SELECTED ENGINEERING PROBLEMS
NUMBER 4
INSTITUTE OF ENGINEERING PROCESSES AUTOMATION
AND INTEGRATED MANUFACTURING SYSTEMS

Artur DUCHACZEK

General Tadeusz Kościuszko Military Academy of Land Forces, Wrocław, Poland
*aduchaczek@poczta.wp.pl

SIMPLIFIED MODELS OF CALCULATION OF WEBS IN LOW-WATER BRIDGE GIRDERS

Abstract: Analysis of development of fatigue fractures with the use of FRANC2D software requires using a simple two-dimensional surface model. The author of this article presented an example of creation of simple calculation models for selected fragments of steel girders, which can be subsequently implemented in FRANC2D software. Creation of a web model is presented upon the example of a low-water bridge main girder.

1. Introduction

The area of application of Finite Element Method (FEM) within the widely-understood mechanics and material strength science is quite considerable. This method was also used in the field of fracture mechanics for analysis of development of fatigue fractures on the basis of a value referred to as stress intensity factor *K*. An example of software enabling calculation of the value of stress intensity factor *K* may include an application referred to as FRANC2D (http://www.cfg.cornell.edu/index.htm), which makes it possible to perform surface analyses relating to development of fatigue fractures in surface structures. Consideration of fractures within assembly holes in fatigue analyses with the use of this software requires using a simple two-dimensional surface model. Construction facilities usually constitute complex and large-sized engineering structures. Therefore, direct use of FRANC2D software for fatigue analyses becomes considerably difficult. The author of this article presented an example of creation of a simplified model of a low-water bridge main steel girder, for which the author plans to analyse development of fatigue fractures within assembly holes and with the use of FRANC2D software.

2. Calculation model of a steel girder

An ordinary INP 400 I-beam with the total length of l_c = 5,60 m was used for the purposes of numerical analyses. The author of this article performed fatigue tests of the structure in 2006 in the testing laboratory of the Institute of Building Engineering of Wroclaw University of Technology [1].

Similar to the study [2], in modelling the analysed I-beam, certain adjustments of its measurements were made, i.e. its measurements were rounded to full millimetres. However, the modifications had little influence on the final value of the field of cross-section of structure A, moment of inertia about bending axis J_x and section modulus in bending stress W_x. Therefore, the above-mentioned adjustment has a marginal influence upon results of analyses performed, which made the very process of the structure modelling even easier.

The analyses commenced with calculation of moment of inertia about bending axis J_x of I-beam model assumed for preliminary numerical calculations. Next, the value of a section modulus in bending stress was determined as W_x = 1490 cm^3. According to the standard [3], the value of section modulus in bending stress W_x for INP 400 I-beam is 1460 cm^3 and, therefore, the assumed I-beam model was characterised by a similar "*resistance*" to bending to the real element.

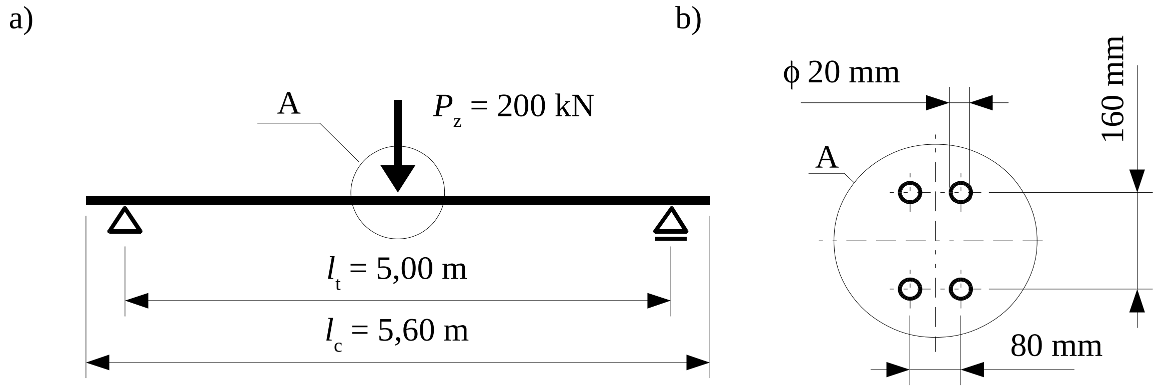

Fig. 1. Diagram: a) of I-beam assumed for the calculations, b) of distribution of assembly holes located in the middle of I-beam span

