Journal of Constructional Steel Research 82 (2013) 164–176

Contents lists available at SciVerse ScienceDirect

Journal of Constructional Steel Research

Steel–concrete composite flange plate connections — finite element modeling and parametric studies

M.S. Smitha, S.R. Satish Kumar *

Department of Civil Engineering, IIT Madras, Chennai 600 036, India

A R T I C L E I N F O

Article history:
Received 28 January 2011
Accepted 13 January 2013
Available online 13 February 2013

Keywords:
Steel–concrete composite connections
Finite element analysis
Semi-rigid connections
Moment–rotation curve
Partial interaction
Bilinear model

A B S T R A C T

Composite connections are defined as the regions where composite beam frames into a steel column. A composite flange plate connection with web cleats is studied here since it is expected to satisfy the seismic performance requirements at a reasonable cost. A modification of the traditional flange plate connection, termed as stiffened flange plate connection, is also studied and its merits over the flange plate connection are assessed. The flange plate and stiffened flange plate connections are analyzed by finite element method and the validity of the model developed established by comparing with the envelope of the cyclic moment–rotation curve and the failure mode obtained from tests. Parametric studies were done to understand the connection parameters influencing the moment–rotation behavior of the connections under hogging and sagging moments. Based on the results of the parametric studies, mathematical equations have been developed for predicting the initial stiffness, moment capacity and ultimate rotation under hogging and sagging moments, and thus the entire moment–rotation curve. A simplified bilinear model is also developed for use in semi-rigid frame analysis and dynamic analysis.

1. Introduction

The behavior of semi-rigid connections is represented in terms of their moment–rotation relationship characterized by the initial stiffness, ultimate moment capacity and rotation capacity. Different approaches can be used for predicting connection rotational behavior including empirical models, analytical models, mechanical models, finite element models and experimental testing [1]. Most accurate knowledge of the joint M–θ curve can be obtained through experiments. Data bases describing the connection moment rotation curves in terms of test details, connection geometry etc. are available [2], but the designer has a low probability of finding in the data bank the exact joint detail he is looking for because of the great variety in connection topologies, geometric properties and panel zone details. Tests are difficult, expensive and time consuming; moreover it is impossible to do tests for all combinations of dimensions. Hence, analytical methods duly calibrated with test results, can be used as a tool to predict the behavior of connection for the different parameters which influence the moment–rotation characteristics.

There are three main analytical approaches for modeling semi-rigid behavior [3]. The first is the simplified approach where a single rotational spring is used to represent the moment–rotation relationship of the connection and requires a data base of analytical or experimental results to calibrate the spring properties. The second approach is the mechanical or component approach where a number of springs are used to capture the material ad geometric parameters based on the information available from tests on the behavior of similar components. The method is most suitable when trying to predict the behavior of connections with components such as end plates or bolts whose behavior is well understood. However, when a connection involves a variety of components whose behavior is yet to be ascertained with an adequate degree of certainty, the only options available are testing and a detailed finite element analysis.

Unlike the case of pure steel semi-rigid connections, the first method has not been developed for steel–concrete composite connections due to the limited test data available. Limited attempts have been made in the literature to model the behavior of certain types of steel–concrete composite connections by the component method with varying degrees of success.

Liew et al. [4] attempted to predict the assessment of the moment-capacity and initial rotational stiffness of end-pated and haunched connections using the information available in Eurocode 3 [5] and Eurocode 4 [6]. They also extended the method to predict the moment rotation behavior under positive bending moment. They concluded that the Eurocode model slightly over-predicts the initial rotational stiffness for joint loaded symmetrically but for joints subjected to opposite moments, the prediction is better only if the controlling factor was the shear deformation of the panel zone. However, only a bi-linear elastic-perfectly-plastic idealization was adopted for the analytical moment–rotation curves as proposed by Eurocode 3 and so non-linear nature of the curve could not be predicted. They have also reviewed the previous attempts

* Corresponding author. Tel.: +91 44 2257 4287; fax: +91 44 2257 6287.
E-mail address: kim@iitm.ac.in (S.R.S. Kumar).

0143-974X/$ – see front matter
http://dx.doi.org/10.1016/j.jcsr.2013.01.002

by various researchers to model the connection behavior by finite element analysis.

Rassati et al. [3] have also applied the component method to predict the monotonic and cyclic response of seat angle type composite connections. Their model takes into account the influence of all the main deformation components, including slip in the bolts, partial interaction between the concrete slab and steel girder, shear deformation of the panel zone, and cracking and crushing of the slab. They have also extended the method to predict the cyclic response. However, using the simpler model, they also obtained essentially a bilinear elastic-perfectly-plastic approximation to the essentially non-linear test results. In an attempt to improve the predictions they have developed a model with several springs and approaches in complexity to the finite element models. Finally, they have concluded that the model can be used for parametric studies to develop design guidelines.

Extensive research has been done on the numerical modeling of steel–concrete composite members. The existing works on the nonlinear behavior of composite members can be grouped into the following two categories: 1) 1-D beam elements that capture the salient features of the nonlinear behavior of composite girders within the framework of Navier–Bernoulli beam theory and 2) finite-element models utilizing beam, plate, shell, or brick finite elements to represent in great detail the constituents of the composite structural element.

