DE GRUYTER OPEN

CIVIL AND ENVIRONMENTAL ENGINEERING REPORTS

ISSN 2080-5187 CEER 2014; 14 (3): 27-36

DOI: 10.1515/ceer-2014-0023

NUMERICAL STRENGTH ANALYSIS OF A COMPLEX, STEEL SHELL STRUCTURE

Grzegorz BURCZYŃSKI[1], Jakub MARCINOWSKI[1,2]

[1] Projektowanie Konstrukcji Budowlanych, Grzegorz Burczyński

Żary, Poland,

[2] University of Zielona Góra, Faculty of Civil and Environmental Engineering, Poland

The paper deals with the numerical modelling of a complex, steel shell structure. The part under analysis is the upper segment of a steel pylon, which consists of several cylindrical shells and one conical segment. Particular parts of the structure are welded together. Geometrical and loading data calculations were performed for the particular material for both an ideally elastic case and an elasto-plastic case. The conclusion that the structural member analysed required strengthening were drawn on the basis of these results. The structural modification was proposed and additional calculations for this modified structure were also performed. Introduced additional shell elements locked the mechanism of plastic flow. The proposed modification can be treated as a possible strengthening concept. The whole analysis was performed by means of the ABAQUS system but some stages of calculations were also verified by the COSMOS/M system.

Keywords: steel, shell structure, tubular members, linear elastic static analysis, geometrically and materially nonlinear analysis

1. INTRODUCTION

Structural members of steel frame structures are usually modelled as truss or bar elements, whose mechanics results from the Euler-Bernoulli theory of beams [1]. Users of commercial, numerical packages often forget about important and very strong assumptions accompanying this theory. The most important of these assumptions concerns slenderness. In terms of accuracy, correct results required

[1] Corresponding author: University of Zielona Góra, Faculty of Civil and Environmental Engineering, Szafrana 1, 65-516 Zielona Góra, Poland,
e-mail: j.marcinowski@ib.uz.zgora.pl, tel.+48683282527

in engineering calculations, can be obtained when the slender members which have a ratio not less than ten between length and the dominant section dimension, are replaced with bar finite elements. If this condition is not satisfied (members are rather squat) and, in addition, the bar has a thin-walled cross section, ordinary bar elements should be used with great attention.
The paper presents the numerical modelling of a crucial portion of a big steel frame structure. It is the upper part of a high pylon (the steeple of a pylon). In the pylon the main load-carrying cables of the suspension roof and stay cables, which stabilize the pylon, are anchored. Simplified calculations, in which beam finite elements were used, showed a significant exertion of the pylon members, but under the predicted loads structural safety was still not endangered.

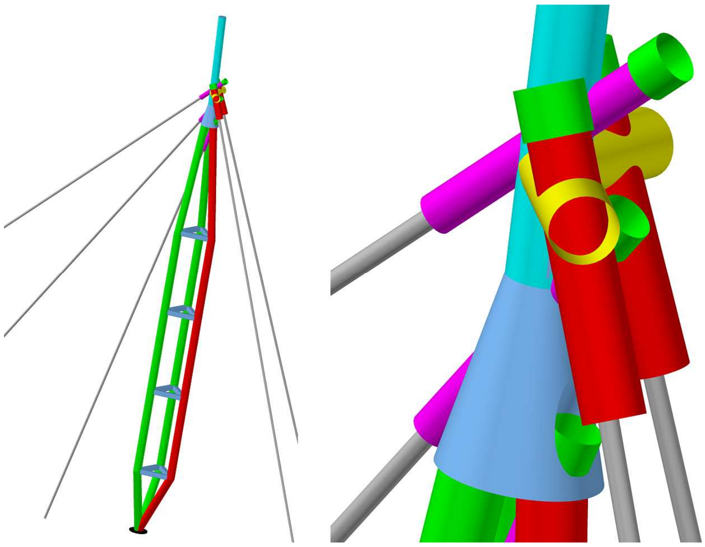

Fig. 1. Isometric view of the pylon - on the left, the steeple analyzed - on the right

The calculations presented in the paper were made for the purpose of verifying the part of the pylon mentioned. All members were modelled as shells since almost all have a cylindrical shape except for one segment, which is a cone. The linear, static analysis performed demonstrated stresses that considerably exceeded the yield limit. Then, another analysis was conducted, in which a plastic flow phenomenon was taken into account. It turned out that with the load intensity of around 70% of nominal intensity, a global plastic flow mechanism would occur and the entire structure would collapse. Additional elements strengthening the crucial portion of the structure were designed and analysed numerically. An elasto-plastic analysis was conducted and proved that the proposed reinforcement was efficient.

