Thin–Walled Structures 111 (2017) 65–79

Contents lists available at ScienceDirect

Thin–Walled Structures

journal homepage: www.elsevier.com/locate/tws

Full length article

Cyclic behavior and finite element modeling of wide flange steel bracing members

Madhar Haddad

Department of Architectural Engineering, United Arab Emirates University, P. O. Box 15551, Al Ain, UAE

A R T I C L E   I N F O

Keywords:
Finite element model
Wide-flange bracing members
k-area
Ultra-low cycle fatigue
Fracture life

A B S T R A C T

Wide-flange (WF) bracing members are being used increasingly in concentrically braced frames (CBFs). The limited number of tests WF members performed to date highlights the need for a detailed finite element model with a reliable fracture rule to understand the hysteresis behavior and to predict the fracture of WF braces subjected to cyclic loading. Building a reliable finite element model is cost effective compared to the high expense of experiments, especially when brace sizes are large and thus require a high capacity test frame with powerful actuators. Nine WF quasi-static cyclic brace tests (Fell et al., 2006; Tremblay et al., 2008; Richard, 2009) are used to validate the finite element model presented here. Material variations on the cross-sectional level are considered. Results indicate that the current model is able to simulate the overall hysteresis behavior of the WF braces accurately. Fracture in the plastic hinge that developed at the mid-length of specimens tested by Tremblay and Richard is predicted through a calibrated cyclic void growth model (CVGM) for ultra-low cycle fatigue. In addition, the current model is able to simulate the shift in the location of mid-length plastic hinge. The effects of mesh density, number of integration points, boundary conditions, initial imperfection, initial yield strength, and loading protocol on the hysteresis behavior of WF bracing members are also investigated.

1. Background and objectives

Finite element and fiber models are commonly used in modeling bracing members. A finite element formulation provides the best utility for modeling local behavior at hinge locations and the progressive failure of a member. Finite element models (Haddad et al. [1–4], Uriz et al. [5,6], Fell et al. [7–9], Myers et al. [10]) are able to simulate the phenomena of local buckling better than the fiber-models (Aguero et al. [11], Uriz et al. [6], Hsiao [12], Santagati et al. [13], Chen and Tirca [14], Haddad, et al. [4]) that are frequently used in the analysis of braced-frames. Thus, fiber-models may not be the most suitable for assessing the localized behavior which can lead to fracture.

In his analytical study, Prathuangsit [15] found that the effect of cross-sectional properties on the hysteresis loops is negligible in cases where lateral torsional buckling does not occur. Based on the limited data, the effect of cross-sectional shape on the hysteresis behavior is generally insignificant except in the maximum compressive load region (Jain et al. [16]). Jain et al. [17] mentioned that the effect of the shape of the cross section on the hysteresis loops could be significant. Popov and Black [18] conducted cyclic experimental tests on 24 struts of different shapes (double channels, double angles, structural tees, wide flanges and square and round hollow structural steel sections (HSS)) and sizes: results indicated that of all the shapes, the HSS gave the best load-displacement hysteresis curve. Therefore, limited tests have been conducted to assess the behavior of wide-flange bracing members with slenderness ratios in the range of 40–175, and compactness in the range of 5–11.5. However, these tests (Popov and Black [18], Gugerli and Goel [19], Fell et al. [7], Tremblay [20], Richard [21], Powell [22], Clark [23], Hsiao [12]) have revealed that wide flanges can withstand larger deformation demands compared to the equivalent HSS tubes. High local strains develop in HSS bracing members because of the geometric nature of the local buckling at the mid-length plastic hinge. The local buckling leads to severe local rotation which in turn, creates high cyclic strain demand. Thus, brace fracture occurs at smaller inelastic deformations for HSS than for wide flange members when all other factors are the same.

Wide flange (WF) sections could be an attractive alternative to HSS sections as bracing members. The slight increase in cost per ton of columns and the low compressive resistance of WF braces could be justified if there was improved fracture life expectancy and better control in terms of expected strength compared to HSS tubes. In addition, WF braces could possibly be used in opposing pairs in frames. The high overstrength value of HSS bracing members is unfavorable and can lead to fracture in the connection if not taken into account, as seen in several earthquakes. Prediction of the fracture life of the HSS tubes is still a concern, despite four decades of cyclic testing and

---

E-mail address: madhar@uaeu.ac.ae.

http://dx.doi.org/10.1016/j.tws.2016.11.006
Received 28 December 2015; Received in revised form 24 October 2016; Accepted 10 November 2016

numerical modeling. That work however, has led to stringent limits on the width-to-thickness ratio as in the AISC 2010 seismic provisions [24].

Brace fracture prediction is a critical ingredient in the assessment of the seismic performance of braced steel frames, especially when verifying life safety and collapse prevention limit states. Fracture of the bracing members under seismic loadings usually results from ultra-low cycle fatigue. Fatigue may be defined as the fracture of the material at stresses below the maximum static strength under repeated stresses (Jastrzebski [25]). Accumulated plastic strain is the main cause of fracture from ultra-low cycle fatigue of the braces. Local buckling increases the accumulation of plastic strain that predisposes fracture (Haddad et al. [2]).

