed by the following damage activation function, defined in the strain space:

$$F^d(\epsilon, \kappa^d) = \bar{\epsilon}(\tilde{\epsilon}()) - \kappa^d = 0$$  \hspace{1cm} (2)

where $\bar{\epsilon}$ is an equivalent strain measure and $\tilde{\epsilon}$ is an averaged (nonlocal) strain measure which satisfies the following diffusion equation:

$$\bar{\epsilon} - c \nabla^2 \bar{\epsilon} = \tilde{\epsilon}$$  \hspace{1cm} (3)

and homogeneous natural boundary conditions. The parameter $c > 0$, assumed here to be constant, has a unit of length squared and is related to an internal length scale. During the damage evolution the history parameter $\kappa^d$ is equal to the largest value of $\bar{\epsilon}$ reached in the loading history and obeys the standard loading/unloading conditions.

The equivalent strain measure $\tilde{\epsilon}$ can be defined in different ways. In this paper the modified von Mises definition is employed involving the first and second strain invariants, $I_1^\epsilon$ and $J_2^\epsilon$, respectively, and depending on the ratio of compressive and tensile strength $k = f'_c / f'_t$.

$$\tilde{\epsilon} = \frac{k - 1}{2k(1 - 2\nu)} I_1^\epsilon + \frac{1}{2k} \sqrt{\left( \frac{k - 1}{1 - 2\nu} I_1^\epsilon \right)^2 + \frac{12k}{(1 + \nu)^2} J_2^\epsilon}$$  \hspace{1cm} (4)

In this paper two examples are considered to verify if the regularized model described above is capable of representing the size effect.
2.2 Direct tension test

The first one is a direct tension test. The discretized double-edge notched bar is depicted in Figure 1, cf. (Hordijk, 1991). Three different specimens are considered, the dimensions of which are summarized in Table 1. However, the mesh density remains the same, i.e. the size of finite element changes together with the dimensions of the specimen. Plane stress conditions and the thickness \( t = 50 \, \text{mm} \) are assumed for all simulations. The so-called ligament width \( B_{\text{lig}} \) is defined, which is the net bar width in the notch region.

**Table 1: Direct tension test – geometry.**

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Length ( L ) [mm]</th>
<th>Width ( B ) [mm]</th>
<th>Measurement base ( L_m ) [mm]</th>
<th>Ligament width ( B_{\text{lig}} ) [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>500</td>
<td>120</td>
<td>70</td>
<td>100</td>
</tr>
<tr>
<td>B</td>
<td>250</td>
<td>60</td>
<td>35</td>
<td>50</td>
</tr>
<tr>
<td>C</td>
<td>125</td>
<td>30</td>
<td>17.5</td>
<td>25</td>
</tr>
</tbody>
</table>

**Table 2: Direct tension test – material model parameters.**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus:</td>
<td>( E = 40000 , \text{MPa} )</td>
</tr>
<tr>
<td>Poisson's ratio:</td>
<td>( \nu = 0.15 )</td>
</tr>
</tbody>
</table>

**GRADIENT DAMAGE (GD)**

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Strengths ratio:</td>
<td>( k = 10 )</td>
</tr>
<tr>
<td>Fracture energy:</td>
<td>( G_f = 115.0 , \text{N/m} )</td>
</tr>
<tr>
<td>Damage threshold:</td>
<td>( \kappa_o = 8.625 \times 10^{-5} )</td>
</tr>
<tr>
<td>Exponential softening:</td>
<td>( \alpha = 0.99, \eta = 500 )</td>
</tr>
<tr>
<td>Internal length scale:</td>
<td>( l = 4 , \text{mm}, \text{i.e.} , c = 8.0 )</td>
</tr>
</tbody>
</table>

The material model data are presented in Table 2. Notice that the internal length scale for gradient damage is the same for all specimens. In this test eight-noded finite elements are employed and indirect displacement control algorithm is applied with imposed horizontal displacement at the point marked in Figure 1 on the right-hand side of the dense mesh zone.
Next computations are performed for the unnotched beam under three point bending. The configuration, where a half of the domain is considered, is based on (Grégoire et al., 2013) and the finite element mesh is illustrated in Figure 2. Now, four specimens are simulated with the dimensions given in Table 3. As in the previous test, it has been assumed that the mesh density remains unchanged. Again, plane stress holds and thickness \( t = 50 \) mm is constant for all simulations.

The material data are listed in Table 4. Here, the internal length parameter for the gradient damage model is larger than for the previous test because the specimen dimensions are larger in comparison to the direct tension, the element dimension grows proportionally to the beam size and, to make the regularization active the simulated width of the localization band cannot be smaller than the element size.

Four-noded elements with full integration are employed and arc length control is used in the computations.

