CZASOPISMO INŻYNIERII LĄDOWEJ, ŚRODOWISKA I ARCHITEKTURY
JOURNAL OF CIVIL ENGINEERING, ENVIRONMENT AND ARCHITECTURE
JCEEA, t. XXX, z. 60 (2/13), kwiecień -czerwiec 2013, s. 77 - 86

Lukáš GÖDRICH[1]
František WALD[2]
Zdeněk SOKOL[3]

TO ADVANCED MODELLING OF END PLATE JOINTS

The behaviour of end plate can be described using various analytical models. The most common analytical approach is component method. However this method can be used only for the joints with specific geometry and specific loading. This method gives reliable results for common endplate joints. On the other hand numerical methods, especially finite element method, can be used for the design of the joints with any geometry and any loading. Many works have been published on the modelling of joints using finite element method. This research have been focused on creating the most accurate numerical models using solid elements. Such models are not applicable for practical design. The aim of author's research is to create rules and recommendations for design of joints using simple elements like shell and beam elements only so that this model can be applied for practical design. Absence of guidelines for creation of numerical model using shell and beam elements and for assessment of the joint components limits using of the numerical methods. The aim of Author's research is creation of guidelines for modelling and assessment of end plate of bolted joints. This paper presents results of experimental and numerical investigation of T-stub, which usually simulate behaviour of the end plate. Two different specimens were prepared and experimentally investigated. Numerical models were created on the base of experimental results.

Key words: endplate joint, finite element method, component method, T-stub, complex geometry

[1] Autor do korespondencji: Lukáš Gödrich, Czech Technical University in Prague, Faculty of Civil Engineering, Department of Steel and Timber Structures, Thákurova 7, 166 29 Praha 6, Czech Republic; +420 224 354 767; lukas.godrich@fsv.cvut.cz,

[2] František Wald, Czech Technical University in Prague, Faculty of Civil Engineering, Department of Steel and Timber Structures, Thákurova 7, 166 29 Praha 6, Czech Republic; +420 224 354 757; wald@fsv.cvut.cz,

[3] Zdeněk Sokol, PhD. Czech Technical University in Prague, Faculty of Civil Engineering, Department of Steel and Timber Structures, Thákurova 7, 166 29 Praha 6, Czech Republic; +420 224 354 767; sokol@fsv.cvut.cz

1. Introduction

Endplate joint is one of the commonly used connections in steel structures. Their behaviour is described by several analytical models. Common analytical method for design is component method. This analytical method is simple and reliable for the design of commonly used joints. However, this method is hardly applicable to the design of complex joints. The component method supposes elastic-plastic material behaviour, which leads to the simplified joints behaviour described by moment-rotation curve. Piluso et al. (2001) suggested a method based on the multi-linear material behaviour, which leads to more accurate description of the joints behaviour. The supposed multi-linear diagram is shown in Figure 1. Strains ε_y, ε_h, ε_m and stresses f_y, f_u are given by tensile test. Values ε_u, E_h and E_u are calculated according to Piluso et. al.

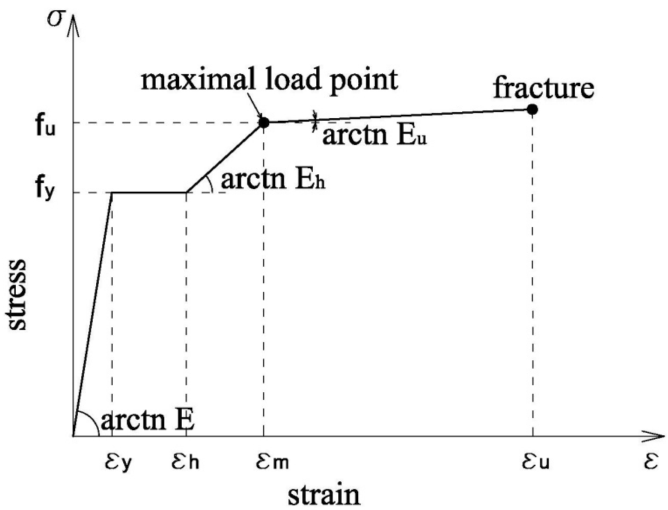

Fig. 1. Multi-linear stress-strain diagram.