In modelling the girder, a static scheme of a simple beam loaded with the concentrated force of P_z = 200 kN (Fig. 1a) was assumed. In the middle of span of the analysed I-beam, assembly holes were made in accordance with the diagram presented in Fig. 1b.

The tested element was modelled both as a volumetric model (Fig. 2b) and a surface model (Fig. 2a). However, the selected type of the simplified calculation model was affected by the fact that fatigue calculations will be made ultimately by the author with the use of software enabling use of surface finite elements only.

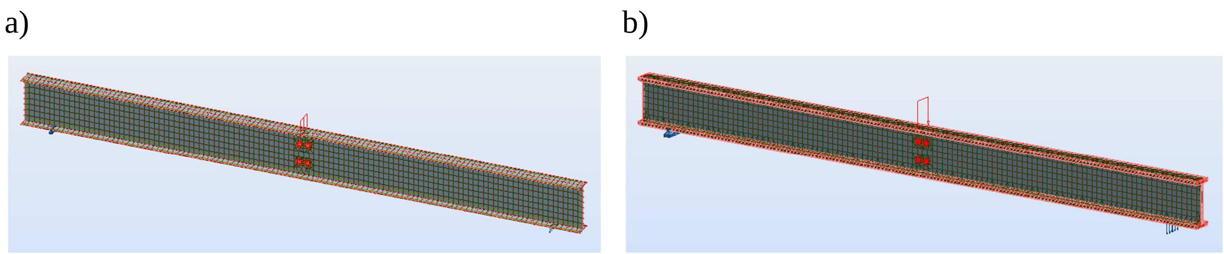

Fig. 2. The calculation model of I-beam as a: a) surface, b) volumetric, model

The modelling process uses a Coons patch meshes method. In case of the volumetric model, there were 16500 finite elements (hexahedral 8-nod elements) used and in case of the surface model there were 3455 finite elements (4-nod quadrangular elements above all and 3-nod triangular elements only within the holes) used. In the modelling process in most of the area of the section, a maximum size of 50 mm of the finite element was assumed. However, the mesh of finite elements was compacted within the analysed assembly holes with assumption of 100 nodes on circumference of a circle (being contours of the hole).

The table 1 presents results of the numerical analyses performed. The results differed slightly from one another, which was probably due to use of various types of finite elements. The presented results (normal stress values) were read from the middle of span of I-beam within the zone of extended fibres and, in particular, around the assembly holes ($\square_{o1}$, $\square_{o2}$), in the bottom flange ($\square_{max}$) and at the connection of the web and bottom flange ($\square_{sr}$).

Fig. 3 presents maps of distributions of normal stresses within the assembly holes in the web of the analysed I-beam. Their analysis showed a distribution of normal stresses within the assembly holes similar for both calculation models. It was also confirmed that the assembly holes constituted stress concentrators.

a) b)

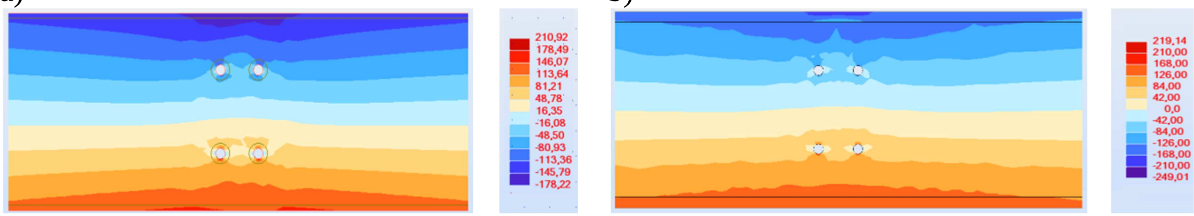

Fig. 3. Fragments of the map of normal stresses within the assembly holes in the web of the analysed I-beam for the: a) surface, b) volumetric, model