Figure 3 shows the diagrams of total force at the right-hand edge versus bar elongation for three specimen sizes. This illustrates the global response of the double-edge notched bar under tension. The concept of so-called ligament stress is introduced to observe the size effect:

\[
\sigma_{\text{lig}} = \frac{F}{B_{\text{lig}} t} \tag{5}
\]

where \( F \) is the force, \( B_{\text{lig}} \) is the ligament width and \( t \) is the thickness. Hence, in Figure 4
Table 4: Beam test – material model parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus:</td>
<td>$E = 37000$ MPa</td>
</tr>
<tr>
<td>Poisson’s ratio:</td>
<td>$\nu = 0.21$</td>
</tr>
<tr>
<td><strong>GRADIENT DAMAGE (GD)</strong></td>
<td></td>
</tr>
<tr>
<td>Strengths ratio: $k$</td>
<td>$k = 10$</td>
</tr>
<tr>
<td>Fracture energy: $G_f$</td>
<td>$G_f = 100.0$ N/m</td>
</tr>
<tr>
<td>Damage threshold: $\kappa_o$</td>
<td>$\kappa_o = 1.05 \times 10^{-4}$</td>
</tr>
<tr>
<td>Exponential softening $\alpha$</td>
<td>$\alpha = 0.99$, $\eta = 600$</td>
</tr>
<tr>
<td>Internal length scale: $l$</td>
<td>$l = 9$ mm, i.e. $c = 40.5$</td>
</tr>
</tbody>
</table>

the results are presented in terms of load divided by the net cross section in the notch area versus average strain calculated as extension of the measurement length over $L_m$.

In Figure 5 the distributions of the averaged strain measure $\bar{\epsilon}$ and the damage parameter $\omega$ are given. Although the damage profile has the unwelcome tendency to broaden with increased bar elongation, it is clearly seen that the width of the damage band (determined by the internal length $l$) is similar for all specimen sizes, which provides the proper representation of size effect.

![Figure 3](image)

**Figure 3:** Direct tension test – force-elongation diagrams for GD model.

Figure 6 shows the results for the GD model when all material model parameters are the same as in the previous simulation, but now the internal length scale $l$ is equal to 0, so in fact local damage model is adopted. It is observed that the size effect does not occur in this version of the model.

2.3 Unnotched beam

Figure 7 shows the diagrams of total force applied to the beam versus a pseudo-CMOD measured at the bottom surface between points $L_m$ away from each other, see Table 3. They are plotted for four beam sizes marked $D_1$-$D_4$ and compared with experimentally determined diagrams taken from (Grégoire et al., 2013). The numerical
Figure 4: Direct tension test – ligament stress vs average strain diagrams for GD model.

Figure 5: Direct tension test – distribution of averaged strain measure $\bar{\epsilon}$ and damage $\omega$ in notch vicinity for final states of gradient damage simulations.
response is much different than the experimental one, but no attempts have been made to follow the experiments more closely by taking different material model parameters. However, as shown in Figure 8, the gradient damage model exhibits a strong size effect. In these computations the so-called nominal stress is analyzed:

$$\sigma_{\text{nom}} = \frac{3}{2} \frac{F S}{t D^2}$$

where $F$ is the force, $S$ is the span, $t$ is the thickness and $D$ is the height of the beam.

Averaged strain is derived from the pseudo-CMOD divided by the base $L_m$. It is noted that for all specimen sizes the adopted value of the internal length scale is constant, while the size of finite element in the densified mesh region grows with the size of the beam (the discretization is the same for all sizes). Figures 9 and 10 show the final distributions of the averaged stain measure $\bar{\epsilon}$ and of damage $\omega$, respectively, for the four beam sizes. Notice that as expected, the widths of the fracture zones are similar because the size of the mesh zone shown grows proportionally with the specimen.
Figure 8: Beam test – nominal stress vs horizontal average strain diagrams for GD model.

Figure 9: Beam test – distribution of averaged strain measure $\bar{\epsilon}$ in final state for GD model.
When the presented results are analyzed and compared, objections can be raised and differences in the simulation of the bending response are noticed. First, quite large flexural strength of the GD model is seen in Figure 8 for the smallest specimen, which is caused by exaggerated gradient influence. Second, the maximum stress is too large for the large specimens, and hence the size effect is underestimated. The physics of the problem is as follows. First cracks occur when the uniaxial tensile strength \( f'_t \) is reached in the bottom fibers. Then, as cracking propagates, the force grows until the flexural tensile strength \( f_{t,\text{flex}} \) (also called the modulus of rupture) is reached. For small beams \( f_{t,\text{flex}} \) is much larger than \( f'_t \) since the fracture zone is large in comparison with the specimen. For large beams the fracture band width is relatively small as compared to the specimen size, the response is more brittle and the flexural strength should converge from above to the tensile strength. This is not simulated properly with the employed model.

3. CONCRETE DAMAGED PLASTICITY IN ABAQUS

Users of the commercial ABAQUS package (SIMULIA, 2014) can perform numerical analyses of concrete structures by three nonlinear material models: smeared cracking, brittle cracking, concrete damaged plasticity (CDP). The most popular and advantageous model seems to be the last one. In fact, the CDP model contains the most crucial and important features for quasi-brittle materials. However, some difficulties in the computational modelling can occur, for instance an unexpected divergence during the iterative process.

