Available online at www.sciencedirect.com
ScienceDirect
ELSEVIER
Engineering Structures 30 (2008) 232–246
ENGINEERING STRUCTURES
www.elsevier.com/locate/engstruct

An alternative design for internal and external semi-rigid composite joints. Part II: Finite element modelling and analytical study

Beatriz Gil, Eduardo Bayo*

Department of Structural Analysis and Design, School of Architecture, University of Navarra, 31080 Pamplona, Spain

Received 1 September 2006; received in revised form 6 March 2007; accepted 8 March 2007
Available online 3 May 2007

Abstract

Tests on semi-rigid composite joints with the alternative configuration proposed in Part I of this paper are simulated by using the finite element method. The finite element models are adjusted, compared and validated with the experimental results until reliable and robust models are achieved. These models are used to compare the performance of the conventional joints with the alternative design that has been proposed. The internal joints with the proposed design show improved resistance and stiffness, more acutely as the moments on either side of the joint become increasingly unbalanced. On external joints, the same values are obtained as with the conventional design, but without the need for a cantilever. A parametric study is also carried out on external and internal joints following the alternative design in order to define the components which intervene in it as well as to develop the corresponding analytical model.

Keywords: Semi-rigid composite joints; Analytical model; Component method; Finite element modelling

1. Introduction

The finite element method (FEM) acts as a link between the experimental tests and the mechanical and analytical modelling, permitting better understanding of the experimental behaviour and the simplified methods. Due to the amount of effort that it requires, it is not useful for the practical design of composite joints, except when it is not possible to use simplified methods.

Some authors (SSEDTA [1]) consider that the FEM is inappropriate to model a composite joint, as it does not consider certain local phenomena such as the interaction between concrete and steel components, cracking of concrete, creep, etc. In other papers such as the one by Ahmed et al. [2], the contribution of concrete in finite element modelling is ignored, with only the shear connectors transmitting the tensile stress to the reinforcement. However, the FEM is currently considered to be an appropriate tool which is used by numerous researchers in order to find out more about the behaviour of beams and composite joints ([3–8], among others). Research also continues on improving the modelling and implementation of the components [8], materials [9], and local phenomena [10] by means of finite elements. Particular difficulty was found in characterising the interaction between the concrete slab and the steel profile, as well as the behaviour of the reinforced concrete. These publications describe different finite element modelling techniques to simulate composite joints. However, there is a lack of information regarding the compatibility of the interfaces when different types of elements are adopted.

This study is carried out with the Abaqus v.6.5 finite element program [11] and involves simulating the composite flush end plate joints between composite beams with full shear connection and steel columns with the proposed alternative configuration which consists of passing central reinforcement bars through the column flanges. The geometry is the same as for the tests which are carried out and described in Part I of this paper, in order to calibrate and validate the resulting finite element model. In this way, once the model has been corroborated, a parametric study is carried out which shows the influence of each parameter on the overall behaviour of the joints. Finally, the opportune modifications are established for the component method which provides us with the design for this type of joints. These same joints are also modelled in their conventional design, meaning without

* Corresponding author. Tel.: +34 948 425731; fax: +34 948 425629.
E-mail addresses: bgilr@unav.es (B. Gil), ebayo@unav.es (E. Bayo).

0141-0296/$ - see front matter © 2007 Elsevier Ltd. All rights reserved.
doi:10.1016/j.engstruct.2007.03.010

