Engineering Structures 152 (2017) 698–708

Contents lists available at ScienceDirect

Engineering Structures

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

General component based cruciform finite elements to model 2D steel joints with beams of equal and different depths

Eduardo Bayo [a,*], Alfonso Loureiro [b], Manuel Lopez [b], Luís Simões da Silva [c]

[a] University of Navarra, Spain
[b] University of A Coruña, Spain
[c] ISISE, Dep. Civil Engineering, University of Coimbra, Portugal

ARTICLE INFO

Article history:
Received 24 April 2017
Revised 20 September 2017
Accepted 21 September 2017

Keywords:
Steel joints
Cruciform finite element
Semi-rigid connections
Structural analysis
Modelling

ABSTRACT

This paper deals with the formulation of cruciform finite elements that model 2D steel joints for frame analysis. Three different cases are contemplated: single rectangular panels suitable for internal joints with beams of equal depth at both sides; trapezoidal panels for joints with beams of different depths at both sides and inclined stiffeners; and double rectangular panels for the case of joints with beams of different depths at both sides and with or without horizontal stiffeners in the panel zone.

The cruciform elements have 4 nodes with 3 degrees of freedom per node, and take into consideration, in a consistent and complete way, the stiffness properties (*components in Eurocode*), including those of the panel zone. The proposed elements allow modelling simple, rigid and semi-rigid connections. In addition, they take into account all the internal forces that coincide at the joint, and the eccentricities of the internal forces coming from the beams and columns that meet at the joint. Numerical examples are solved that compare the proposed approach with complete finite element models of sample frames. The results validate the approach and demonstrate its advantages.

1. Introduction

Modern structural steel codes [1–4] include procedures and formulations to define both the stiffness and resistance of the joints (simple, rigid and semi-rigid) so they may be used for the global analysis and design of the structure. However, semi-rigid design has not received widespread attention due to its perceived complexity and the lack of effective tools for frame analysis.

The most common approach to modelling connections in structural analysis programs is by means of zero length springs attached to the end of the beams at both sides of the joint. In the USA, these springs are characterised by different models such as the Frye-Morris polynomial model [5], the modified exponential model [6] or the three-parameter power model [7], among others. These models accurately represent the characteristics of the connections at both sides of the joint based on empirical statistical calibrations with experimental test results, but do not take into account the panel deformations due to shear and bending.

Early experimental studies [8] and more recent research [9–11] have stressed the need for a correct definition of the panel zone deformations under static conditions due to its influence on the overall behaviour of the frame. An increase in frame drift due to panel-zone shear deformation may render the frame unserviceable and also affect its global stability. Modelling of the panel is also important for the avoidance of local failure of the columns under ultimate limit state conditions. Eurocode 3 (EC3) in its part 1.8 [3] takes into account the deformation of the panel zone as one of the components of the joint. Furthermore, EC3 provides the so-called *component method*, that allows determining strength and stiffness characteristics of all the different parts of the joint, including the panel zone. Under EC3 the different components (springs) are assembled to form a resultant elastic-plastic *spring element* that models the connections at each side of the joint to be attached at the end of each beam.

Frame analysis is performed in computer programs using these spring elements attached at the end of the beams to simulate the behaviour of the connections [12]. Dhillop and O'Malley [13] used semi-rigid generalised beam elements in the context of the LRFD format for the interactive design of semi-rigid frames. Other zero length semi-rigid spring elements have been proposed for analysis of steel [14,15] and composite [16] frames.

The zero length end of beam spring model does not account for either the real dimension of the joint or the interaction of all the forces that intervene in the deformations of the panel zone.

* Corresponding author.
E-mail addresses: ebayo@unav.es (E. Bayo), loureiro@udc.es (A. Loureiro), manuellopez@udc.es (M. Lopez), luisss@dec.uc.pt (L. Simões da Silva).

https://doi.org/10.1016/j.engstruct.2017.09.042
0141-0296/

Consequently, EC3 introduces a transformation parameter, called β that depends on the internal moments and shear forces acting on the panel, and which affects both the stiffness and resistance of the components (springs). Since these internal forces and moments are not known in advance, the frame and joint analyses require an iterative process [17].

A complete description of joint mechanical models is presented in [18]. These models are composed of rigid bars and elastic-plastic springs that are attached to the beams and columns for frame analysis. Although they consider the shear deformation of the panel and the properties of the left and right connections they do not include the axial and bending deformations of the column. In addition, they require a significant amount of work from the user to define their properties and dimensions. Furthermore, the mechanical models increase considerably the number of bars and degrees of freedom of the frames, and may cause numerical instability in the analysis due to the high stiffness of the rigid bars.

Recently, a combination of the finite element method and the component method has been proposed for the characterization of structural connections [19]. These models take into account in an accurate manner the material properties, shear deformation, extension and bending of the complete joint. However, they become computationally expensive, particularly in the case of non-linear analysis, and require special features to accommodate the resulting joint properties with the rest of the structural elements, beams and columns, that compose the frame. Also recently, a component-based nodal finite element was also proposed that was implemented in OpenSEES [20].

