Engineering Structures 165 (2018) 11–26

Contents lists available at ScienceDirect

Engineering Structures

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

Plasticization conditions and strength of a welded beam-to-column connection in case of various detailing

Todor Vacev*, Stepa Paunović, Ivan Nešović, Andrija Zorić, Miloš Milić

Faculty of Civil Engineering and Architecture, University of Niš, Aleksandra Medvedeva 14, Niš, Serbia

ARTICLE INFO

Keywords:
Steel structures
Beam-to-column connection
Welding
Stress concentrations
Plasticization
FEA

ABSTRACT

Beams and columns in steel structures are regularly connected using bolted or welded connections. In this paper, one typical welded connection was analysed using Eurocode standard methodology and advanced finite element (FE) analysis in materially nonlinear domain with fully 3D modelling of the fillet weld. The FE analysis revealed stress concentrations and plasticization regions, which are out of reach in Eurocode methods. Those phenomena were the main source of disagreement in weld strength obtained by the two methods. Since the classical solutions admit existence of the stress concentrations and plasticization, but do not take them into calculation, an attempt was made to investigate them and to introduce them into the design process, proposing a controlled degree of plasticization. Such methodology is not part of established codes, and it was not given here as a solution ready-to-use, but as an open question for a broader research, pointing out at the phenomena that are neglected by classical design methods. Besides that, the subject of research was the influence of the column flange thickness on connection strength, as well as different construction detailing, including some non-standard solutions.

1. Introduction

Welded connections are, generally speaking, studied a lot using analytical, numerical, and experimental methods. Complexity of those methods varies, and some of them are strictly limited to scientific purposes, while, on the other hand, some are adjusted for daily engineering practice. Results synthesized from analytical and experimental investigations resulted with established procedures implemented in many national or international building codes, e.g., Eurocode. Thereat, for the sake of simplicity, many phenomena that accompany behaviour of the welded connections are omitted or taken with approximation. This stands in particular for stress concentrations and possible occurring of plasticization in the weld, or in its vicinity, i.e., in the base material.

Classical, i.e., analytical solution of the fillet weld connections assumes calculation of the stress components in the plane of its throat. According to Eurocode [1] design resistance of the fillet weld is:

$$[\sigma_\perp^2 + 3(\tau_\perp^2 + \tau_\parallel^2)]^{0.5} \leqslant f_u/(\beta_w \gamma_{M2}) \text{ and } \sigma_\perp \leqslant 0.9 f_u/\gamma_{M2} \quad (1)$$

where parameters f_u, β_w, and γ_{M2} relate to the characteristics of the material and safety factors of the Code. The stress symbols denote as follows: $\sigma_\perp$ is the normal stress perpendicular to the throat, $\tau_\parallel$ is the shear stress (in the plane of the throat) parallel to the axis of the weld, and $\tau_\perp$ is the shear stress (in the plane of the throat) perpendicular to the axis of the weld [1]. Reducing the volume of the weld to the throat plane deprives the design process of the spatial distribution of the stresses, which is inevitably present. Representation of such distribution is, however, possible using numerical modelling and analysis based on the Finite Element Method (FEM) and application of advanced engineering software. Advanced modelling enables completely 3D simulation of the weld and observing all phenomena inside or on its surface. Of course, numerical methods carry certain limitations and drawbacks that one must be aware of.

However, although welded beam-to-column connections are vital, but at the same time, sensitive parts of structures, they are not studied in literature sufficiently, to the best of our knowledge. There are many papers on bolted beam-to-column connections, but rather few of them treat the welded connections, and not all of them use the possibilities of FEM analysis fully [2–5]. Moreover, even when these connections are analysed, this is rarely done from the viewpoint of the ultimate limit state design criteria, but rather from the viewpoint of fatigue and cyclic loading [6], or from the residual stresses due to the welding process and their prediction using the FEM analysis [7,8]. The few papers related to ultimate limit state analysis of a welded connection mostly do not deal with welds in some particular real structure, but with some "general purpose" welds instead, such as general T-element connection [9], or

* Corresponding author at: Faculty of Civil Engineering and Architecture, University of Niš, A. Medvedeva 14, 18000 Niš, Serbia.
E-mail address: todor.vacev@gaf.ni.ac.rs (T. Vacev).

https://doi.org/10.1016/j.engstruct.2018.03.016
Received 18 August 2017; Received in revised form 7 March 2018; Accepted 9 March 2018

longitudinal weld in some tension connections [10,11]. In addition, in the most of the papers that do concern the welded beam-to-column connections, only the relatively small part of the structure surrounding the analysed connection is modelled [12,13]. In this way, the global effects such as the deformation of the connected structural elements and the effects of the remote loads are not adequately accounted for. The FEM is very convenient and maybe the only method accurate enough for investigating this class of problems beside experimental methods. In spite of that, the FEM has not been used much for analysing welded beam-to-column connections. At the same time, the analytical design procedures proposed in some structural codes do not take into account some very important phenomena such are stress concentrations.

However, there are a few papers attempting to analyse beam-to-column welded connections more thoroughly [14,15], but they do not explicitly address the issue of stress concentrations at the toe of the weld.

