9th International Conference on
Steel and Aluminium Structures (ICSAS19)
Bradford, UK, 3-5, July, 2019

CRITICAL ASSESSMENT OF THE DESIGN OF AN ELECTRIC TRANSMISSION TOWER

Marios-Zois BEZAS[a], Mike TIBOLT[b], Jean-Pierre JASPART[a] and Jean-François DEMONCEAU[a]

[a] Steel and Composite Construction, UEE Research Unit, Liege University, Belgium
Emails: Marios-Zois.Bezas@uliege.be; jean-pierre.jaspart@uliege.be; jfdemonceau@uliege.be

[b] ArcelorMittal Global R&D, CiA, Luxembourg
Email: mike.tibolt@arcelormittal.com

Keywords: Transmission tower; steel; angle cross-sections; instability; modelling; non-linear analysis.

Abstract. *Present paper focuses on the critical assessment of the response of an electric transmission tower designed under gravity and wind loads according to available dedicated norms. Available numerical tools are used, with reference to different types of structural analyses. The tower has been initially designed according to the existing rules and under some common assumptions, using the TOWER software. In order to validate this design, the tower has then been simulated with the FINELG non-linear finite element software using beam elements, considering relevant imperfections as well as geometrical and material non-linearities. Different types of analyses have also been performed: first and second order linear elastic analyses, elastic instability analysis and second order plastic analyses. The results of those analyses as well as their critical assessment are summarized in the present paper. This study is a part of an ongoing European-funded RFCS project called ANGELHY involving National Technical University of Athens (NTUA - coordinator), CTICM (France), Liège University and ArcelorMittal and SIKA companies.*

1 INTRODUCTION

In Europe, the design of transmission towers for overhead electrical lines can be carried out according to EN 1993-3-1 [1] and for electrical lines exceeding 1 kV according to EN 50341-1: 2012 [2] and EN 50341-2-4:2016 [3]. According to these norms, the design of a transmission tower is carried out through a first order linear elastic analysis, adopting some common assumptions for the modelling. Then the members and sections are checked by means of specific design rules. In the present paper, the tower has been initially designed according to EN 5034-1:2012 by means of a software called TOWER [4].

The objective of the present study is to validate the initial design made with the TOWER software. For this purpose, the tower will be simulated with the non-linear finite element software FINELG [5]. TOWER and FINELG software will be compared at two levels: results of the frame analysis in the elastic range and then at factored design loads. For this study EN 1993-3-1 will be mainly used, with references to EN 1991-1-4 [6] and EN 50341-2-4: 2016, when this is requested in [1].

But to have a global overview of the actual tower's response, an elastic instability analysis will be performed and will be complemented by a second order linear elastic one. Then a non-linear plastic analysis will be achieved so as to evaluate the influence of plasticity on the tower response. Finally, the influence of the initial imperfections will also be investigated.

However, material and geometrical non-linearities combined with imperfections (member out-of-straightness and structural out-of-plane) will affect the response of the tower. Therefore, a full non-linear analysis will be performed to check the validity, in terms of resistance and stability, of the initial design made with the TOWER software.

2 THE INITIAL DESIGN BY TOWER SOFTWARE

The Danube tower is the typical typology of current and future transmission lines and is therefore selected for this case study. The geometry of the tower is indicated in Figure 1. In the framework of the present study, only a suspension lattice steel tower is designed and not all the transmission line. The tower is supposed to be erected in the “Erzgebirge” in Saxony (Germany) and the line is considered as straight along the segment the tower is part of. According to the German national annex of EN 1991-1-4 & EN 50341-2-4, the region is located in wind zone 2. The wind span between two successive towers is 350 m, while the weight span, due to significant height differences, equals 1,5·350 = 525 m [7].