Rys. 1. Wielolinowy wykres zależności naprężenie – odkształcenie.

An alternative method for the design of endplate joints could be a numerical finite element method-FEM. However, this method is nowadays used for the investigation of many problems, this method is applied for the design of joints only in same special cases. Many works have been published on the modelling of joints using FEM such as COST C1 (1999). This research has been focused on creating the most accurate numerical model using solid elements. Such models are not applicable for practical design. The aim of this research is to create rules and recommendations for design of joints using simple elements, so that this model can be applied for practical design. The absence of guidelines and recommendations for modelling and assessment of the components limits using of finite element method. If these guidelines will be created, finite element method can be used as an alternative to analytical methods and FEM can even bring many benefits. FEM can be used for the design of complex endplate joints with complex loading. This method provides a real stresses distribution. Familiar user can then easily recognize the critical components and can optimize the design of the joint. The aim of current research is the elaboration of guidelines and recommendations for modelling and assessment of an endplate.

2. Experiments

T-stub model is commonly used to predict the behaviour of the endplate. Two samples of T-stubs connected by two bolts M24 8.8 were designed and experimentally tested to investigate the behavior of the endplate. T-stubs were performed by separating the upper flange of rolled HEB-sections. Dimensions of the samples are given in Figure 2 and Table 1. T-stub's webs were fixed to clamps and samples were subjected to tension force. Scheme of the experiment is shown in Figure 3 right.

Table 1. Dimensions of specimens.

Tabela 1. Wymiary próbek.

Sample	T-stub									
	Form section	t_f	t_w	b_f	r	b	w	e_1	m	e
		[mm]								
1	HEB 300	17,8	10,6	300	27	98,8	164	49,4	5,1	68
2	HEB 400	23,1	13,6	300	27	99,6	169	49,8	6,1	65,5

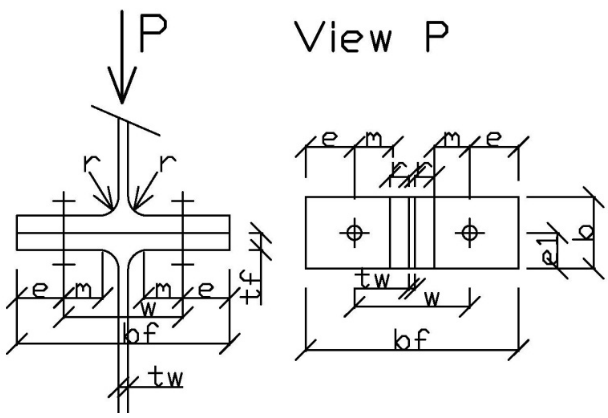

Fig. 2. Dimensions of specimens.

Rys. 2. Wymiary próbek.

Twelve strain gauges were fixed on each T-stub. These strain gages were primarily located in the areas of expected plastic hinges in the T-stub's flanges. Forces in bolts were measured by force washers KMR400 placed under the bolt head. Force washer is a ring containing four embedded strain gauges. The force in the bolt is derived from the deformation of these strain gauges. Deformation of the T-stubs was measured using two inductive sensors. The placement of measuring devices is shown in Figure 3 left.

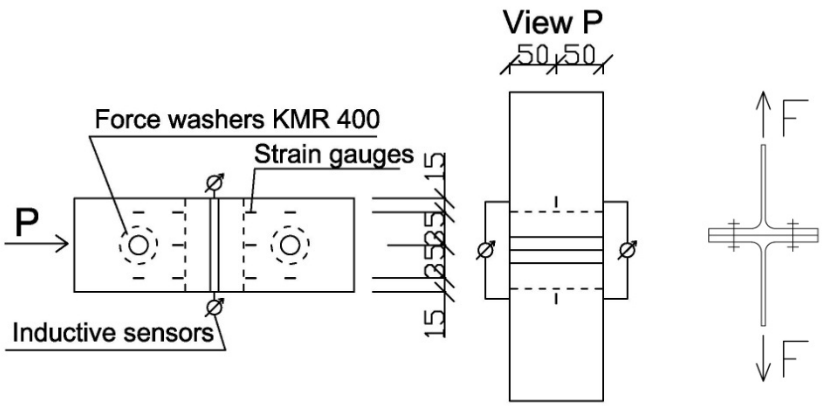

Fig. 3. Placement of measuring devices and scheme of experiment.
Rys. 3. Rozmieszczenie czujników pomiarowych i schemat badania.