If the software is used in the ABAQUS/Explicit version then quasi-static analysis is
performed and only the so-called fracture energy trick is permitted to simulate behaviour of concrete structures in softening regime. This remedy is local and does not assure fully mesh-objective results. In the paper the static analysis is performed using ABAQUS/Standard and the viscous term is activated in the CDP model. The viscous regularization causes that the spurious mesh-sensitivity is reduced and better convergence can be obtained.

The paper is focused on the details of the plasticity theory contained in the CDP model, so the presence of damage and crack closing is ignored. Crucial features of the CDP model are described only in the context of the subject of discussion, i.e. the dilatancy angle and its impact on the computations for RC slab-column connection.

The stress rate $\dot{\sigma}$ is in the constitutive relation with the elastic strain rate $\dot{\epsilon}^e$. The plastic strain rate $\dot{\epsilon}^p$ according to the classical flow rule is expressed in terms of the plastic multiplier $\dot{\lambda}$ and direction of plastic flow $m$. The standard additive decomposition of strain rate into elastic and plastic parts is adopted:

$$\dot{\epsilon} = \dot{\epsilon}^e + \dot{\epsilon}^p, \quad \dot{\epsilon}^e = D^{-1} \dot{\sigma}, \quad \dot{\epsilon}^p = \dot{\lambda} m$$

where $D$ is the elastic stiffness operator.

The direction of plastic flow comes from the derivative of the plastic potential $G$:

$$m = \frac{\partial G_{CDP}}{\partial \sigma}$$

where $\alpha_d$ indicates a dilatancy factor.

### 3.1 Yield function

![Figure 11: Influence of parameter $K_c$ for yield surface $F_{cdp}$, 3D principal stress space, $f'_l = 4$ MPa, $f_c = 10 f'_l = 40$ MPa, $f'_c = 0.4 f_c = 16$ MPa, $f'_{bc} = 1.16 f'_c = 18.56$ MPa.](image)

**(a) $K_c = 0.6$.**  **(b) $K_c = 0.8$.**  **(c) $K_c = 1.0$.**

Non-associated plasticity is assumed with the following yield function

$$F_{CDP} = \frac{q + 3 \alpha p + \beta (\dot{\epsilon}^p) (\sigma_{max}) - \gamma (-\sigma_{max})}{1 - \alpha} - \sigma_c (\dot{\epsilon}^p) = 0$$

(9)
The variable $\tilde{\epsilon}^p$ denotes equivalent plastic strain, while $\tilde{\epsilon}_p^c$ is its compressive part. The subscript $\max$ in Eq. (9) refers to maximum principal stress. The well-known relationship is applied to distinguish the positive value of $\sigma_{\max}$ (if minus appears before this variable – then negative):

$$\langle \sigma_{\max} \rangle = (\sigma_{\max} + |\sigma_{\max}|) / 2$$  \hspace{1cm} (10)

Parameters $\alpha$ and $\gamma$ are calculated in the following way:

$$\alpha = \frac{f_{bc}' - f_c'}{2f_{bc}' - f_c'}, \quad \gamma = \frac{3(1 - K_c)}{2K_c - 1}$$  \hspace{1cm} (11)

where $f_{bc}'$ is initial biaxial compressive yield stress, $f_c'$ is initial uniaxial compressive yield stress and $K_c$ denotes the ratio of equivalent stress $q$ on the tensile and compressive meridians, which influences the shape of the yield surface. It is seen from Eq. (11) that parameter $K_c$ can be adopted in the interval $(0.5, 1.0]$. Three yield surfaces $F_{CDP}$ specified in three-dimensional principal stress space are illustrated in Fig. 11. The following data are applied: uniaxial tensile strength $f_t' = 4$ MPa, maximum compressive stress $f_c = 10f_t' = 40$ MPa and hence the initial uniaxial and biaxial compressive yield stresses are $f_c' = 0.4f_c = 16$ MPa and $f_{bc}' = 1.16f_c' = 18.56$ MPa, respectively. This set of data is typical for concrete. In general, the transition from tension to compression involves a change of surface shape, however for the above data and $K_c = 0.6$ three almost perpendicular patches of yield surface $F_{CDP}$ presented in Fig. 11(a) are smooth. For the second surface when $K_c = 0.8$ the shape transforms and becomes rounded because of the growth of this parameter, see e.g. Fig. 11(b). Finally, for $K_c = 1.0$ the yield function depicted in Fig. 11(c) changes into a cut cone. The default value for ABAQUS users is proposed $K_c = 2/3$ and will be applied in further computations. One more parameter which should be determined is the ratio $f_{bc}' / f_c'$. Experiments show that it ranges between $[1.10, 1.16]$ and value 1.16 is given as default. Function $\beta(\tilde{\epsilon}^p)$ is defined in the following way:

$$\beta(\tilde{\epsilon}^p) = \frac{\sigma_c(\tilde{\epsilon}_p^c)}{\sigma_t(\tilde{\epsilon}_t^p)} (1 - \alpha) - 1 - \alpha$$  \hspace{1cm} (12)

where stress-strain relationships should be previously described: $\sigma_c(\tilde{\epsilon}_p^c)$ – for uniaxial compression, $\sigma_t(\tilde{\epsilon}_t^p)$ – for uniaxial tension. Variable $\tilde{\epsilon}_t^p$ denotes tensile equivalent plastic strain. It means that different uniaxial stress state evolution for tension and compression regime is provided by the model.