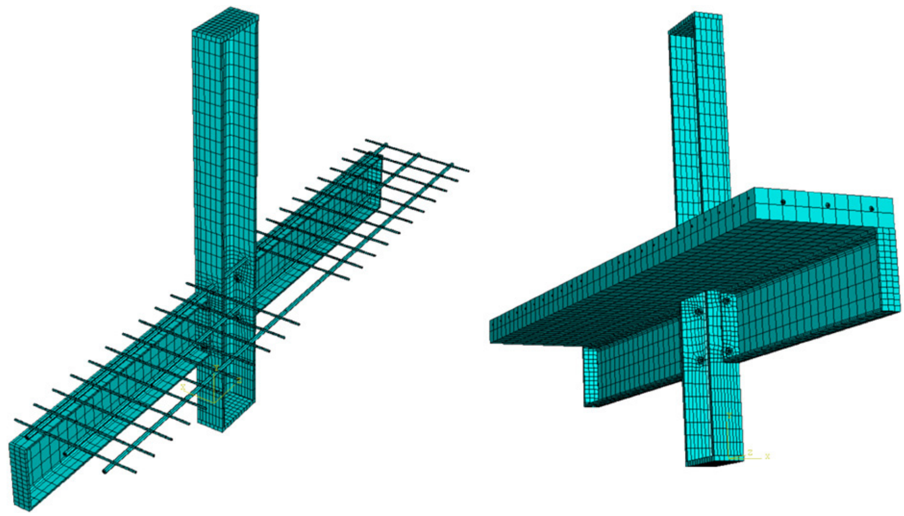
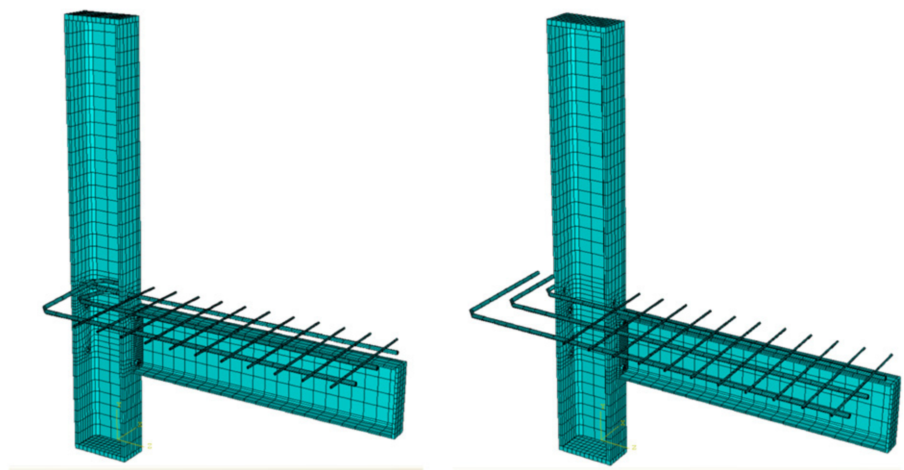


Fig. 1. Mesh from the internal joint model with through-flange reinforcement.



External semi-rigid joint with through-flange reinforcement

Conventional semi-rigid external joint with cantilever

Fig. 2. Mesh from the external joint model with through-flange reinforcement and from the conventional design.

through-flange reinforcement, to be able to compare the appreciable differences in their behaviour.

2. Finite element model

2.1. Geometry, definition of the elements and mesh

The three-dimensional finite element model has the configuration and dimensions of the tested specimens. With the objective of achieving better computational efficiency, half joints have been modelled using symmetry conditions for both internal and external joints.

The concrete slab is modelled with 8-node solid elements (C3D8). The same element is chosen for the slab reinforcement, as the 2-node beam elements which were initially chosen (B31) did not capture the stress generated in the rebars very well. The reinforcement is embedded in the solid element which represents the slab. Abaqus presents the option of modelling the slab as a shell element and the reinforcement as distributed or spread over this shell using the *rebar layers* option, meaning, the reinforcement would be another shell with a cross-section equivalent to the steel rebars it represents. This method is used by a large number of authors, such as Amadio and Fragiacomo [5] and Bursi et al. [3]. However, this mode does not allow us to model changes in the spacing and geometry of the rebars. This is the reason why the solid element was chosen.

For the steel beams and the column, we have opted for 8-node solid elements (C3D8) like those in the slab, to avoid the conflicts which are occasionally generated in Abaqus involving interactions between different types of elements. This tends to give more accurate results, although it implies greater computational cost. The shear connectors have been modelled with nonlinear springs with normal and tangential stiffness located in the beam–slab interface in its real position. The bolts have been modelled with solid elements (C3D8) with the same bolt cross-section and with head and nut.

Figs. 1 and 2 show the model mesh for the different types of simulated joints, meaning internal joints with conventional