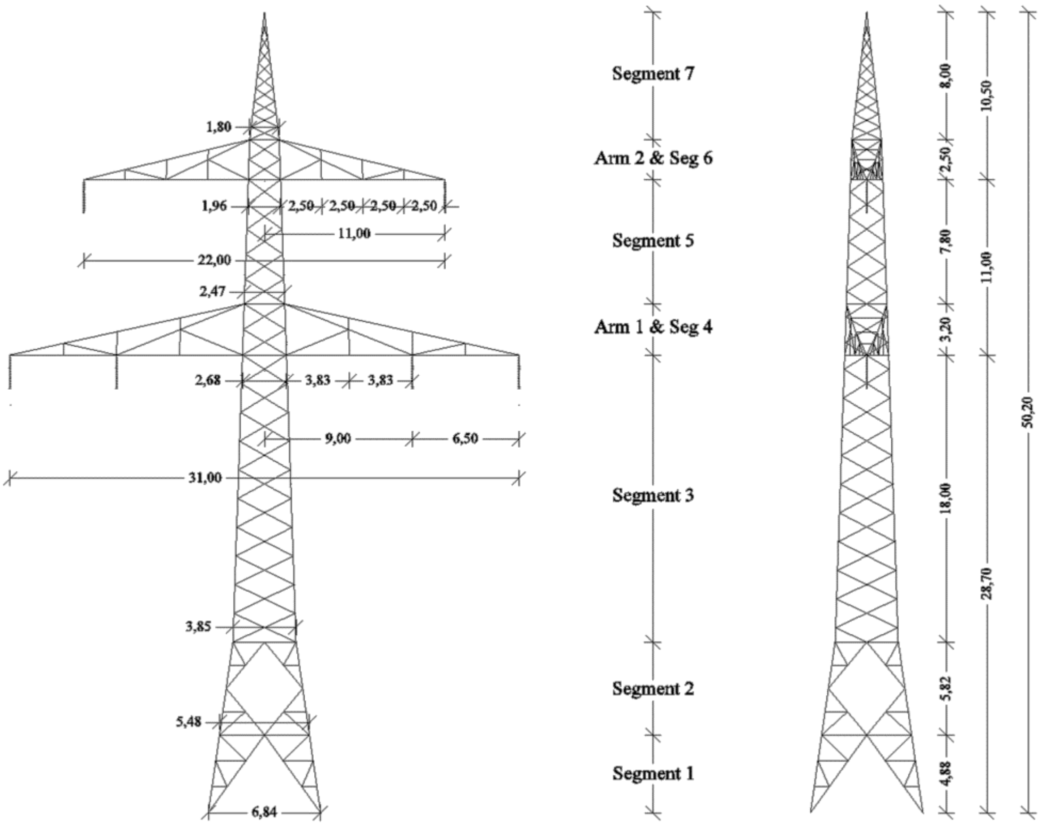

Figure 1: Geometry of the Danube tower

The tower supports two 380 kV circuits on each tower’s side. Each circuit consists of 3 phases and each phase is made of a bundle of 4 conductors. On its top, it carries one single earth wire for lightening protection. The conductors and the earth wire are made of steel fibres enveloped by several fibres of aluminium. Based on EN 50182:2001 [8], a “94-AL1/15-ST1A” and a “264-AL1/34-ST1A” have been selected for the earth wire and the conductors respectively. Each conductor is connected to a suspension insulator, which transfers the conductor loads to the cross arms. The six insulators are made of silicone rubber (Quadri*Sil Insulator / Hubbell company). The length of each insulator is about 5 m so as to ensure a safe distance between the conductors of the 380-kV line and the tower structure. In addition, the insulator weight is about 87 N and the wind area is equal to 0,150782 m^2.

The initial design has been done by TOWER finite element software, which is dedicated to design transmission and communication steel lattice towers according to different international standards. A major advantage of the software consists in its automatic optimization process. The full optimization algorithm automatically adapts the size and the steel grade of the angle profiles, to finally propose the lightest structure with, at the same time, the highest utilization degree. Here the solution has been optimized by adapting only the size of the angle profiles; the steel grade being fixed.

The tower is designed under gravity and wind loads. And, according to [2], 12 different load combinations are considered in the analysis, as well as their partial safety factors. The wind loads are based on DIN EN 1991-1-4/NA: 2010-12 [9]. The German National annex applies method 1 of [2]; the tower is so subdivided into several segments (see Figure 1) and the wind force acts at the centre of gravity of each segment. The wind loads are calculated automatically by the software and are then applied as concentrate loads at the nodes. Ice loads are only considered on the conductors, under some load combinations.

The tower has been modelled with pin-end truss members. The eccentricities of the connections are not modelled, but their influence is considered via effective non-dimensional slenderness λ_{eff} in the member buckling checks. The foundations are assumed as simple supports. At the level of a global analysis, bolts and gusset plates are not simulated. However, their self-weight is taken into account by an adjustment factor equal to 1,20 which artificially increases the dead load of the tower. The total weight of the structure exported from TOWER, which includes the weight of the angle profiles, the weight of the insulators and the weight of the bolts and gussets, equals 16,176 tons. The conductors cannot be modelled in TOWER, so their loads are calculated apart for each load combination and introduced as point loads acting on the top of the insulators. Tension loads in the conductors are not considered since it is a suspension tower in a straight line.