### 3.2 Plastic potential function

The plastic flow potential in ABAQUS is defined in a similar fashion to the Burzyński-Drucker-Prager function:

$$G_{CDP} = \sqrt{(\tan \psi e f_t')^2 + q^2 + \tan \psi p}$$  \hspace{1cm} (13)

where $\psi$ is the dilatancy angle in the CDP model. The parameter $e$ is called the eccentricity and it is connected with the shape of the surface, particularly at its tip. Different
curvatures of the surface $G_{CDP}$ in three-dimensional principal stress space are depicted in Fig. 12. The default value of parameter $e$ is 0.1 and will be adopted in further computations. In the $p-q$ plane this function is rounded off, see Fig. 13(a). If eccentricity $e$ goes to 0 then plastic potential $G_{CDP}$ tends to the asymptote which is the simple linear relation:

$$G_{CDP}^a = q + \tan \psi p$$  \hspace{1cm} (14)

Here angle $\psi$ sets the slope of function $G_{CDP}^a$ in the $p-q$ plane. Both functions $G_{CDP}$ and $G_{CDP}^a$ can be expressed depending on the deviatoric variable $q$ in the following way:

$$G_{CDP} : \quad p(q) = -\sqrt{(ef'_t)^2 + (q / \tan \psi)^2}$$  \hspace{1cm} (15)

$$G_{CDP}^a : \quad p^a(q) = -q / \tan \psi$$  \hspace{1cm} (16)

and it can be proven that the limit of the difference $p^a(q) - p(q)$ for $q \to \infty$ is 0. If $q = 0$ then the difference of $p^a(q) - p(q)$ equals $ef'_t$. The strength $f'_t$ for uniaxial tension also has to be determined for function $G_{CDP}$.

The gradient of plastic potential in the CDP model is obtained as follows:

$$\frac{\partial G_{CDP}}{\partial p} = \tan \psi, \quad \frac{\partial G_{CDP}}{\partial q} = \frac{q}{\sqrt{(ef'_t \tan \psi)^2 + q^2}}$$  \hspace{1cm} (17)

The tangent to the plastic potential function $G_{CDP}$ is inclined at variable angle $\eta$ to axis $p$. For a sufficiently distant point $P$, see Fig. 13(b), the latter derivative which determines
angle \( \eta \) approaches the derivative for the asymptotic line:

\[
\frac{\partial G_{\text{CDP}}^a}{\partial p} \bigg|_P = \tan \psi, \quad \frac{\partial G_{\text{CDP}}^a}{\partial q} \bigg|_P = 1
\]

The tangent of angle \( \eta \) can be calculated at point \( P \) as:

\[
\tan \eta = \left. \frac{\partial G_{\text{CDP}}}{\partial q} \right|_P \rightarrow \tan \psi
\]

so it is shown that \( \eta \rightarrow \psi \). Thus this angle refers to high confining pressure.

### 3.3 Presence of viscous term

The local material model can be enhanced to avoid mesh-dependent results. Regularization can be achieved if an additional viscous term is incorporated in the constitutive relation. If ABAQUS/Standard is employed in the computations then the relaxation time \( \mu \) has to be set in the CDP model. In fact, the model includes the viscoplastic regularization according to a generalization of the Duvaut-Lions approach. The viscoplastic strain rate \( \dot{\epsilon}^{vp} \) is derived from the inviscid state as follows:

\[
\dot{\epsilon}^{vp} = \left( \epsilon^{p} - \epsilon^{vp} \right) / \mu
\]

The parameter \( \mu \) should be larger than zero and represents the viscosity. The relaxation time \( \mu \) decides about mesh-insensitive response and prevents numerical instabilities in the simulations.

### 4. PUNCHING SHEAR

#### 4.1 Geometry and material data

The geometry and material data come from the experiment (Adetifa and Polak, 2005), where the slab without shear reinforcement was examined as the control specimen SB1. The entire schedule of the experiment contained specimens with strengthening of the slabs by shear bolts, but these cases are not taken into account here. In the experiment the configuration with dimensions of the slab \( 1800 \times 1800 \times 120 \) mm and of the column
(a) Boundary conditions and load (displacement control) for mesh A.  
(b) Reinforcement in analyzed specimen.

**Figure 14:** Definition of specimen for slab-column connection.

**Table 5:** Detailed data for reinforcement.

<table>
<thead>
<tr>
<th>Reinforcement</th>
<th>Spacing [mm]</th>
<th>Cover (to axis) [mm]</th>
<th>Section area [mm²]</th>
<th>Yield strength [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>In slab:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tensile</td>
<td>100</td>
<td>24</td>
<td>100</td>
<td>455</td>
</tr>
<tr>
<td>Compressive</td>
<td>200</td>
<td>24</td>
<td>100</td>
<td>455</td>
</tr>
<tr>
<td>In column:</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bar</td>
<td>25</td>
<td>300</td>
<td>455</td>
<td></td>
</tr>
<tr>
<td>Ties</td>
<td>50</td>
<td>455</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

150 × 150 × 420 mm represented the behaviour of the actual interior connection in RC structures.

