Estimating the ultimate energy dissipation capacity of steel pipe dampers

J. Utomo\textsuperscript{a}, M. Moestopo\textsuperscript{b}, A. Surahman\textsuperscript{c}, D. Kusumastuti\textsuperscript{b}

\textsuperscript{a}First affiliation, Address, City and Postcode, Country
\textsuperscript{b}Second affiliation, Address, City and Postcode, Country

Abstract

In passive control systems, the energy input from earthquakes is delivered to special devices called seismic dampers. Passive energy dissipation systems are now recognized as an effective and inexpensive way to mitigate earthquake risks to structures. This paper presents two purposes. The first one is to elaborate a practical component integrity assessment using finite element ductile fracture simulation based on local approach. The second one is to estimate the ultimate energy dissipation capacity of steel pipe dampers using energy based damage model. Damper specimen tests are very important but expensive and time consuming therefore efficient tools are needed to minimize the expensive and time-consuming tests. Ductile fracture in steel components that happens in fewer than twenty constant amplitude loading cycles is known as Ultra Low Fatigue Cycle (ULCF). Under ULCFs load steel pipe dampers experienced extensive plasticity and limited cyclicity. Ductile fracture controls the ultimate strength and ductility of structural components, therefore accurate preliminary prediction of ductile fracture is critical to estimate the performance of steel pipe dampers. Steel pipe dampers absorb seismic energy input through hysteresis of the metal. The hysteretic behavior and ultimate energy dissipation capacity are investigated via finite element simulation after the component integrity assessment has been done. A micromechanics-based model which provide accurate criteria for predicting ductile fracture and an energy-based damage model to quantify the ultimate energy dissipation capacity of steel pipe dampers are applied. Using these approaches, ultimate energy dissipation capacity of steel pipe dampers can be estimated under various patterns of loadings. The approaches described here can also be applied to other steel dampers subjected to randomly applied flexural/shear stress reversals, such as those induced by earthquakes.

© 2015 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of organizing committee of The 5th International Conference of Euro Asia Civil Engineering Forum (EACEF-5).

Keywords: ductile fracture; energy dissipation capacity; hysteretic behavior; micromechanics-based model; steel pipe damper

1. Introduction

A try-out research to investigate the potential of circular steel pipes as metallic dampers had been done. Vertical steel pipes, strengthened with two tapered plates attached at the outside of the pipe and lead filled inside the pipe, were identified as one of potential dampers. For practical reason some modifications were done to the identified damper, the lead was substituted by three inner rings and the two tapered plates were substituted by two trapezoidal plates. The modified damper and its components are shown in Fig. 1. Under cyclic loading the role of the rings was to stabilize the pipe cross section and was not designed to yield while the pipe and trapezoidal plates were designed to yield to dissipate induced earthquake input energy in parallel. Finite element ductile failure simulations based on local approach was conducted to predict fracture at identified spots in the damper. The hysteresis curve of the circular steel pipe was then used to estimate the ultimate energy dissipation capacity and the accumulated plastic deformation of the
strengthened vertical steel pipe damper. The dampers can be installed in buildings or bridges as passive energy dissipaters to enhance the seismic protection of those structures.

![Vertical steel pipe damper](image1)

**Fig. 1.** Vertical steel pipe damper: strengthened with three rings and trapezoidal plates (the pipe is not shown on the right picture)

### 2. Numerical simulation and ductile fracture prediction

Schedule 80 carbon steel pipe and steel plate bought at local market were used as the material of the damper. Fig. 2 shows the results of simple tensile tests done to obtain the steel pipe and steel plate properties. These properties are used as material data for ABAQUS.

![Stress-strain relation](image2)

**Fig. 2.** Stress-strain relation of the steel pipe and steel plate: tensile test results

From practical consideration, steel pipes with diameter greater than 100 mm were considered to be useable as dampers. In this study the steel pipe of 114.3 mm diameter, available in the local market, was chosen. Most part of the pipe is expected to yield due to cyclic loading. Abebe, Kim and Choi[1] demonstrated that in order the developed stresses caused by both bending and shear occurred simultaneously, the height to diameter ratio of the pipe should be equal to $\sqrt{3}$. Therefore the height of the pipe is equal to $\sqrt{3} \times 114.3 = 197.97 \text{ mm} \sim 200 \text{ mm}$. The thickness of the pipe is 8.6 mm. A trial based on the needed strength of the welded connection at the top and bottom of the damper was done to find the minimum width of the top and bottom of the trapezoidal plate strengthener. The minimum width at the top and bottom was found about 50 mm and the width of the middle part was kept minimum (20 mm). The thickness of the plates were 12 mm. Each ring for stabilizing the cross-section of the pipe was welded at four points to the inner side of the pipe wall (Fig. 1). The thickness of the rings was 12 mm.