Tab. 1. Results of numerical analyses of the steel section

No.	The structure modelled with the use of elements	Normal stress values in [MPa]			
		in the bottom flange $\square_{max}$	within assembly holes in the area extended from the support side		at the contact of the web and bottom flange $\square_{sr}$
			non-sliding $\square_{o1}$	sliding $\square_{\square\square}$	
1.	volumetric	168,40	211,69	208,58	148,88
2.	surface	163,79	204,69	208,84	144,43

Analysing the static scheme of the beam presented in fig. 1a, simple arithmetic calculations were made and a maximum bending moment in the middle of span of the analysed structure was obtained as M_{max} = 250 kNm. Therefore, maximum normal stresses in the bottom flange of I-beam amount to $\square_{max}$ = 167,78 MPa and maximum normal stress values in the web of the analysed I-beam amount to $\square_{sr}$ = 149,33 MPa.

Thus, analysing normal stress values obtained both in analytical calculations and numerical analyses (table 1), it may be stated that they are similar at a satisfactory level.

3. Simplified models of a web of a steel girder

The study proposes four simplified calculation models for a selected fragment of a web of a steel girder (fig. 4 and table 2). The assumed simplified calculation models should ensure stress values in the selected web σ_{sr}, which correspond to stress values in the web of the girder modelled as a whole structure (INP 400 I-beam) presented in fig. 2a.

Table.2. Description of variables used in fig. 4

No.	Symbol	Description of a given value and method of calculation	Value
1.	h_{sr}	Web height	0,356 m
2.	h_{max}	I-beam height	0,400 m
3.	b_{sr}	Web thickness	0,014 m
4.	W_x	The calculation section modulus in bending stress for the entire I-beam	0,001490 m^3
5.	W_{zast}	$W_{zast} = \frac{b_{sr} h_{sr}^2}{6}$	0,000296 m^3
6.	σ_{max}	$\sigma_{max} = \frac{M_{max}}{W_x}$; $M_{max} = \frac{P_z l_t}{4}$; P_z = 200 kN, l_t = 5,00 m (fig. 1a)	168,92 MPa
7.	σ_{sr}	$\sigma_{sr} = \frac{h_{sr}}{h_{max}} \sigma_{max}$	149,33 MPa
8.	q_1	$q_1 = \frac{4P}{h_{sr}}$; $P = \frac{3M_{zast}}{2h_{sr}}$; $M_{zast} = W_{zast}\sigma_{sr}$	2092,54 kN/m
9.	q_2	$q_2 = \frac{4P_{kor}}{h_{sr}}$; $P_{kor} = \frac{3M_{kor}}{2h_{sr}}$; $M_{kor} = W_{zast}\sigma_{kor}$; $\sigma_{kor} = \sigma_{sr} - \sigma_1$; $\sigma_1 = \frac{M_1}{W_{zast}}$; $M_1 = \frac{P_z l_t}{4}$; P_z = 200 kN, l_t = 1,00 m	- 274,53 kN/m
10.	q_3	$q_3 = \frac{8M_{zast}}{l_t^2}$; $M_{zast} = W_{zast}\sigma_{sr}$; l_t = 1,00 m	353,60 kN/m
11.	P_1	$P_1 = \frac{4M_{zast}}{l_t}$; $M_{zast} = W_{zast}\sigma_{sr}$; l_t = 1,00 m	176,80 kN

Using the presented four calculation models (fig.4), normal stresses in simplified models of a web of the steel section were made. In the table 3 and in fig. 5 results of the numerical analyses are presented.