In this paper, component based cruciform 4-node joint finite elements are introduced that take into account the actual size of the joint, its deformation characteristics (*components*), including those of the panel zone, local phenomena and all the internal forces that concur at the joint. As a consequence, these elements avoid the use of the transformation parameter, and the inherent iterative process that it requires. Since the elements are attached to the adjacent beams and columns, the eccentricities of the internal forces coming from them are also taken into account.

The first cruciform element for single rectangular panels was proposed in [17] and it was based on a flexibility method. Later, a stiffness approach [21] was used for composite joints, as well as steel joints with trapezoidal shear panels [22]. More recently a different displacement based cruciform element has been introduced for steel joints with beams of different depths and stiffened double rectangular panels [23]. A more general and efficient stiffness approach is proposed herein that relies on the superposition of displacement modes and constraint conditions. This method is applicable to the three possible 2D joint cases: rectangular, trapezoidal and double rectangular panels, respectively. The formulation of the new element for single rectangular joint panels (which corresponds to the case of internal joints with beams of equal depth at both sides) is derived in Section 2. The formulation for the case of trapezoidal panels (for joints with beams of different depths at both sides and inclined stiffeners) is introduced in Section 3, and the case of two rectangular panels (joints with beams of unequal depths at both sides and with or without horizontal stiffeners in the panel zone) is presented in Section 4. Section 5 contains numerical simulations that compare the proposed formulation with complete finite element models of the frames. These results validate the proposed approaches and set the stage for the conclusions that follow in Section 6.

2. Formulation for single rectangular panels

A standard mechanical model that represents the behaviour of a single rectangular panel joint, which corresponds to beams of equal depth at both sides, is depicted in Fig. 1. This mechanical model is composed of springs and rigid bars. The height of the joint is represented by h and the width by h_c; the sub indexes 1 and 2 signify the right and left sides of the joint, respectively. Also, the sub indexes t and b stand for the top and bottom sides of the joint. The symbols K and Δ stand for the spring stiffness and displacements, respectively. The values of the panel rotational spring K_p as well as the side springs K_{1t}, K_{2t}, K_{1b} and K_{2b}, and their respective resistances are defined in Chapter 6 of EC3-Part 1.8 [3] for bolted and welded joints. These values will determine whether the joint is simple, rigid or semi-rigid. The variable ϕ_p represents the rotation of the panel. The right and left rotations ϕ_1 and ϕ_2 are shared with the right and left beams, respectively; and the same applies to the moments M_1 and M_2. Similarly, the displacements δ_t and δ_b, and corresponding shear forces V_t and V_b are shared with the top and bottom columns.

The mechanical model can be directly assembled to the adjacent beams and columns and used in a frame analysis. However, since the bars are rigid, the axial and bending deformations of the column are not included in this model. The axial deformation is generally small, however, as reported in [24], the bending part cannot be disregarded. In addition, as mentioned above, the direct use of this mechanical model increases considerably the number of bars and degrees of freedom of the frame model, and may cause numerical instability and round-off errors during the analysis due to the high stiffness of the bars.

These problems can be avoided by using the cruciform element shown in Fig. 2 that will be formulated so that it will contain, in addition to the deformation characteristics of the mechanical model, the axial and bending properties of the column. Furthermore, this type of element avoids the use of rigid bars. The element has 4 nodes (A, B, C and D) with three degrees of freedom (dof) per node (see Fig. 2) represented by a vector **r** with elements r_1 to r_{12}. The cruciform element has the same dimensions as the mechanical model and will share the moments and forces (represented by the vector **F**) with the contiguous elements (beams and columns) and the corresponding nodal points.

In order to formulate the stiffness of the element a superposition of deformation modes is followed. For this purpose, the degrees of freedom of the element are partitioned in three groups as shown in Fig. 3. The first group (see Fig. 3a) considers the panel-column displacements and rotations under vertical and shear forces as well as bending deformations. Its behaviour can be modelled as a beam-column element that incorporates the axial, shear and bending deformations [25]. The parameters needed for defining the corresponding stiffness matrix are:

- The column-panel height h
- The column area A_c
- The column moment of inertia I_c
- The column shear area A_v which can be obtained as follows

$$A_v = K_p/(Gh) \tag{1}$$

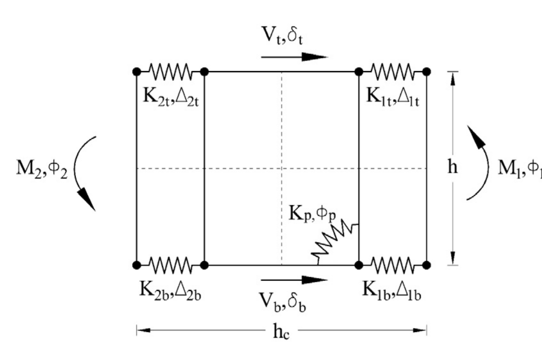

Fig. 1. Mechanical model for single rectangular panels.