Engineering Structures 173 (2018) 28–51

Contents lists available at ScienceDirect

Engineering Structures

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

Finite element analysis on the seismic behavior of fully prefabricated steel frames

Hao Yin, Gang Shi*

Key Laboratory of Civil Engineering Safety and Durability of China Education Ministry, Department of Civil Engineering, Tsinghua University, Beijing 100084, PR China
Beijing Engineering Research Center of Steel and Concrete Composite Structures, Tsinghua University, Beijing 100084, PR China

ARTICLE INFO

Keywords:
Finite element analysis (FEA)
Seismic behavior
Fully prefabricated steel frame
Bolted end-plate joint
Flexible brace
Prefabricated slab

ABSTRACT

This paper presents a method of finite element (FE) modeling and analysis of the seismic behavior of fully prefabricated steel frames with end-plate joints, flexible braces and composite slabs. The main idea and objectives of this paper is to develop a finite element model with high accuracy, good stabilization and acceptable computational costs for the simulation of the cyclic behavior of multi-story steel frames with bolted end-plate joints and concrete slabs. Because of the apparent tension-compression asymmetry of flexible braces, and the complex connection details between slabs and steel structures, as well as the huge number of contact interactions between interfaces, the cyclic behavior of this type of frame cannot be simulated accurately with commonly used line-element models, shell element models, or multi-scale models. A quasi-static test of a full-scale three-story fully prefabricated steel frame under cyclic horizontal loads by the present authors was simulated with the finite element model. Hollow section box columns, I-section beams, end-plates, inner diaphragms and stiffeners were modeled using shell elements; high strength bolts and concrete slab were modeled with solid elements; and flexible braces and rebar were modeled with truss elements. In order to develop a mesh skill to reduce computational costs while ensuring calculation accuracy, several FEM models were built and validated against previous experimental studies: static testing of bolted T-stub connections and bolted tension splices, static and cyclic testing of bolted end-plate steel joints, push-out tests of stud shear connectors, as well as static and cyclic testing of bolted end-plate composite joints. To simulate the "elastic-yield-hardening in tension, and buckling-without capacity in compression" behavior of the flexible braces, a simplified model in ABAQUS based on truss elements was developed and validated against previous tests. Results showed that the proposed FE modeling method could accurately simulate the static and cyclic performance of bolted T-stub connections, bolted tension splices, bolted end-plate steel joints, stud shear connectors, bolted end-plate composite joints and flexibly braced steel frames. Deformation capacity, cyclic behavior, horizontal loading performance, energy dissipation and stiffness degradation of steel frames with bolted end-plate joints, prefabricated slabs and flexible braces could be accurately simulated by this FEM model, providing a practical and accurate modeling method for similar structures. In addition, further research on the structural seismic performance simulation, parametric study and seismic design method could be carried out using the finite element model developed in this paper.

1. Introduction

Prefabricated buildings whose members are fabricated in factories and assembled together on site have relatively better construction quality and faster construction speed. High performance connections are the guarantee of the seismic behavior of the pre-engineered buildings. In order to develop a new type of fully prefabricated multi-story steel building with high seismic performance, the present authors designed and tested a full-scale three-story frame with cold-formed hollow box section columns, I-section beams, bolted extended end-plate joints, flexible braces and prefabricated slabs [1,2]. The test layout and the details of a joint are shown in Fig. 1. All the structural members were prefabricated in the factory and assembled on site: beams and columns were bolt connected; slabs were connected with the frame using stud shear connectors; and flexible braces were connected with the frame using fillet welds. Because little concrete casting and field welding were involved during the structural assembly process, construction efficiency was improved effectively. More specific details of this new type of prefabricated steel frame were introduced in paper [1,2].

Many tests of steel or composite frames have been conducted and

* Corresponding author at: Key Laboratory of Civil Engineering Safety and Durability of China Education Ministry, Department of Civil Engineering, Tsinghua University, Beijing 100084, PR China.
E-mail address: shigang@tsinghua.edu.cn (G. Shi).

https://doi.org/10.1016/j.engstruct.2018.06.096
Received 25 January 2018; Received in revised form 10 May 2018; Accepted 26 June 2018
0141-0296/

Nomenclature

μ	Poisson's ratio
Δ_y	yield displacement
Δ	brace elongation
E_s	Young's modulus of steel
σ	nominal stress
ε	nominal strain
σ_{eq}	equivalent stress
ε_{eq}	equivalent strain
σ_r	true stress
ε_r	true strain
σ_y	yield stress
ε_y	yield strain
l_R	the original length of brace
l_b	the length of the strut member
$D_{x\text{-}E1}$	the displacement in X direction of reference point RP-E1
$D_{x\text{-}E2}$	the displacement in X direction of reference point RP-E2
$D_{x\text{-}E3}$	the displacement in X direction of reference point RP-E3
$D_{x\text{-}E0}$	the displacement in X direction of hypothetical point RP-E0
$D_{x\text{-}W1}$	the displacement in X direction of reference point RP-W1
$D_{x\text{-}W2}$	the displacement in X direction of reference point RP-W2
$D_{x\text{-}W3}$	the displacement in X direction of reference point RP-W3
$D_{x\text{-}W0}$	the displacement in X direction of hypothetical point RP-W0