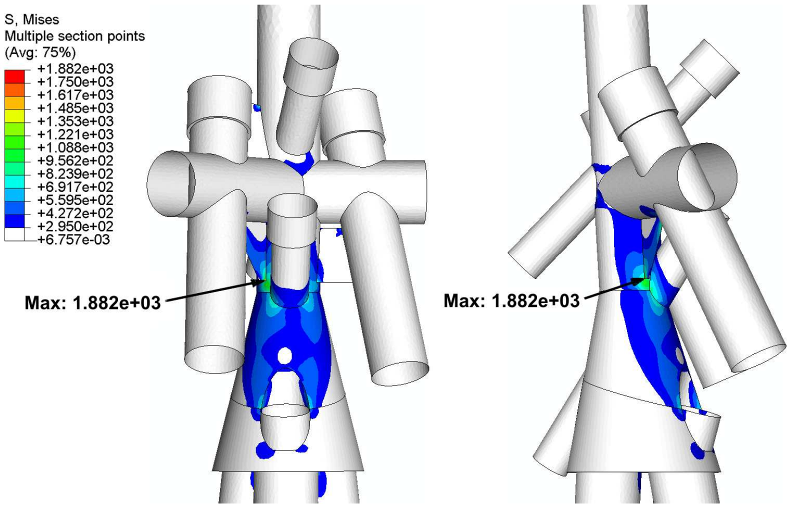

Fig. 2. Maps of equivalent stresses in two views in MPa – a linear analysis

The analysis presented confirms the necessity of applying more sophisticated numerical modelling, particularly in those cases, when utilizing simple bar elements can be doubtful. Using the results of such a simplified analysis could have serious consequences in this particular case.

The ABAQUS commercial software [2] was used to obtain the results presented in this paper. Some stages of the calculations were verified by means of the COSMOS/M package [3] as well.

2. TECHNICAL SPECIFICATION

The pylon under analysis is about 27 m high. Three of its edges (legs) are connected together by triangular lacings with a side length of 1.5 m (see Fig. 1). The pylon is made of 18G2 steel (equivalent to E355 grade according to EN 1993-1-1) and is supported by an elastomeric pot bearing having a 6000 kN capacity. The load-carrying cables of the suspension roof are attached to the pylon via guide pipes and retaining rings. These cables form three clusters, each comprising three cables, and each cable having a $5\times1.5\ \mathrm{cm}^2$ section area. Structural stability is ensured by two main stay cables consisting of 27 wires, each having a $1.5\ \mathrm{cm}^2$ section area. In isometric view (Fig. 1) all three load-carrying clusters of cables are replaced by a single cable (on the left-hand side of the pylon in Fig. 1). The main elements (the longitudinal bars, the stay cable cantilevers, the tube of the steeple) are made of Ø355.6×20 pipes, the stay cable guide pipes have a Ø323.9×20 section and the load-carrying cable guide pipes -

Ø219.1×16.The pylon's components are connected by welding, except for three bolted assembly joints placed in the middle of the pylon.

3. LINEAR STATIC ANALYSIS

In order to create an appropriate numerical model, the geometry of the structure was generated. On the basis of drawings delivered by the manufacturer of the pylon, the three-dimensional solid model was prepared by means of AUTOCAD software. Then, this 3D solid model was used to create a three-dimensional surface model, which was subsequently exported to separate ACIS files. Next, all the surfaces created were imported to ABAQUS. A triangular FE mesh was generated automatically, and individual parts of the model were attached together using the *tie* constraint (comp. [2]).