3. Numerical Model

3.1. General

Numerical models were created in Midas FEA software and calibrated according to experiment's results. Using of simple numerical model brings benefits in reduction of time for creation of numerical model and calculation. The time required to perform a calculation is for practical design the decisive factor, therefore only shell and beam elements are used in simple numerical model. T-stubs are modeled as shell elements and bolts and contact between flanges of T-stubs is modeled using beam elements.

Two different numerical models have been developed. These two variants of numerical models differ in using of stress-strain diagram of T-stubs material. Multi-linear stress-strain diagram was used in the first variant to validate the numerical model according to experiments. Tensile tests provided data for the stress-strain diagram as on the Figure 1, and the characteristic values are summarized in the Table 2. Elastic-plastic stress-strain diagram was used in the second variant to verify the numerical model according to analytical component method. Module of elasticity and yield strength were again given by material tensile test.

Table 2. Material characteristics.

Tabela 1. Właściwości materiałowe.

Sample	E	E_h	E_u	ε_y	ε_h	ε_m	ε_u	f_y	f_u
	[GPa]			/	/	/	/	[MPa]	
1	190	2,2	0,4	0,00213	0,01213	0,08077	0,6733	405	557
2	190	2	0,46	0,00152	0,01152	0,10052	0,62499	288	467

3.2. Validation of numerical model

The first variant of the numerical model with multi-linear stress-strain diagram was validated according to experiment. Thickness of web and flange elements corresponds to the real measured values. There is a rounding with radius r in the intersection of web and flange. To consider this rounding, which stiffens this part, the reinforcing shell elements are used. Location of the reinforcing elements connection to the flange significantly affects the behaviour of T stub. After validation of numerical model the reinforcing elements were connected to the flange at a distance of 0.5 t_w + 0.5 r from the center of the flange. t_w denotes the web thickness and r radius of rounding.

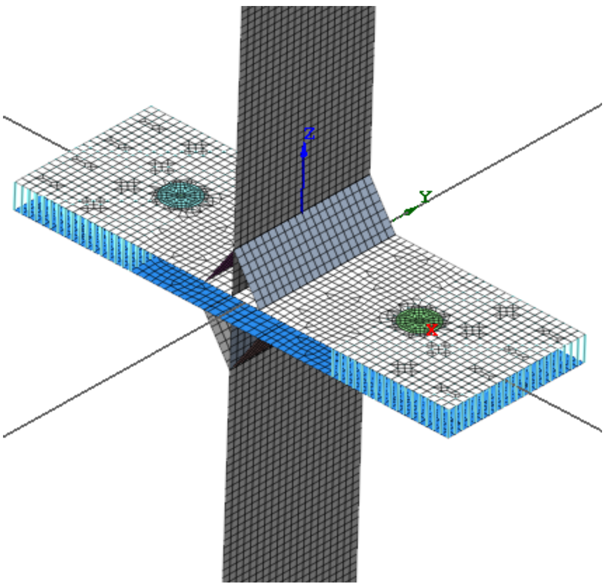

Fig. 4. Validated numerical model.

Rys. 4. Model numeryczny poddany weryfikacji.

Bolts are modeled as beam elements with six degrees of freedom in every node. Cross section of bolt element is circular, constant along the entire length of the element. To consider the real axial stiffness of the bolt element the cross section area of bolt element is given by $A = A_s\ l\ /\ l_b$, where A_s is the area of the real bolt, l length of the beam element l_b clamping length of the bolt.