Some simplifications are employed in the numerical model. Firstly, the configuration is limited by simple supports, so dimensions 1500 × 1500 × 120 mm are considered. One quarter of the domain is taken into account, so boundary conditions for symmetric planes are imposed as it is displayed in Fig. 14(a). Moreover, in the experiment corners of the slab are clamped to avoid lifting. In the simulation the clamp is not included, but only two thirds of the nodes along each bottom edge at the corner are vertically supported, see again Fig. 14(a). This simplification is adopted to avoid unwanted cracking in the neighbourhood of the corner at the bottom surface. Static loading acts downward through the column as shown in Fig. 14(a). Displacement control is used. Perfect bond between concrete and reinforcement is assumed.

The spacing of the main reinforcement in the slab is doubled for the compression side as in the experiment. The location of the reinforcement is presented in Fig. 14(b) and detailed parameters are listed in Table 5. For steel Young’s modulus is $E_s = 205000$ MPa, Poisson’s ratio is 0.3 and linear hardening with modulus $h_s = 0.005E_s$ is given. The reinforcement is discretized by truss elements called T3D2 in ABAQUS.

The elastic constants for concrete are $E = 34400$ MPa and $\nu = 0.2$. The initial uniaxial tensile strength is $f'_{t1} = 2.13$ MPa. After cracking in tension regime linear softening is assumed to the point where the tensile strain reaches $\epsilon_{t1} = 0.0031$. The residual strength
The 2019 World Congress on
Advances in Structural Engineering and Mechanics (ASEM19)
Jeju Island, Korea, September 17 - 21, 2019

\(\sigma_\text{res} = 0.1 f'_c\) is then assumed in order to avoid numerical instabilities for zero stress.

Stress-inelastic strain relationship for compression is determined based on maximum uniaxial compressive strength \(f_c = 42.6\) MPa and depicted in Fig. 15(c). The initial yield surface \(F_{\text{CDP}}\) in 3D principal stress space is shown in Fig. 15(a). The plastic potential surface \(G_{\text{CDP}}\) is presented for dilatancy angle \(\psi = 25\) deg in Fig. 15(b). During the computations the dilatancy angle \(\psi\) is employed in the range between 5 deg and 55 deg to highlight its influence on the results. The analysis is performed using ABAQUS/Standard, so the viscous term can be activated in the model. Two different values of relaxation time \(\mu = 0.002\) s and 0.005 s are applied to compare results in the context of viscosity and dilatancy changes. The other model parameters are defaults as it is suggested in ABAQUS documentation.

4.2 Numerical results

The computations are performed for three finite element (FE) discretizations. For the better distinction there are introduced, according to the division along the height of the slab, the following letter symbols: mesh A for 5 finite elements (FEs), mesh B for 10 FEs.
and mesh C for 20 FEs. It means that the subsequent mesh has doubled density in each direction relative to the previous one. The total number of nodes are: for mesh A – 5862, for mesh B – 41225, for mesh C – 309489. The mesh refinement is shown in Fig. 16 together with example deformations in side view.

![Meshes](image)

**Figure 16:** Deformation of applied meshes for deflection $w \approx 12.5$ mm, $\mu = 0.005$ s, $\psi = 25$ deg.

![Load-displacement diagram](image)

**Figure 17:** Load-displacement diagrams, influence of dilatancy angle, mesh B, $\mu = 0.002$ s.

Fig. 17 shows load-displacement diagrams for different values of the dilatancy angle in comparison to the experiment. Only mesh B is now selected for this aspect of the analysis. The viscosity parameter $\mu$ is equal to 0.002 s. Quite large differences between equilibrium paths are clearly visible. For case $\psi = 5$ deg maximum force $P$ is over 140 kN, while for $\psi = 55$ deg final observed value of force $P$ is 300 kN and that curve could continue to grow up. The experimental load carrying capacity is about 250 kN. A similar value for numerical results is observed when $\psi = 35$ deg. However, it should be emphasized now that the slope of load-displacement diagram can be adjusted to the experimental curve also via other model parameters. This issue will be explained in further discussion.

The distributions of tensile equivalent plastic strain $\tilde{\epsilon}_p^t$ are depicted for the following cases: $\psi = 5$ deg – Fig. 18, $\psi = 25$ deg – Fig. 19, $\psi = 35$ deg – Fig. 20 and $\psi = 55$ deg – Fig. 21.

The strain field $\tilde{\epsilon}_p^t$ is also called PEEQT in the ABAQUS software. The tensile equivalent plastic strain distribution in the CDP model can be interpreted as smeared crack.
Figure 18: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh B, deflection $w \approx 12.5$ mm, $\mu = 0.002$ s, $\psi = 5$ deg.