In the numerical model the cables were removed and the concentrated forces were placed at the same points. The values of these forces were known from the previous analysis of the whole structure performed by means of ROBOT MILLENNIUM software [4]. Forces in the load-carrying clusters of the cables starting from the top of the pylon equal 1716 kN, 1212 kN and 642 kN respectively, whereas forces in the stay cables - 2470 kN and 2076 kN. These values were established during the static analysis of the complete roof. This part of the analysis is beyond the scope of the present work.

It was assumed that the directions of the concentrated forces coincide with the axes of all five guide pipes.

In the place, where longitudinal bars turn from tapered into parallel (about 10 m down from the top of the steeple), fixed supports were introduced; it means that the lower portion of the pylon was removed. Thus, a certain part of the longitudinal bars, lacings and pylon's base plate were excluded from the analysis. Only the upper part of the pylon is the subject of the analysis.

A global analysis of the whole pylon was also conducted, but differences in results were so minor that a detailed analysis was performed only for the separated part of the pylon and these results are presented in the paper.

Material parameters adopted for linear elastic analysis were as follows: the Young modulus E=205 GPa and the Poisson ratio ν=0.3. The FE mesh generated consisted of 81737 triangular S3-type elements, suitable for modeling thin as well as thick shells [2]. The average element size of most, triangular finite elements oscillated within about 20 mm. Because of a significant mesh density, the outline of the mesh has been removed from the drawings presented in the paper. The discrete model that was obtained consisted of ca. 250 000 DOF. A PC with an AMD 3.5 GHz processor and 2 GB RAM memory was used during the calculations.

The map of equivalent stresses relevant to the Huber-Mises-Hencky exertion hypothesis is presented in Fig. 2. The highest design value of stresses, allowable according to the Polish code [5] for a particular steel grade is 295MPa, so lower stresses were not presented in the figure. The absolute maximum (1882MPa) exceeds over six times the allowable level. The highest stress concentrations occur around the places where stiffening ribs are connect to the main pipe of the steeple, however, they are not caused only by local notches, but spread across a significant area of the main pipe of the steeple, the cone, the stay cable cantilevers and the middle load-carrying cable guide pipe. It is obvious that the ultimate limit stress was distinctively exceeded.

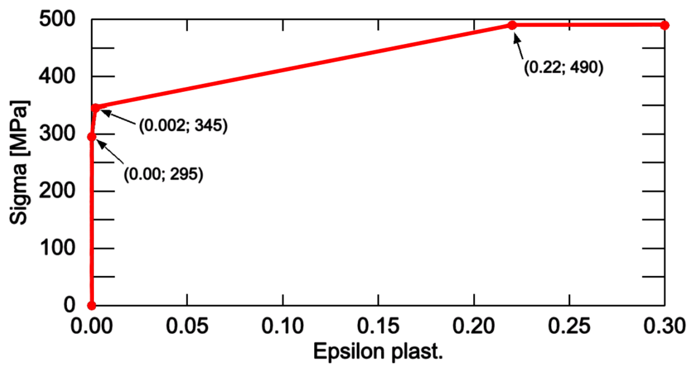

Fig. 3. The adopted stress-plastic strain relation in uniaxial stress state

4. NONLINEAR STATIC ANALYSIS

Obviously, in a real structure stresses could not be so high. It is certain that the material yielding and the stress redistribution took place. In order to analyse such processes the authors decided to carry out calculations in a nonlinear range, both in a material and a geometrical sense.