In the first category, Wegmuller and Amer [7] in their study on composite bridge girders used layered plate and beam elements to model slab and steel section respectively. Studs were not modeled; instead compatibility of the displacements between beam and slab was enforced. Hirst and Yeo [8] used a layered beam-plate element in order to study partially-connected composite beams. Quadrilateral elements were used to model stud connectors and their material properties were modified to make them equivalent in strength and stiffness to actual stud connection. A three-dimensional bar element has been formulated by Razaqpur and Nofal [9] for modeling the shear connectors in composite beams. The stiffness properties of the bar element were defined by a shear–slip relationship obtained from the experimental data. In view of the high computational expense of 2D and 3D models, Arizumi and Hamada [10] and Daniels and Crisinel [11] proposed the displacement-based beam elements which can be formulated easily. Sebastian and McConnel [12] presented a finite element program for the nonlinear analysis of continuous composite beams, using a two-dimensional (2D) and a one-dimensional (1D) layered bending membrane element for the concrete slab and the steel beam, respectively, and a stub element for the shear connector. Salari et al. [13] developed a force-based finite element capable of modeling softening due to the crushing of slab and failure of shear connectors, while Ayoub and Filippou [14] formulated a two-field mixed element; both of them exhibit a superior performance in the non-linear case.

Finite element modeling of steel–concrete composite members is complex due to the presence of reinforced concrete and also due to the presence of deformable shear connection between the concrete slab and the steel beam section. Kattner and Crisinel [15] presented a two dimensional FE model for composite joints in which both the slab and steel sections are represented by beam elements. Translational spring elements represent the shear connectors and the connection between the concrete slab and column flange. Various material models and element types available in the finite element software ABAQUS [16], were tried by Baskar et al. [17] in their study on composite plate girders. Liang et al. [18] have also used the ABAQUS software to study the behavior of continuous composite beams in bending and shear. Shell elements were used to model slab and steel section and stud connectors were modeled by 3D beam elements. Fu et al. [19,20] have undertaken parametric studies of composite connection with precast hollow core slabs. The concrete slab, steel sections and studs were modeled with solid elements using ABAQUS. They have modeled concrete under tension using the elastic–plastic material property as explained in more detail later in this paper.

Since the composite connection involves several such factors such as shear lag in the slab, slip at the steel–concrete interface, bearing of concrete slab against the steel column and consequent crushing of the concrete and possibility of brittle failure at the welds, it was decided to use a simple finite element model to capture accurately the moment–rotation behavior and thereby generate information which can be used to develop both the simpler model with a single non-linear rotational spring as well as the component method. Further, the parametric study sheds light on the effects of the various geometric parameters on the connection response parameters.

In this paper, the issues related to the finite element analysis of composite connections are deliberated and a balance between the complex modeling vis-à-vis the models giving speed and accuracy, is arrived at. The model is then used to study the effect of various connection parameters on the moment–rotation curve in both sagging and hogging directions. Equations are presented which will enable the designer to choose suitable connection parameters and to model the connection behavior as semi-rigid. For seismic response analysis, an empirical equation to evaluate the rotation capacity of the connection and a suitable bilinear model are presented.

2. Finite element analysis of flange plate connections

Modeling the behavior of composite joints beyond the peak strength requires robust material models capable of simulating the material nonlinearity and other damaging effects. In the present study, a three dimensional finite element model, which accounts for material and geometric nonlinearity, is developed to simulate the moment–rotation envelope curves obtained from cyclic tests on flange plate and stiffened flange plate connections using the FEA software ABAQUS [16]. The finite element model developed is validated with experimental results and it is used for further parametric studies. Since both the flange plate and stiffened flange plate sub-assemblages are symmetric with respect to a longitudinal plane passing through the web of the steel beam section, only one half of the connection was modeled. Symmetry boundary conditions were applied to all nodes lying in the plane of symmetry.

2.1. Elements used for modeling

The steel sections and concrete slab were modeled using 4 noded reduced integration shell elements (S4R). The reinforcing bars are assumed as a smeared layer of constant thickness within the shell element representing concrete slab. Welds and bolts were modeled using CONN3D2 elements since the HSFG bolts were designed for no slip at ultimate. Finer mesh was used in the joint region to capture the effect of stress concentration. The finite element model developed for the specimens FP and SFP is shown in Fig. 1(a) and (b). More details on the model adopted can be obtained from [21].

2.2. Modeling of partial interaction

The presence of deformable shear studs connecting the concrete slab and steel section causes slip at the interface of the slab and steel section leading to partial interaction. This is modeled by using nonlinear spring elements (SPRING2) between the slab and the steel section. The 'OFFSET' option is used to define the nodes in concrete slab at its bottom surface. The 'SPRING2' elements were used to connect the midsurface of the steel section to bottom surface of slab. Three spring elements are provided at each discrete stud location. The first spring element is in the longitudinal direction and is used to simulate the slip at the interface due to stud deformation. The following force