#### 2.1. Finite element simulation

ABAQUS was used to simulate the damper under lateral cyclic loading. A general purpose linear brick element (C3D8R) was used to mesh the damper. This element has one integration point located at the middle of the element. Stresses and strains were evaluated at the integration point. For the purpose of simulating hysteretic behavior and fracture prediction of the damper, bilinear model of the material with combined hardening were used for the ABAQUS simulation. The yielding load ($Q_y$) and the yielding displacement ($\delta_y$) corresponding to the yielding load $Q_y$ were estimated by first applying monotonic lateral load to the damper. It was found that $Q_y \sim 430000 \text{ N}$ and $\delta_y \sim 0.821 \text{ mm}$. Modified Krawinkler loading protocol was then used to simulate the hysteretic behavior of the damper. This protocol consists of: (1) three set of six cycles; the amplitude of the cycle is constant within each set but it is increased for every consecutive set of cycles, following the sequence $0.375\delta_y$, $0.50\delta_y$, and $0.75\delta_y$; (2) one set of four cycles with constant amplitude $1.0\delta_y$; (3) five set of two cycles; the amplitude of the cycle is constant within each set but it is changed for...
every consecutive set of cycles, following the sequence 0.375δ_y, 1.50δ_y, 0.375δ_y, 2.0δ_y, and 0.375δ_y; (4) finally the amplitude in each consecutive cycle is increased by 1.0 δ_y up until failure if necessary.

2.2. Ductile fracture prediction

A simple criterion has been established to predict the failure of the steel pipe dampers due to the interaction effect of fracture and fatigue known as ULCF. Fracture occurs when micro voids initiating at sulphide or carbide inclusions grow under plastic strains, leading to micro void coalescence. Stress Modified Critical Strain (SMCS), developed based on the concept of tracking micro void growth and coalescence, is one of such criteria [2]. In SMCS model, a critical value of plastic strain, ε_p critical, is related to stress triaxiality T and parameter a by the following equation:

$$\varepsilon_p \text{ critical} = a \cdot \exp (-1.5T) \hspace{1cm} (1)$$

The SMCS criterion, defined as the difference between the critical plastic strain and the calculated equivalent plastic strain (ε_p), is:

$$\text{SMCS} = \varepsilon_p - \varepsilon_p \text{ critical} \hspace{1cm} (2)$$

Fracture is predicted to occur when SMCS = 0. SMCS model is simple to be applied to preliminary predict when ductile fracture will occur under ULCF condition.

For ductile fracture prediction a circumferentially notched tension bar (CNT), assumed to be extracted from the pipe, was simulated. The result of the simulation is shown in Fig. 3. Ramberg Osgood model was used to model the strain hardening of the material. Fracture displacement, which should be obtained from CNT test, was needed. However no CNT test was done. Therefore base on the CNT test result of similar steel done by Myer, Deierlein & Kanvinde[3], the fracture displacement (Δ_f = 0.92 mm) was used to calculate the equivalent plastic strain and the material resistance to fracture (a). Parameter a was found equal to 1.97. Once a is determined, it can be implemented through finite element simulations to predict fracture initiation in steel pipe dampers.