Shell elements with higher stiffness than the stiffness of the flange were used in the flange in the area where the bolt elements are connected to flange. These elements are applied in circles with a diameter corresponding to the diameter of the bolt. These elements ensure distribution of point-forces from the bolt element to flange elements. Contact between the flanges of T-stubs is ensured by compression beam elements only. Meshes on both flanges of T-stubs are the same and the nodes of these meshes are connected vertically by the compression beam elements only. Validated numerical model is shown in Figure 4.

Results of calculations conducted on validated numerical model were compared with the results of experiments. Comparison of T-stubs deformation was done. Results of computed and measured in tests deformations are compared in the Figures 5 and 6.

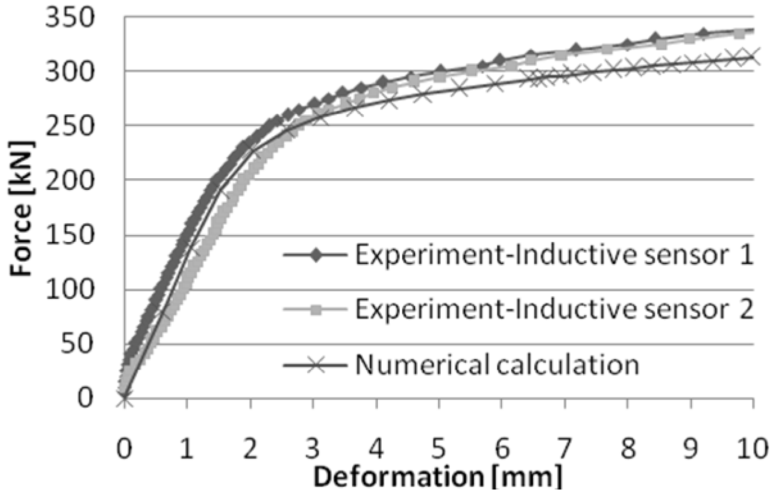

Fig. 5. T-stub deformation, sample 1.

Rys. 5. Odkształcenia króćca teowego, próbka 1.

These results indicate that the numerical model shows slightly higher initial stiffness than the real connection. The yielding of the flange occurs in numerical model slightly earlier than in the experiment. Stiffness after yielding of flange is in numerical model slightly lower in comparison to the experimental results.

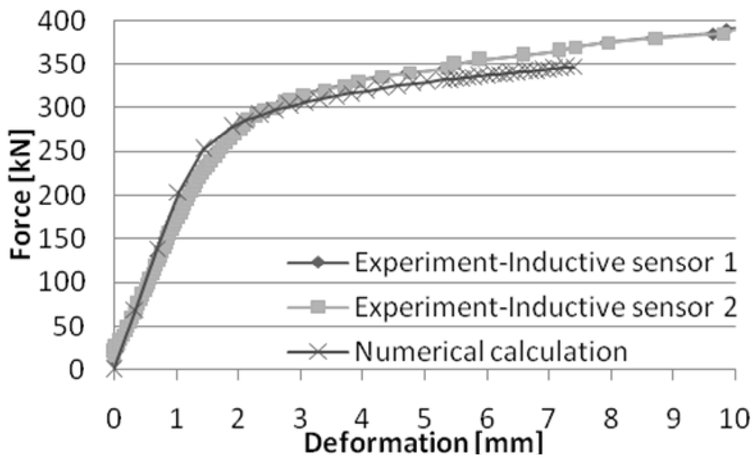

Fig. 6. T-stub deformation, sample 2.

Rys. 6. Odkształcenia króćca teowego, próbka 2.

Very good agreement was achieved for all measured and compared values. Despite small deviations can be said that a validated numerical model shows satisfactory agreement with experiments.

3.3. Verification of numerical model

Numerical models were verified with analytical models. Two different analytical models were used for verification, component method and method with multi-linear material behaviour of T-stub according to Piluso et al. (2001).

Validated numerical model was verified with analytical method according to Piluso and second variant of numerical model with elastic-plastic stress-strain diagram was verified with component method. Verification was done according to deformation of T-stub. Comparison of validated numerical model with multi-linear stress-strain diagram and analytical model according to Piluso is shown in Figures 7 and 8.