Given the lack of knowledge of the potential behavior of WF bracing members, a detailed finite element model capable of simulating the hysteresis behavior of WF bracing members subjected to cyclic loading protocols has been developed. The model utilizes a calibrated fracture model capable of predicting the fracture life of WF bracing members as tested. A sensitivity analysis is described in which the effects of mesh density, number of integration points, boundary conditions, initial imperfection, initial yield strength, and loading protocol on the hysteresis behavior of WF bracing members are studied. The model is then used to provide a better understanding of the modes of failure of WF bracing members. Finally, limitations in the seismic design codes are addressed concerning the effective slenderness ratio and width-to-thickness ratio of WF bracing members for both the near-fault and far-field loading protocols. Thus, the finite element model gives the opportunity to go well beyond experiment.

2. Specimens and loading protocols

The geometric properties of the WF specimens modeled herein are listed in Table 1. The specimen of Fell et al. [7] was designed without using a doubler plate connecting the web of the WF to the gusset plate. An access hole was sufficient to prevent the early failure of the connection. On the other hand, a doubler plate was used to connect the web to the gusset plate in the tests of Tremblay et al. [20] and Richard [21]: these tests were designed according to the weak-brace strong-gusset approach while verifying all possible failure modes. The results of design are shown in Appendix.

The three WF tests conducted by Fell et al. [7] were subjected to near-fault compressive or tensile and far-field cyclic loading protocols. The six tests conducted by Tremblay et al. [20] and Richard [21] were subjected to a far-field cyclic loading protocol that produced the median demand anticipated for moderately-ductile, braced steel frames with the largest inter-storey drift being 1.5% at cycle number 20. After the two smaller amplitude cycles, the loading protocol followed the demand expected in higher seismic zones similar to the far-field cyclic loading protocol of Fell et al. [7]. The loading protocols were developed in terms of inter-storey drift. For modeling, they were transformed into axial brace displacements assuming that all the deformations take place over the brace length between the end hinges. The cyclic loading protocols for the nine WF tests are shown in Fig. 1.

Table 1
Geometric properties of the WF braces.

Number	WF section, (mm)	KL/r	b_0/t	Gusset plate thickness, (mm)
W1[a]	360×134	40	10.3	45
W2	310×129	40	7.5	38
W3	310×129	60	9.9	38
W4	310×97	60	7.5	32
W5	310×86	60	7.8	32
W6	250×115	60	5.9	38
W14, W15, W16[b]	310×23.8	153	7.5	13

[a] Tremblay et al. [20] and Richard [21].

[b] Fell et al. [7], specimens are subjected to various loading protocols.

3. Finite element model description

In the finite element analyses, the Patran pre-processor [26], was used to create the nodes and to sketch the geometric boundaries of the model components. A separate attribute was created/assigned for each model component, specifically the brace flanges, the brace web, the k-area, the gusset and connection plates, and additionally the doubler plate if present. Neither the weld nor the access hole was modeled. The reduced integration quadrilateral shell element S4R, was used to mesh these components, which were connected at the appropriate nodes along their boundaries with the equivalence-mesh option. The connection plates and the doubler plate used in Tremblay's WF tests were connected to the flanges and the web of the WF brace using beam-type multi-point constraints. All elements were given the same nominal elastic modulus (200 GPa) and Poisson's ratio (0.3). Fixed boundary conditions were applied at the ends of the braces. The analysis deck option in Patran was used to create the input file. That file was adjusted manually to account for the material model and the applied steps of the cyclic displacements. A combined isotropic-kinematic hardening plastic material model with data type equal to a half cycle was defined for all elements of the model while specifying the corresponding true stress-strain curves for each component. Using data type equal to parameters resulted in high a computational time that was similar to applying the cyclic displacements in one step using the Abaqus explicit code. Geometric nonlinearity was also implemented in the current finite element model. Initial imperfection – first mode linear-static buckling analysis through the ramp-amplitude perturbation technique option – was necessary to allow for buckling of the brace. The input file was introduced to the Abaqus 6.11 [27] environment where the nonlinear equilibrium equations were solved in the stability based approach for each increment through the Newton-Raphson method. 1000 increments were implemented in each step. To study the effect of mesh density four mesh densities were used with element sizes equal to 0.5, 0.75, 1.0, and 1.5 the thickness of the flanges. In addition, the effects of the number of integration points, using simple or fixed end boundary conditions, and the size of the initial imperfection on the hysteresis behavior were investigated. The end connections are shown in Fig. 2(a), and (b) for specimens W6 of Tremblay et al. [20] and W15 of Fell et al. [7], respectively.

4. Material and fracture models

Wide flange members are manufactured by reheating and passing beam blanks through a series of rollers to form the beam in the precise geometric WF shape. As the shape cools down after hot rolling, a bow often develops. In addition, residual stresses are present in WF members due to uneven cooling: the tips of the flanges and web cool faster than the rest of the wide flange cross section. Tensile residual stresses are maximum at the k-area. The shapes are straightened by passing the section through a series of offset rollers that impose contact forces on the web, which plastically deform the section about the weak axis. The contact between the rollers and the web in the vicinity of the k-area may lead to a change in the mechanical properties at this location (FEMA-355A [28]). Coupons taken from the web-to-flange junction were considerably more affected by roller bending than coupons taken from other locations (Spoorenberg et al. [29]). The final cold straightening alters the properties of the steel in the k-area where extremely high yield stress occurs (Tide [30]). Material variations in the k-area may include a reduction in the ductility and toughness, increases in hardness, yield strength and ultimate strength, and an increase in the ratio of yield stress to ultimate strength. The true stress-strain curves for the flanges/web and k-area in addition to the plates