![Fig. 3. Circumferentially notched tension bar (CNT) simulation](image)

The von Mises yield criterion was used to identify spots with intense stresses. The identified spot for the vertical steel pipe damper was at the top and bottom of the trapezoidal plate strengtheners as shown in Fig. 4. During nonlinear simulation, a request was sent to ABAQUS to generate three principal stresses (S11, S22 and S33), von Mises stress (σ_v) and plastic strain equivalent (PEEQ) of the spot. The hydrostatic stress was calculated as σ_m = (S11+S22+S33)/3 and the triaxial stress was T = σ_m/σ_v. Plot of triaxial stress T vs. plastic strain equivalent PEEQ is also shown in Fig. 4. It can be seen that the triaxial stress T does not change much with the increase of the plastic strain equivalent. Therefore it can be concluded that ε_p critical in equation (1) and (2) depends on instantaneous T so that the loading history can be neglected. The positive and negative triaxial stress T was then separated and the results were shown in Fig. 5. From equation (1), using a = 1.97 and T = 3.7, the critical plastic strain is ε_p critical = 1.13. The SMCS criterion in equation (2) can be applied with the help of Fig. 5 which corresponds to the time t ~ 190 second. From loading protocol, the t = 190 second corresponds to the amplitude = 20.52 mm.

The location of the spot of intense stress is close to the welded connection meaning fracture is expected to happen at heat affected zone (HAZ). Myers, Deierlein and Kanvinde[3] have shown that fracture toughness is degrading in the HAZ of a welded connection under ULCF loads. Fracture toughness in the HAZ is approximately 50% smaller than that of base metal. Using engineering judgement, ductile fracture is expected two or three cycle less than the number of cycle predicted using SMCS. Therefore ductile fracture is expected to happen at reduced amplitude corresponding to the amplitude = 17.1 mm. The estimated Q–δ curve of the
damper before ductile fracture happened is shown in Fig. 5. The $Q$–$\delta$ curve can then be used to estimate the energy dissipation capacity and accumulated plastic deformation of the vertical steel pipe damper.

Fig. 4. The spot with intense stress and the corresponding plot of $T$ against $\text{PEEQ}$

Fig. 5. Positive and negative triaxiality of the spot with intense stress, and reduced $Q$–$\delta$ curve of the damper

3. Energy dissipation capacity and accumulated plastic deformation

Based on the researches done in Japan since the mid of 1970, Benavent-Climent[4] proposed an energy-based damage model for seismic response of steel structures. Benavent-Climent, Morillas & Vico[5] applied the proposed model for seismic damper design. The proposed model is time consuming if it is applied manually. Therefore some routines were coded using MATLAB to digitalize the proposed model. Outputs from numerical simulation using ABAQUS were exported into Excel. All data needed by the proposed model were then calculated in Excel and imported into MATLAB. On the top of the imported data, some routines were built that can be used to estimate the energy dissipation capacity and accumulated plastic deformation of the vertical steel pipe damper. Later on the same routines can be used to calculate the actual energy dissipation capacity and actual accumulated plastic deformation using the hysteresis curve from the test result of the damper.

Following energy-based damage model proposed by Benavent-Climent[4], the $Q$–$\delta$ curve is decomposed into skeleton part and Bauschinger part, in the positive and negative domain of loading, as shown in Fig. 6. The paths that exceed the load level attained by the preceding cycle in the same domain of loading are connected sequentially resulting two curves shown in Fig. 7. The skeleton part was approximated by three lines. The first line starting from the origin was the elastic stiffness $K_e$, the second starting from the end of the first line was the first plastic stiffness $K_{p1}$ and the third line was the second plastic stiffness $K_{p2}$.

Fig. 6. Original and decomposed $Q$–$\delta$ curve
The area enveloped by each segment in the skeleton and Bauschinger part was calculated using a MATLAB function `polyarea`. Parameters were expressed in nondimensional form as follows: the first plastic stiffness $k_{p1}=K_{p1}/K_e$, the second plastic stiffness $k_{p2}=K_{p2}/K_e$ and $\tau_B=Q_B/Q_y$ (see Fig. 7). From numerical simulation of the steel pipe damper, the values of $Q_y=43000N$, $\delta_y=0.821$ mm, $Q_B=466000N$, $k_{p1}=0.0434$ and $k_{p2}=0.0048$ were found. If $\eta_p$ represents the cumulative plastic deformation ratio (normalized by $\delta_y$) on the skeleton part and $\eta$ represents the ultimate energy dissipation capacity of the damper (normalized by $Q_y\times\delta_y$) then $\eta$ can be calculated using the following equations proposed by Benavent-Climent, Morillas & Vico[5]:

$$\eta_p \leq 2(\tau_B - 1)(1 - k_{p1})/k_{p1}$$

$$\eta = 0.25\eta_p(1 - k_{p1}) + \eta_p(1+a) + b \quad (3)$$

$$\eta > 2(\tau_B - 1)(1 - k_{p1})/k_{p1}$$

$$\eta = 2(\tau_B^2 - 1)(1 - k_{p1})/k_{p1} + (\eta_p/2 - (\tau_B - 1)(1 - k_{p1})/k_{p1} \times 2\tau_B + (k_{p2}/(1 - k_{p2}))\times \eta_p/2 - (\tau_B - 1)(1 - k_{p1})/k_{p1}) + a\eta_p + b \quad (4)$$