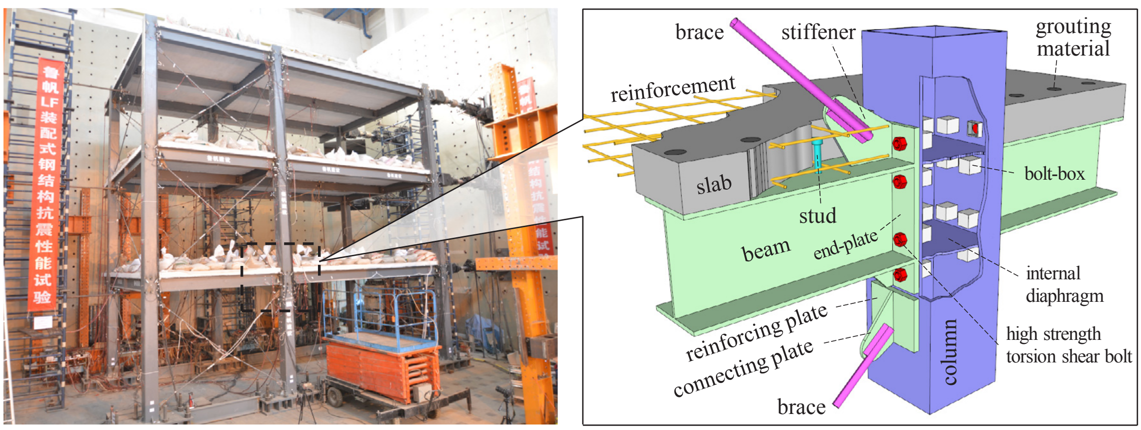

Fig. 1. Layout of the test and details of a joint.

their results reported over the years and such tests are becoming increasingly comprehensive, profound, and complex [3–9]. However, due to the huge costs and operational workload of such tests, the number of specimens in each test has been very limited. Thus, the FE method, which may be regarded as the extension or supplement of such testing, can play a very important role in the study of structural seismic behavior and design methods.

Three main types of model have been used for nonlinear simulation of steel and composite frames: macromodels (line-element models), multi-scale models, and micromodels (continuum models). As summarized by Spacone [10], the macromodels that use line elements and spring connectors had been fundamentally matured before 2004: models for columns, composite beams and joints were developed and combined to investigate the global behavior of composite frames. In the following years, macromodels were further developed and used in test simulation, parametric study and engineering design by Zhou et al. [11], Faella et al. [12], Liu et al. [13], Iu et al. [14], Chellini et al. [15] and Nie et al. [16]. Although computational cost could be saved by using macromodels, local geometric nonlinearities such as local buckling of steel components, plastic deformation in end-plate connections and shearing rotations at panel zones were neglected in the frame simulation. Parameter calibration that required much test data also made a model of spring connectors hard to obtain.

The multi-scale models, in which shell elements or solid elements are used to simulate joints and member ends and line elements are used to simulate the middle part of members, have been widely used in recent years to address the aforementioned problems. Khandelwal et al. [17], Helldörfer et al. [18], Yu et al. [19], Wang et al. [20] and Tao et al. [21] conducted comprehensive research into the simulation of steel frames or composite frames using multi-scale models.

However, with the continuous improvement of software functions and exponentially increasing computing power, micromodels in which shell elements or solid elements were used could be adopted in the simulation of the seismic behavior of an entire framework. Besides, the number of elements could be reduced by adjusting mesh density in the middle part of members [22–24] and complex mixed-dimensional coupling problems [17] could be avoided. Bursi et al. [25] established three-dimensional FEM models of the substructures of composite frames with ABAQUS, and calibrated a one-dimensional model conceived with Drain-3DX relying on beam-column elements, for the simulation of overall frames. Han et al. [26] proposed a FE modeling method in ABAQUS to simulate the seismic performance of a composite frame with concrete-filled steel tubular (CFST) columns and steel beams, in which shell elements and solid elements were used to model steel components and in-filled concrete respectively. Guo et al. [27] developed a FEM model in ABAQUS to analyze the monotonic behavior of a bolted end-plate composite frame, in which solid elements were used to simulate slabs, bolts, end-plates and column flanges, shell elements were used in other steel components, and truss elements were used to model the reinforcement bar. Wang et al. [28] simulated the seismic performance of semi-rigid CFST frames with composite wall panels, and fine meshed solid elements were used in all the steel and concrete components. So far, no further micromodel numerical studies of the performance of composite frames have been reported. Furthermore, the FEM models in Refs. [26,28] could only be monotonically loaded, and the cyclic behavior of steel frames with concrete slabs could not be