The previously generated linear model was used to prepare a nonlinear one. The only feature modified was the material model – now assumed as the ABAQUS built-in classical plasticity [2]. Apart from the Young modulus and the Poisson ratio, ABAQUS requires the relation between the stress and plastic strain in uniaxial stress state (Fig. 3). It was assumed that below 295 MPa steel had ideal elasticity. Beyond this value plastic strains occur and they equal 0.002

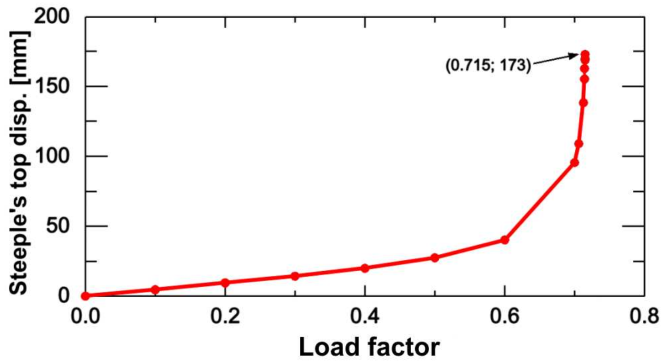

Fig. 4. Resultant displacement of the steeple's top as a function of the load factor

for stresses at the level of 345 MPa. If stresses are outside the range of 345 MPa, the steel exhibits purely elastic behaviour up to the value of 0.22, corresponding

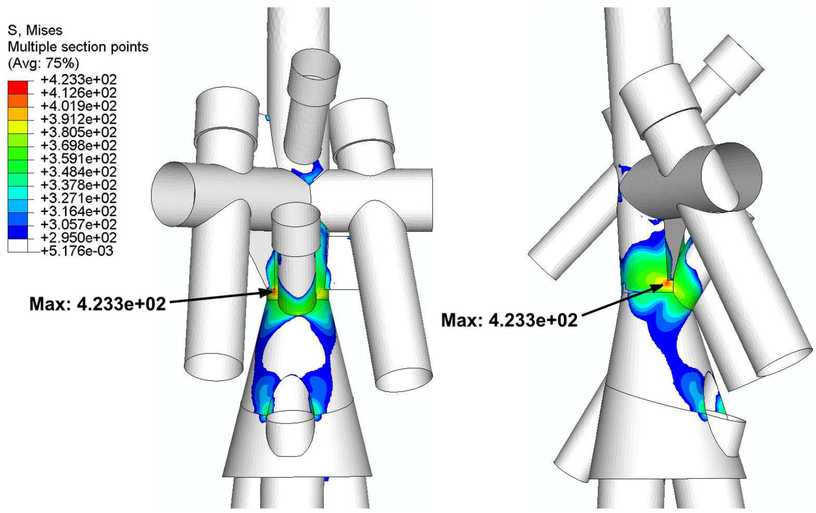

Fig. 5. Map of equivalent stresses in two different views in MPa – nonlinear analysis

to the stress value of 490 MPa. For strains beyond 0.22 the stress-strain relation is ideally flat, so the stress cannot exceed 490 MPa. Characteristic points of the graph have been taken from the Polish code [5].

A gradual, smooth increase of the loads applied was assumed (one parameter loading was assumed). The load factor was increased until the solution algorithm diverged. The automatic step length cutback was set in ABAQUS options [2].
The resultant displacement of the steeple's top versus the load factor graph (Fig. 4) shows how much the structure stiffness changes. The first, local yielding occurs just at the 0.2 level of nominal load intensity. Starting from a load factor of 0.6 a distinct decrease of structural stiffness can be observed. Beyond the value of 0.7 the plot is almost vertical, the stiffness falls dramatically and the plastic flow mechanism appears.
The solution algorithm fails for a load factor equal to 0.715. Equivalent stresses greater than 295 MPa, at the instant of pylon's collapse, are presented on the stress map (Fig. 5). Large yielding areas can be observed, which in connection