For all these reasons, it was decided to conduct a full 3D FEM analysis of one real case of steel structure connection, modelling in detail the structural elements, as well as the welded connection, and to investigate global and local phenomena occurring in the structure. The final aim was to validate the assumptions and recommendations given in Eurocode, and to shed some light on the stress distribution in the region of the weld.

2. Setting of the problem

In this paper subject of investigation was a common type of connection of two steel structural elements – a beam made of *IPE* cross-section and a column made of welded *I* cross-section. The two elements are connected using a fillet weld across the perimeter of the beam section (Fig. 1). The column is fixed at its base, and the beam is a cantilever loaded at its end by a vertical force. Such system loads the welded connection by a shear force and by a bending moment.

An initial assumption was made that all structural elements and the weld are made of the same material – structural steel S355. Details for

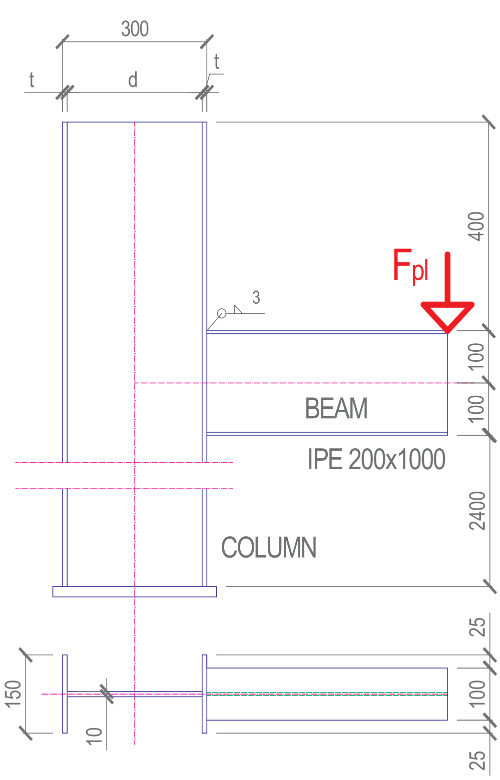

Fig. 1. Technical details of the analysed structure (dimensions in mm).

the material model are given in the Section 4.

The goal was to determine the value of the force F_{pl} that leads to the first occurrence of the plasticization in the structure, and to investigate how the thickness of the column flange influences this value. For this purpose, the models with column flange thicknesses of 10, 15, 20 and 25 mm were analysed. This problem was solved analytically first, and then by using the software package ANSYS, which is based on the FEM. The comparison between analytical and numerical results has led to some interesting conclusions.

3. Analytical solution of the connection

As it has been mentioned above, the problem was first solved analytically, that is, the value of the applied force F_{pl} that leads to plasticization of the analysed structure was determined, i.e., to the first occurrence of plastic strains within the structure, and it was done in accordance with the methodology given in Eurocode [1]. In the following text, this force value will be referred to as the plasticization force.

For the defined problem, an appropriate static system and loading were adopted (Fig. 2a). For such system, the diagrams of the bending moments (M), shear forces (V), and normal forces (N) are shown in Fig. 2b.

Since the design procedure proposed in Eurocode is somewhat different for the three distinct structural elements (the beam, the column, and the weld), the force of plasticization has been determined for each element separately, and then the obtained values compared to get the minimum value as the final plasticization force value for the whole structure. Since the goal was to determine the theoretical force value that leads to plasticization, the EC safety factors regarding the base material (β_w), load (γ) and connections (γ_{M2}) have been omitted.

The Eurocode is commonly used for structural design according to the ultimate limit state analysis, which implies fully plasticized cross sections. However, the aim of this paper was to determine the load value that would lead to the first occurrence of plastic strains in the structure, but not its full plasticization. That was the reason for using the Eurocode design guidelines for welded connections, which are based on the Theory of elasticity [1].

The calculated plasticization force value for the beam was 59.88 kN, and 177.92 kN for the column with the thinnest flange investigated, i.e. the 10 mm thick flange. For the weld the simplified method proposed in EN1993-1-8 was used, and the calculated plasticization force was 28.77 kN. Since the lowest obtained plasticization force value is in the weld, the plasticization is expected to occur there first. Therefore, the plasticization force for the whole structure was adopted as F_{pl} = 28.77 kN, and it is the same for all of the analysed column flange thicknesses.

4. Numerical solution of the connection using FEM and ANSYS Workbench software

4.1. Material model and finite elements used

The FEM analysis involved geometrical and material nonlinearity. The working diagram for steel (stress-strain diagram) was adopted as bilinear kinematic (Fig. 3). Modulus of elasticity was adopted as E = 210 GPa, and the Poisson's coefficient ν = 0.30. Steel is behaving as linearly elastic material up to the yield point f_y = 355 MPa, and after that it enters the plastic range, where its stress-strain behaviour is proportional to the tangent modulus, here adopted as E_t = 2100 MPa (1% of the elasticity modulus value).

FE meshing was done using solid element SOLID186 and SOLID187, which are general homogeneous solid material finite elements with midside nodes included, thus enabling the quadratic approximation [16] (Figs. 4 and 5).