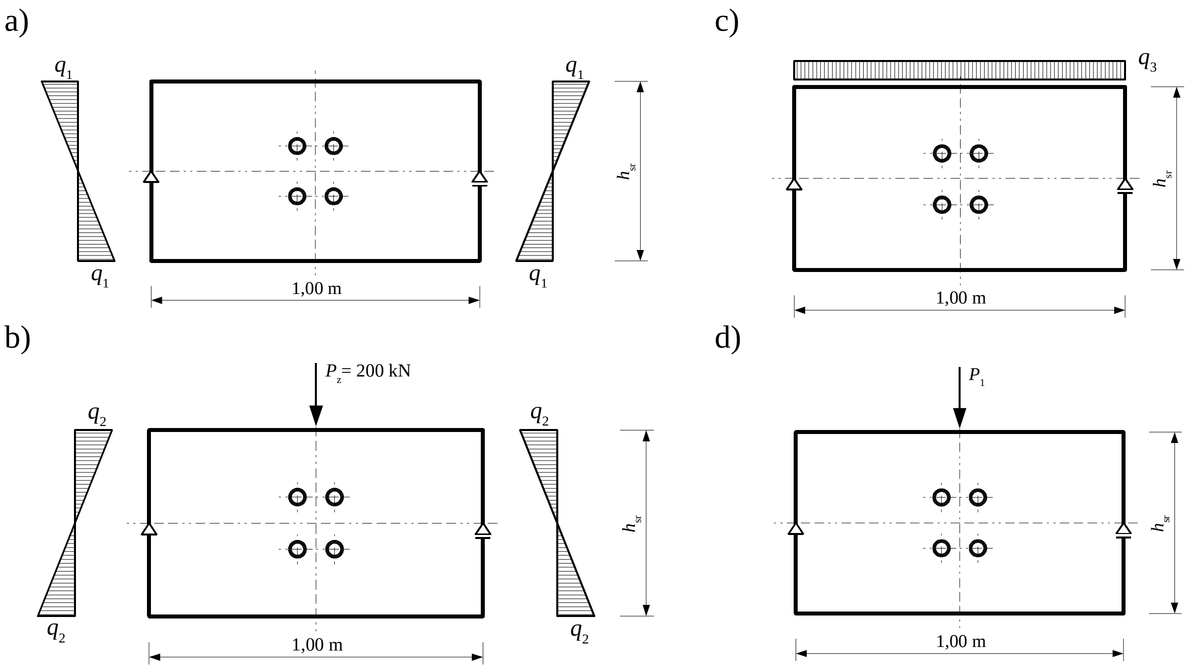

Fig. 4. Simplified model: a) No. 1, b) No. 2, c) No. 3 and d) No. 4 of the web fragment