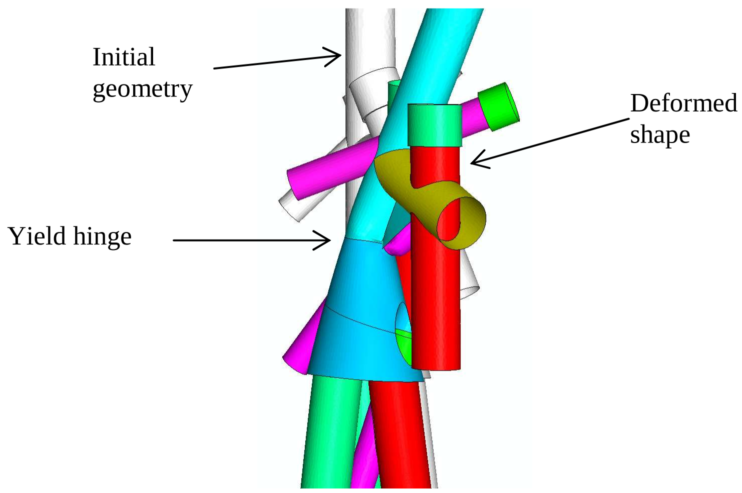

Fig. 6. The failure mechanism, scale 10:1

with the deformation plot (Fig. 6) indicate the weakest point on the steeple, namely the central tube of the steeple, just over the cone. Figure 6 depicts the plastic flow mechanism (deformations are amplified 10 times). We deal with the failure state at the level of 70% of nominal load intensity.

5. NONLINEAR ANALYSIS OF THE STRENGTHENING CONCEPT

The main purpose of the designed strengthening was to lock the founded failure mechanism. A number of concepts were considered. During the designing process not only were structural issues taken into consideration, but also architectural aspects were treated equally seriously. The steeple is the highest part of a representative building and its form should be aesthetic.

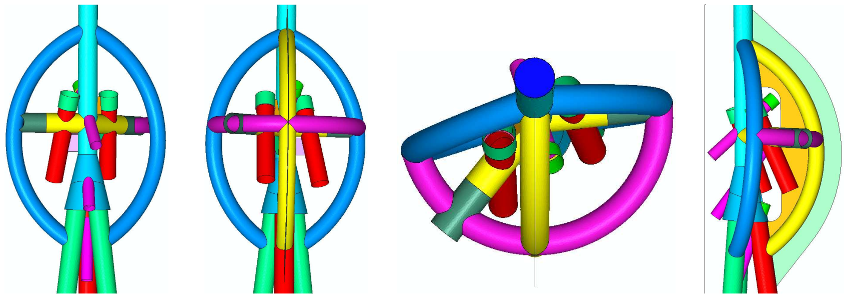

Fig. 7. Model of the strengthening in different views

The strengthening concept selected is shown in Fig. 7. Its main elements are three meridian tubular semi-rings, one circumferential tubular semi-ring and two additional strengthening plates, welded to the middle meridian semi-ring. The stay cable cantilevers were elongated by pipes of the same diameter up to the circumferential semi-ring. Justification for such a solution was obvious: it was necessary to pass round the very weak area of the structure by the introduction of additional structural members and to support these members in places to which the stream of stresses could be safely transmitted.

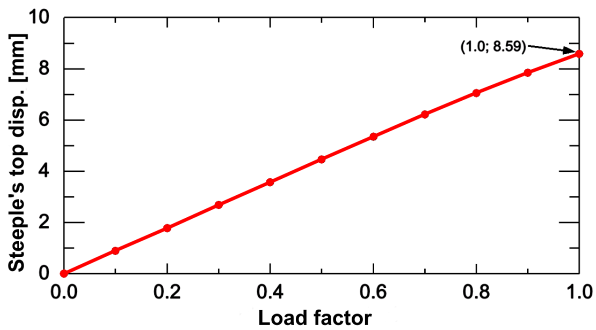

Fig. 8. Resultant displacement of the steeple's top as a function of the load factor for the strengthened structure

The whole geometry was modeled once again similarly to the procedure previously presented. An identical material model, loads and boundary conditions of the same segment of the pylon were assumed. A physically and geometrically nonlinear analysis was conducted in the way previously described.

In Fig. 8 a nonlinear equilibrium path as the resultant displacement of the top of the steeple in a function of a dimensionless load factor is shown. The plot presented is almost linear, which proves the preservation of the constant stiffness of the structure during the entire loading process.