Figure 19: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh B, deflection $w \approx 12.5$ mm, $\mu = 0.002$ s, $\psi = 25$ deg.

Figure 20: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh B, deflection $w \approx 12.5$ mm, $\mu = 0.002$ s, $\psi = 35$ deg.

Figure 21: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh B, deflection $w \approx 12.5$ mm, $\mu = 0.002$ s, $\psi = 55$ deg.
pattern. All the contour maps of PEEQT are prepared when deflection $w$ of the slab corresponds to 12.5 mm. Three perspectives are presented: side view which is simultaneously a vertical section, bottom view showing the tensile surface of the RC slab and a diagonal section. The punching cone is reproduced for two presented cases, when the dilatancy angle $\psi$ is equal to 5 and 25 deg. It is apparent for the side view, see Fig. 18(a) and 19(a). Firstly the slab is cracked near the column, but additional plastic strains are produced along skew conical surface at some distance from the connection. When the bottom view is observed, it is seen that the crack pattern is also reproduced along the flexural reinforcement of the slab in radial direction. On the other hand, for $\psi = 35$ deg and especially for $\psi = 55$ deg the PEEQT distribution is spreading and it is more associated with the bending mode than with the punching-shear failure phenomenon in the RC slab. The bottom view in Fig. 21(b) depicts the plastic zone which equally propagates from the center to supported sides and its character seems to be rather ductile than brittle.

![Figure 22: Load–displacement diagrams, influence of dilatancy angle $\psi$ and viscosity parameter $\mu$, mesh B.](image1)

The influence of the dilatancy angle $\psi$ together with non-zero time relaxation $\mu$ is illustrated in Fig. 22 for load-displacement diagrams. The equilibrium paths for $\psi = 25$ deg
Figure 24: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh A, deflection $w \approx 12.5$ mm, $\mu = 0.005$ s, $\psi = 25$ deg.

Figure 25: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh B, deflection $w \approx 12.5$ mm, $\mu = 0.005$ s, $\psi = 25$ deg.

Figure 26: Equivalent tensile plastic strain $\tilde{\varepsilon}_p^t$ for mesh C, deflection $w \approx 12.5$ mm, $\mu = 0.005$ s, $\psi = 25$ deg.
and $\mu = 0.002$ s as well as $\psi = 20$ deg and $\mu = 0.005$ s are close to each other and pass by each other. It is known that increasing of the viscosity parameter $\mu$ results in a stiffness increase and the response can involve artificially too high load-carrying capacity. For $\mu = 0.002$ s a smooth decrease can be noticed in both diagrams (red solid and purple dashed lines), while curves for $\mu = 0.005$ s (blue solid and brown dashed lines) go higher. On the other hand, it is clear that larger dilatancy angle $\psi = 25$ deg (blue and red solid lines) also raises the equilibrium path and load-carrying capacity. Hence, a given diagram can be controlled by both parameters, i.e. the value of the dilatancy angle $\psi$ and the viscosity parameter $\mu$ entail similar effects. Both parameters of the CDP model also decide about the distribution of PEEQT field. The numerical response for $\psi = 25$ deg and $\mu = 0.005$ s seems to be nearest to the experiment in the global sense, so this combination of the parameters is considered in the next comparison.

Mesh sensitivity study is performed to demonstrate not only the differences between diagrams but also the quality of simulation for crack patterns. The load-displacement diagrams in Fig. 23 are prepared based on the same material parameters and only the mesh densities (see Fig. 16) vary. A smaller difference is observed between mesh B and C, as expected. This indicates that the results for both meshes are acceptable. Fig. 24 depicts the distribution of the equivalent tensile plastic strain $\tilde{\varepsilon}_p^E$ for mesh A in three perspectives and for deflection $w \approx 12.5$ mm, analogically to the previously presented contour plots. It seems that the crack propagation is diffuse and the bending mode dominates over the punching failure. Moreover, the mesh with 5 FEs along the thickness of the slab is not able to reproduce the PEEQT distribution correctly. The crack patterns in Figs 25 and 26 are more pronounced. The punching cone is clearly visible for mesh C, when 20 finite elements are applied along the thickness of the slab. It is also visible in Fig. 27 for the isosurface of $\tilde{\varepsilon}_p^E$ which equals 0.02. The red inclined cone spreads from the column to the tensile surface the slab. It is in accordance with the experimental crack patterns.
5. FRAME CORNER UNDER OPENING MOMENT

5.1 Comparison with experiments

In (Johansson, 2001) performed laboratory tests on RC frame corners under closing and opening bending moment are reported. Numerical simulations are made and compared with laboratory test of the specimen marked as RV9 in research. Geometry of the specimen and test setup is presented in the Fig. 28. In the test a displacement control was applied with a displacement rate from 0.15 to 0.30 mm/min. Reinforcement provided for the laboratory test consisted of looped main bars and a diagonal bar.

Figure 28: Geometry and test setup of specimen RV9.