However Figure 7 shows very good agreement of numerical model with analytical model, for the second sample shown in Figure 8 is the analytical model very inaccurate.

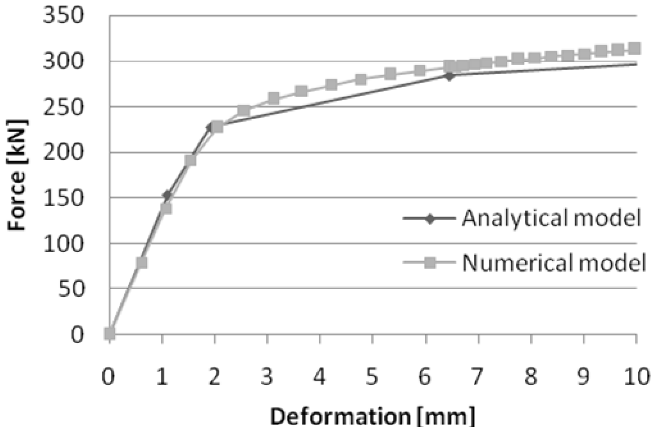

Fig. 7. T-stub deformation, multi-linear stress-strain material behaviour, sample 1.

Rys. 7. Odkształcenia króćca teowego przy zastosowaniu wieloliniowego modelu materiału, próbka 1.

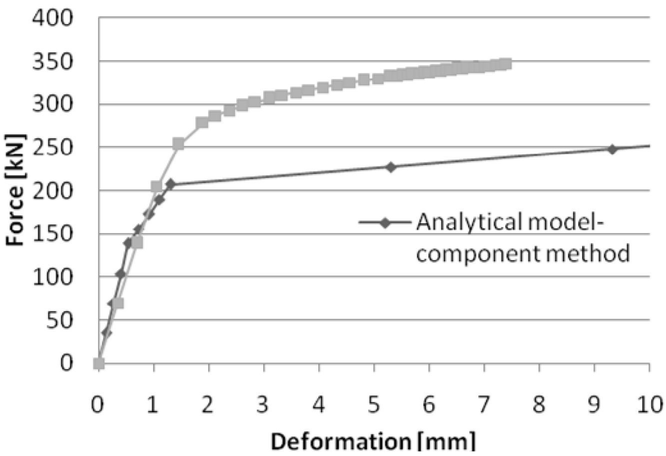

Fig. 8. T-stub deformation, multi-linear stress-strain material behaviour, sample 2.

Rys. 8. Odkształcenia króćca teowego przy zastosowaniu wieloliniowego modelu materiału, próbka 2.

Comparison of component method with numerical model with elastic-plastic stress-strain diagram was done in the next step. It is possible to observe a good agreement of component model and numerical model for both samples. The comparison is shown in Figures 9 and 10. However the numerical model shows a slightly lower resistance, this model describes better the overall behaviour of T- stub.

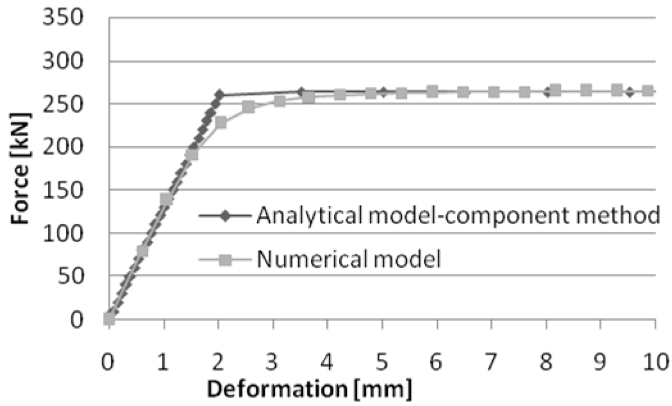

Fig. 9. T-stub deformation, elastic-plastic stress-strain material behaviour, sample 1.

Rys. 9. Odkształcenia króćca teowego przy zastosowaniu sprężysto-plastycznego modelu materiału, próbka 1.