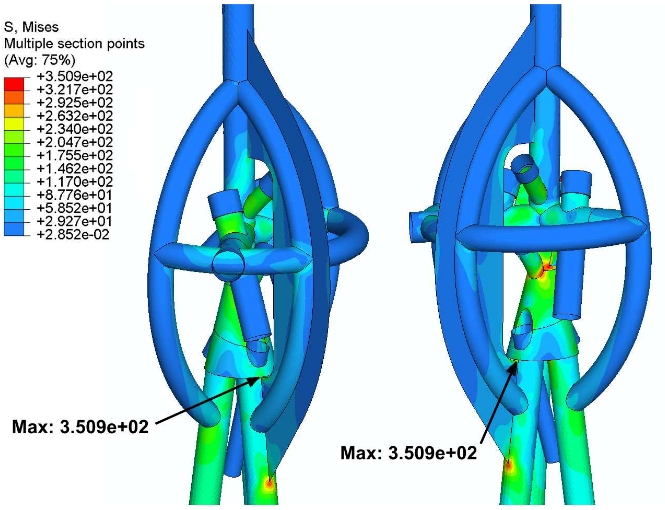

Fig. 9. Map of equivalent stresses in two different views in MPa – nonlinear analysis of the strengthened structure

The map of equivalent stresses for the nominal load state (load factor = 1.0) is shown in Fig. 9. Relatively high stresses occur in various areas, with a local maximum at the level of 351 MPa.
The resistance of the pylon strengthened in this manner can be recognised as sufficient.

6. RECAPITULATION

The paper presents a numerical static analysis of a segment of a complex, steel shell structure. The structure was modelled as a system of many shell parts, in order to render with possible accuracy the behaviour of the structure subjected to external loads. The linear analysis pointed out that the allowable stress level had been significantly exceeded, although the initial static analysis, in which simple bar elements had been used, had not suggested any threats to structural safety.
The nonlinear analysis conducted enabled the identification of the plastic flow mechanism, which appeared at the level of 70 % of nominal load intensity.
A concept for strengthening the crucial structure segment was proposed. The analysis performed proved that the reinforcement presented was efficient. However, a still high level of received stresses and significant technological

problems suggest that the total re-designing and complete replacement of the steeple should be considered.

REFERENCES

1. Gere J. M.: *Mechanics of materials*, Brooks/Cole, 2001.
2. ABAQUS, Inc., *Analysis User's Manual,* Abaqus 2006.
3. COSMOS/M, *Finite Element Analysis System*, *Version 2.9*, Structural Research and Analysis Corporation, Electronic Manual, Los Angeles, California, 2002.
4. ROBOT MILLENNIUM, *User's electronic manual*. Kraków, 2007.
5. PN-90/B-03200 *Steel structures. Static calculations and design.*

NUMERYCZNA ANALIZA WYTRZYMAŁOŚCIOWA PEWNEJ ZŁOŻONEJ, STALOWEJ KONSTRUKCJI POWŁOKOWEJ

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

Przedmiotem pracy jest numeryczne modelowanie pewnej bardzo złożonej, stalowej konstrukcji powłokowej. Analizowana szczegółowo część jest górnym fragmentem stalowego pylonu, na który składa się kilka odcinków powłok cylindrycznych oraz jeden segment stożkowy. Te poszczególne fragmenty konstrukcji były ze sobą połączone spawaniem. Dla znanych parametrów materiałowych, geometrycznych i obciążeniowych wykonano obliczenia w zakresie idealnie sprężystym oraz w zakresie sprężysto-plastycznym. Na podstawie tych obliczeń wyciągnięto wniosek o konieczności wzmocnienia tej części pylonu. Zaproponowano istotną modyfikację istniejącej konstrukcji i wykonano dla niej ponownie obliczenia. Wprowadzone dodatkowe elementy powłokowe zablokowały mechanizm plastycznego płynięcia. Zaproponowaną modyfikację można potraktować jako jedną z możliwych koncepcji wzmocnienia konstrukcji. Wszystkie analizy numeryczne zostały wykonane za pomocą systemu ABAQUS. Pewne wybrane fragmenty obliczeń były weryfikowane także z pomocą systemu COSMOS/M.

Słowa kluczowe: stalowa konstrukcja powłokowa, elementy rurowe, liniowa analiza statyczna, analiza geometrycznie i materiałowo nieliniowa

Editor received the manuscript: 26.09.2014