For numerical modelling material properties were assumed the same as in the experimental research: concrete: $f_c = 32.2$ MPa, $E_c = 31$ GPa, $\nu = 0.2$, $f_t = 2.6$ MPa, $G_f = 136.4$ N/m; reinforcing steel: $f_y = 570$ MPa, $E_s = 200$ GPa, $\nu = 0.3$ (ideal elasto-plastic). Main reinforcement of adjoining members was assumed as $5\phi_{16}$ bars, additional diagonal bars as $3\phi_{16}$. A general view of the specimen, boundary conditions applied in ABAQUS and reinforcing bars are presented in the Fig. 29. The RV9 specimen was meshed using mainly quadrilateral finite elements which size differs from 75 to 5 mm – see Fig. 30. Calculations were performed in plane stress state using ABAQUS/Implicit and full-Newton solution technique. A nodal displacement was imposed identically as in laboratory tests (displacement control was applied). Parameters of the CDP model were assumed as follows: dilatation angle 15 degrees, eccentricity 0.1, $f_{60}/f_{c0} = 1.16$, $K = 0.667$ and relaxation time equal to 0.0001 s.

For calculations of the crack width a formula proposed in (Červenka et al., 2012) was applied

$$w = \epsilon_{cr} \left[1 + \gamma_{max}(1 - \frac{\theta}{45})\right] L_t$$

(21)

where $L_t$ is a finite element width perpendicular to the direction of a crack, $\epsilon_{cr}$ denotes tensile strain in a cracked element, $\gamma_{max}$ is assumed as equal to 1.5 and $\theta$ is a crack propagation angle (given in degrees). Determination of $L_t$ and $\theta$ is presented in the Fig. 31, wherein $\theta$ is assumed as a smaller value from $\theta_1$ and $\theta_2$. 
Figure 29: Boundary conditions and reinforcement of specimen RV9.

Figure 30: Meshing of specimen RV9 with a zoomed corner zone.

Figure 31: Determination of $L_t$ and a crack propagation angle.
A relationship between horizontal reaction force and an imposed displacement is presented in the Fig 32 and compared with the relationship obtained in laboratory test (dashed line). Both paths seem to be similar, however the curve obtained in ABAQUS has no peak and no decreasing part. Instead of this it has a clear plateau. Moreover, the curve obtained in FEM shows a higher stiffness at the beginning of a loading process. Despite this both curves are comparable and a FEM analysis reflects properly laboratory tests.

A crack pattern obtained in FEM analysis is presented in the Fig. 33 in four selected step time values, where time value \( t = 1 \) s stands for a maximal displacement in a displacement-control procedure (120 mm). The first figure presents a map of PEEQT at the very beginning of a loading process. A clear but still not that large crack propagating from the concave angle of the corner zone is visible. Using Eq. 21 for this step a value \( w = 0.25 \) mm is obtained. Afterwards two new concurrent cracks appear further from the concave angle as shown in the second map of PEEQT. At this step time the maximal crack width is \( w = 0.57 \) mm which is a very large value comparing to typical values of ultimate crack widths given by codes and handbooks. In the third map a new crack propagating in the corner zone where reinforcement is loop-shaped is seen. The current crack width is \( w = 1.84 \) mm. The fourth figure presents PEEQT map at the end of FEM calculations and some new cracks and extensions of previous crack appear. Eventually the crack width is \( w = 8.10 \) mm.

The final crack pattern can be compared with the crack pattern obtained in laboratory test and presented in the Fig. 34. There are some similarities between FEM simulation and laboratory results: crack propagation from the concave corner and cracks outside the corner zone perpendicular to the axes of adjoining elements. There is also a very similar crack propagation from the concave corner. However, there is one clear difference – there are cracks located along loop-shaped fragments of reinforcing bars and none of them appears in FEM analysis.

5.2 Parametric studies

In order to investigate influence of the reinforcement detailing on the corner efficiency some parametric studies are made. The analysed reinforcing details are shown in Fig. 35. Reinforcement provided in different details is explained in Table 6.

Meshing of concrete specimen and reinforcing steel are presented in the Fig. 36 on the example of reinforcement detail No 7. Boundary conditions and loading of the corner are presented in the Fig. 37. To avoid a localization of strain and stress and numerical problems because of concentrated forces acting on specimen, some fragments of main reinforcement and concrete outside the corner zone were defined as ideally elastic. A range of the elastic fragment is shown in the Fig. 37.

Each corner is loaded by pure opening bending moment modelled as a pair of forces. The applied load is defined with load parameter \( \lambda \) which value 1 stands for bending moment equal to 30 kNm. The corners are calculated both in plane stress and in plane strain state.
Figure 32: Force-displacement relationship obtained in FEM analysis and laboratory test.

Figure 33: Equivalent plastic strains in tension PEEQT in four selected step time values.
The following values of the CDP model parameters are applied in analyses: dilatation angle $\psi = 15$ deg, flow potential eccentricity $e = 0.1$, ratio $f_{00}/f_{0} = 1.16$, ratio $K = 0.667$, viscosity parameter $\eta = 0.0001$ s, fracture energy $G_f = 146.5$ N/m.