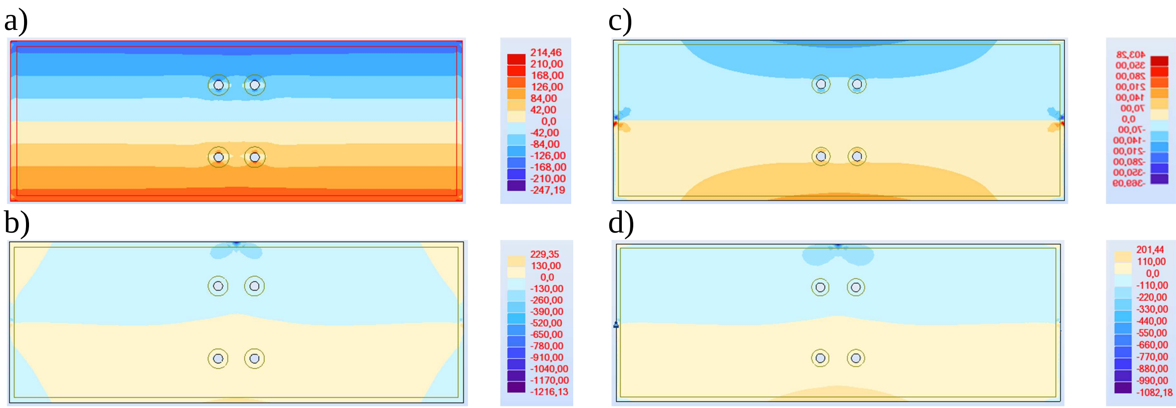

Fig. 5. Maps of distribution of normal stresses within the assembly holes in the simplified model of a web of the analysed I-beam, for the load model number: a) 1, b) 2, c) 3 and d) 4.

Analysing maps of stresses shown in fig. 5 as well as normal stress values in points characteristic for the analysed models, i.e. in extreme fibres of the web in the middle of its span $\square_{sr}$ and within the assembly holes $\square_{o1}$ and $\square_{o2}$ (table 3), it turns out that the load model No. 1 (fig. 4a) gives values similar to the values obtained both for calculation models of the entire steel I-beam (fig. 3a and table 1) and analytical calculations.

However, considering this load scheme during fatigue calculations relating to potential fractures developing from the assembly holes, one should be aware that the influence of P_z concentrated force is totally disregarded, which, considering the author's present experience, may have considerable effects on the obtained test results [4].

Tab. 3. Results of numerical analyses of the simplified model of a steel section

No.	For the calculation model (fig. 4)	Normal stress values in [MPa]		
		within assembly holes in the area extended from the support side		at the contact of the web and bottom flange $\square_{sr}$
		non-sliding $\square_{o1}$	sliding$\square\square_{02}$	
1.	1	209,26	214,45	153,06
2.	2	184,12	186,62	142,65
3.	3	201,02	209,42	158,35
4.	4	185,99	189,88	143,88

4. Conclusion

It seems that results of numerical tests made in the future with the use of FRANC2D software can be more reliable (i.e. similar to laboratory calculations), if the model No. 1 and, possibly, model No. 2 (fig. 4a and 4b) is used. Despite the fact that the model No. 2 (fig. 4b) creates considerable interferences in distribution of normal stresses, in particular, in the compressed area of the web, this, however, considers influence of the cutting force, including in the area of the section. This fact may be significant, if we wish to consider both fractures developing according to the fracture mode I and II and a combined system of the conditions [5], [6], [7].

Results of fatigue analyses relating to development of fractures within the assembly holes for the presented calculation models as made with the use of FRANC2D software will be presented by the author in subsequent studies.

References

1. Mańko Z, Duchaczek A.: *Badania niskocyklowe dwuteowników stalowych wykorzystywanych do budowy mostów niskowodnych. Cz. 1.* Zeszyty Naukowe WSOWLąd., 2006, nr 4, s. 54–65.
2. Duchaczek A., Mańko Z.: *Optymalizacja położenia otworów montażowych w dźwigarach stalowych mostów niskowodnych*. Redakcja Zeszytów Naukowych Wyższej Szkoły Oficerskiej Wojsk Lądowych, 9 marzec 2013.
3. PN-91/H-93407: Stal. Dwuteowniki walcowane na gorąco.
4. Duchaczek A., Mańko Z.: *Uwzględnienie mieszanego sposobu pękania w analizach zmęczeniowych dźwigarów stalowych*. Czasopismo Techniczne. Budownictwo. Politechnika Krakowska. 2012, R. 109, nr 20 (3-B), s. 51-59.
5. Nemitz A.: *Mechanika pękania*. Wydawnictwo Naukowe PWN, Warszawa 1998.
6. Rozumek D.: *Mieszane sposoby pękania zmęczeniowego materiałów konstrukcyjnych*. Politechnika Opolska, Opole 2009.
7. German J.: *Podstawy mechaniki pękania*. Politechnika Krakowska, Kraków 2011.