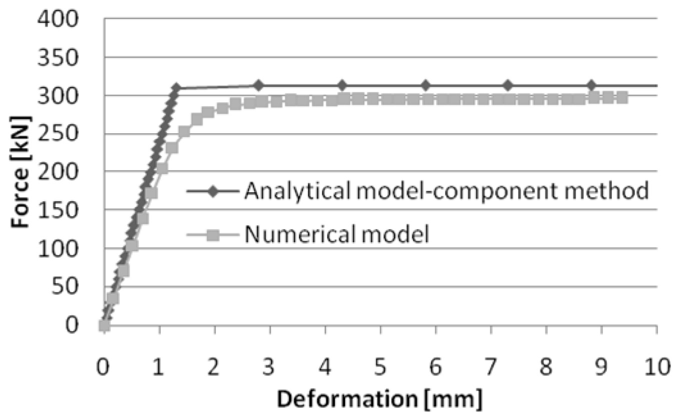

Fig. 10. T-stub deformation, elastic-plastic stress-strain material behaviour, sample 2.

Rys. 10. Odkształcenia króćca teowego przy zastosowaniu sprężysto-plastycznego modelu materiału, próbka 2.

4. Conclusion

Above presented comparisons indicate that the numerical model shows very good agreement with experimental results and results of analytical methods too. It will be possible to establish general rules for the modeling of the end plate based on this validated numerical model. Research will focus now on creation of rules for the assessment of T-stubs flange and bolts. These rules will be verified and extended to the whole end plate joint.

Acknowledgements

This work is supported by TACR - TA03010680 and SGS13/122/OHK1/2T/11.

Reference

[1] Piluso V., Faella C. a Rizzano G.: Ultimate Behaviour of Bolted T-Stubs. I: Theoretical Model. Journal of Structural Engineering, 2001, s. 686-693

[2] Virdi K. S.: COST C1 - Numerical simulation of semi-rigid connections by the finite element method. Report of working group 6 – numerical simulation, 1999

ZAAWANSOWANY MODEL DOCZOŁOWEGO POŁĄCZENIA ŚRUBOWEGO

S t r e s z c z e n i e

Zachowanie się blachy czołowej może być opisane za pomocą różnych modeli analitycznych. Najczęściej podejściem analitycznym jest metoda składnika. Jednak ta metoda może służyć tylko do połączeń o określonej geometrią i sprecyzowanym obciążeniu. Metoda ta daje wiarygodne wyniki dla wielu połączeń doczołowych. Z drugiej strony metody numeryczne, a zwłaszcza metoda elementów skończonych, może służyć do projektowania węzłów z dowolną geometrią oraz poddanych dowolnemu obciążeniu. Wiele prac zostało poświęconych modelowaniu połączeń metodą elementów skończonych. Badania koncentrowały się na tworzeniu najbardziej dokładne numeryczne modeli za pomocą elementów bryłowych. Takie modele nie są stosowane jednak do stosowania w praktyce projektowej. Celem prac autorów jest stworzenie zasad i zaleceń dotyczących projektowania węzłów przy użyciu prostych elementów jak powłokowych czy belkowych, możliwych do zastosowania w praktyce projektowej. Brak wytycznych dotyczących tworzenia modelu numerycznego przy użyciu elementów belkowych i powłokowych ogranicza użycie metod numerycznych. Celem prac autorów jest stworzenie wytycznych w zakresie modelowania i oceny blachy czołowej w połączeniu śrubowym. W artykule przedstawiono wyniki badania eksperymentalne i modeli numerycznych króćców teowych, które zazwyczaj służą symulacji zachowania się blachy czołowej. Zostały przedstawione wyniki badań doświadczalnych dwóch różnych próbek. Na podstawie wyników badań doświadczalnych zostały stworzone modele numeryczne.

Słowa kluczowe: węzeł doczołowe, metoda elementów skończonych, metoda składnikowa, króciec T, złożonej geometria

DOI: 10.7862/rb.2013.18

Przesłano do redakcji: w lipcu 2013 r.
Przyjęto do druku: w sierpniu 2013 r.