Maps of equivalent plastic strains in tension PEEQT for some chosen reinforcement details in plane stress and strain state are presented in Fig. 38. For the sake of brevity only details No 1, 3, 4 and 7 are discussed. The use of at least one diagonal stirrup causes that cracks occur outside the corner zone, which is clearly visible both in plane stress and plane strain state.

A relationship between nodal displacement and loading parameter is presented in Figs 39 and 40, where the loading parameter $\lambda = 1.09$ corresponds to the corner efficiency factor $\eta = 1$. Analyzing this relationship stiffer behavior of corners equipped with

| Table 6: Detailed data for reinforcement. |
|-----------------|-------------------------------------------------------------------------------------------------|
| Detail No       | Provided reinforcement, diameters given in [mm]|  |
| 1               | main reinforcement: 2$\phi$ 20 top and 2$\phi$ 20 bottom, no loops |  |
| 2               | 2 diagonal bars $\phi$ 8 each, no loops |  |
| 3               | 2 diagonal bars $\phi$ 8 each, no loops |  |
| 4               | diagonal stirrup $\phi$ 16, looped main bars |  |
| 5               | central diagonal stirrup $\phi$ 16, outside stirrups $\phi$ 10, looped main bars |  |
| 6               | central diagonal stirrup $\phi$ 16, outside stirrups $\phi$ 12, looped main bars |  |
| 7               | 2 diagonal bars $\phi$ 16 each, central diagonal stirrup $\phi$ 12, outside stirrups $\phi$ 16, looped main bars |  |
Figure 35: Reinforcing details No 1 – 7.

the reinforcement details from No 4 to 7 can be observed both in plane stress and in plane strain state. Finally, the relationship of crack width versus loading parameter presented in Figs 41 and 42 confirms that the use of diagonal stirrups alone or combined with diagonal bars is recommendable for corners under opening bending moment. In these graphs a commonly used value of ultimate crack width 0.3 mm is marked with a vertical line. It is clearly seen that for the reinforcement details No 4 to 7 this ultimate value is reached for relatively high values of the loading parameter.

Values of the efficiency factors for all details are listed in the Table 7. Higher values of the efficiency factor appear for reinforcement details No 4 to 7 which confirms that these details are recommendable for a practical use. Moreover, the results obtained in FEM in plane stress state are comparable with those obtained in S&T method (Szczecina, 2018).

<table>
<thead>
<tr>
<th>Detail No</th>
<th>Efficiency factor S&amp;T</th>
<th>Efficiency factor FEM – plane stress state</th>
<th>Efficiency factor FEM – plane strain state</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.76</td>
<td>0.75</td>
<td>1.01</td>
</tr>
<tr>
<td>2</td>
<td>0.70</td>
<td>0.79</td>
<td>1.11</td>
</tr>
<tr>
<td>3</td>
<td>0.72</td>
<td>0.82</td>
<td>1.17</td>
</tr>
<tr>
<td>4</td>
<td>1.03</td>
<td>1.23</td>
<td>1.26</td>
</tr>
<tr>
<td>5</td>
<td>1.32</td>
<td>1.24</td>
<td>1.29</td>
</tr>
<tr>
<td>6</td>
<td>1.36</td>
<td>1.27</td>
<td>1.29</td>
</tr>
<tr>
<td>7</td>
<td>1.43</td>
<td>1.23</td>
<td>1.32</td>
</tr>
</tbody>
</table>

To summarize the obtained results – it is observed that the use of at least one diagonal stirrup significantly helps to limit a crack propagation and a crack width in the corner zone and to reach relatively high values of the corner efficiency factor. The best results seem
Figure 36: Meshing of concrete and reinforcing steel (e.g. detail No 7.) and analyzed nodal displacement.

Figure 37: Boundary conditions and ideally elastic fragments of reinforcement (in red color).
Figure 38: Equivalent plastic strains in tension PEEQT – Reinforcing detail No7.
Figure 39: Nodal displacement vs loading parameter in plane stress state.

Figure 40: Nodal displacement vs loading parameter in plane strain state.
Figure 41: Crack width vs loading parameter in plane stress state.

Figure 42: Crack width vs loading parameter in plane strain state.
to occur for the reinforcement detail No 7. On the contrary, the use of a diagonal bar without diagonal stirrups (details No 2 and 3) does not help at all.

6. FINAL REMARKS

In the paper the results for three different problems have been presented: deterministic size effect using a gradient enhanced damage model developed by authors (GD), punching shear failure for flat slabs without shear reinforcement using Concrete Damaged Plasticity model (CDP) available in ABAQUS FEA and failure of frame corners under opening bending moment, also using the CDP model. The results show that the modern material models for concrete are able to capture real behaviour of concrete elements in a reliable way.

However, the presented analyses point out clearly that the choice of proper values of material model parameters has to be made with great care.

Moreover, the last presented example shows that the advanced numerical simulations for concrete elements can be used as a design tool - in the analysed case giving a valuable insight into proper reinforcement detailing for RC frame corners.

REFERENCES