The last two terms in equation (4) represent the energy dissipated on the Bauschinger part. Benavent-Climent, Morillas & Vico[5] demonstrated that the relation between $\eta_p$ - $\eta$ is linear and can be expressed as:

$$\eta = a\eta_p + b$$

The relation between $\eta_p$ - $\eta$ is graphically shown in Fig. 8. Testing two specimens under different patterns of cyclic loading is required to determine the actual parameter $a$ and $b$. The estimated value of $a = -12$ and $b = 1050$ are used to draw the eqn. (3) and eqn. (4).
In non-dimensional form, the values of ultimate energy dissipation capacity and accumulated plastic deformation of the vertical steel pipe damper are predicted as \(593.57\) and \(40.57\). The history of the predicted values at each cycle of loading up to 20 cycles is shown in Fig. 8 as well. If desired, applying the same procedures, the energy dissipation of the damper using other pattern of cyclic loading can be explored.

After defining the shape of the skeleton part and the Bauschinger part using parameters \(Q_y, Q_B, \delta_y, kP_1, kP_2\), and making two lines approximation on the Bauschinger part, Benavent-Climent, Morillas & Vico[5] simulated the hysteresis curve as shown on Fig 9. If one would like to obtain the performance of the damper under any real earthquake record, the upper peaks corresponding to half lifetime of the hysteretic curve can be extracted to construct a backbone curve. The backbone curve, which represents the strength envelope of the damper, can be inputted to commercial software such as Peform-3D.

![Fig. 9. Simulated hysteresis curve and backbone curve](image)

4. Conclusion

The main purpose of developing steel pipe dampers is to seek dampers can be utilized as stable source of energy dissipation. Anticipating damper’s failure due to ductile fracture under applied cyclic load, prior to testing the specimen of the damper, is one of the crucial steps in developing damper. Ductile fracture is a limit state which should be considered because it controls the ultimate strength and ductility of the damper. The stiffness, strength and ductility which characterize the damper can be quantified using nonlinear finite element simulation combined with damper integrity assessment. Studying these parameters and the shape of the \((Q–\delta)\) curve will indicate whether the damper has a stable hysteretic behavior and high energy dissipation capacity. The digitalized energy based damage model applied to the \(Q–\delta\) curve from finite element simulation allows one to predict of the energy dissipation capacity of the damper. After the specimen test is done, the same digitalized procedures can be applied to the test result \(Q–\delta\) curve to quantify the actual energy dissipation capacity of the damper.

Acknowledgement

This research was supported by the Decentralization Research Grant (FTSL PN-1-08-2014) of Directorate General of Higher Education (DIKTI), Ministry of National Education, Indonesia.

References

[1] D. Y. Abebe, J. W. Kim, and J. H. Choi, “Hysteresis Characteristics of Circular Pipe Steel Damper Using LYP225,” 2013, no. February, pp. 3–10.
[2] A. M. Kanvinde and G. G. Deierlein, “Prediction of Ductile Fracture in Steel Connections Using SMCS Criterion,” no. February, pp. 171–182, 2006.
[3] A. T. Myers, G. G. Deierlein, and A. Kanvinde, “TESTING AND PROBABILISTIC SIMULATION OF DUCTILE FRACTURE INITIATION IN STRUCTURAL STEEL COMPONENTS AND WELDMENTS,” 2009.
[4] A. Benavent-climent, “An energy-based damage model for seismic response of steel structures,” Earthq. Engng Struct. Dyn., no. January, pp. 1049–1064, 2007.
[5] A. Benavent-climent, L. Morillas, and J. M. Vico, “A study on using wide-flange section web under out-of-plane flexure for passive energy dissipation,” Earthq. Engng Struct. Dyn. 2011; , no. July 2010, pp. 473–490, 2011.