3 THE MATERIAL

All the members of the tower are made of steel grade S355J2. Two cases are considered in terms of material law: a linear elastic one and a non-linear perfectly plastic one. The Young's modulus, the Poisson's coefficient and the material specific weight equal respectively 210.000 MPa, 0,3 and 7850 kg/m^3. The yield stress is 345 MPa. The safety factor for the material resistance is taken as equal to 1,0.

For each element, residual stresses resulting from hot-rolling are considered in material at non-linear analyses; the pattern is shown in Figure 2. This pattern, found in many scientific papers [10], is also been used as a reference for the development of Eurocode 3 design rules.

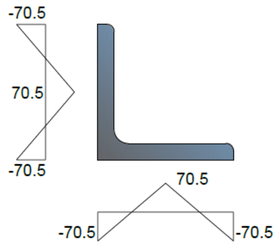

Figure 2: Residual stresses of angle cross-section

4 LOADS AND LOAD COMBINATIONS

Twelve different load combinations regarding the wind direction and the definition of the actions (favourable/unfavourable) are considered automatically in TOWER. Two of these have been here selected which correspond to unfavourable actions (for the determination of the axis, see Figure 3):

- **<u>X direction</u>**: Gravity loads (G) and wind forces perpendicular to the cross arms (W_x).
- **<u>Y direction</u>**: Gravity loads (G) and wind forces in direction of the cross arms (W_y).

The self-weight of the tower itself is calculated automatically in FINELG according to the geometry. The self-weight of the bolts and gusset plates is taken into account by an adjustment factor 1,20. The self-weight of the conductors and the earth wire, are evaluated according to EN 50182 [8]. The weight of an insulator is about 0,087 kN.

The calculation of the wind loads on the tower is based on EN 1993-3-1/Annex B/B.3.2.2.1 and EN 1991-1-4. The tower is subdivided into several segments (see Figure 1) and, for each, a mean wind load is evaluated for X and Y directions (see Figure 3).

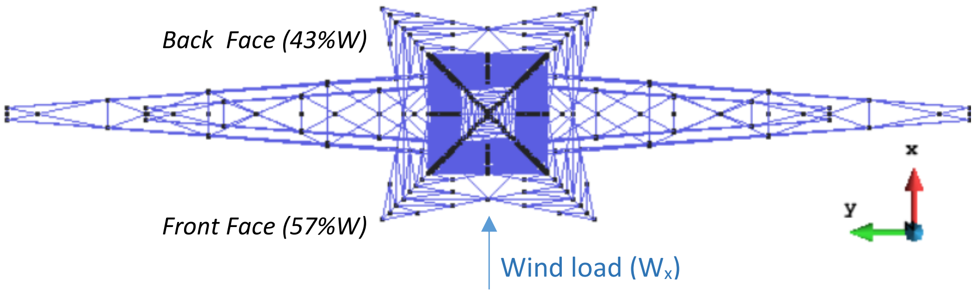

Figure 3: Definition of wind direction

Then, the mean wind load in each direction is distributed on the front and back face of the tower. The front and back faces varies obviously according to the wind direction. It is assumed that the front face of each segment is supporting 57% of the total wind load, and the back face 43% (see Figure 3). At the end, the wind load acting on a face is distributed to each bar according to its normal area, as a constant linear load along each bar, so to achieve a realistic simulation of the action. The wind loads on the conductors, the earth wire and the insulators are based on EN 1993-3-1/Annex B/B.3.2.2.4.

In the next paragraphs, the safety load factors on loads vary according to the analysis.

5 CONTENTS OF THE STUDY

The main objective of the study is to validate the initial design of the tower made through the TOWER software. To achieve this goal, an elastic analysis and a full non-linear analysis of the tower are successively made by means of the finite element software FINELG so as to check the design in terms of elastic response and resistance/stability respectively. Same safety load factors will be used for the applied loads in those two analyses (the same ones than in TOWER - specifically, in case of unfavourable actions, $\gamma_G=\gamma_W=1{,}35$ according to EN 50341-2-4).

As said in the introduction, further studies (elastic instability analysis, elastic second-order analyses) have also been achieved so as to understand better how the structure behaves. For some different reasons, the decision has been taken to perform these analyses under un-factored loads.

To summarise, the following analyses have so been performed:

- a first order linear elastic analysis with safety load factors equal to $\gamma_G= \gamma_W=1{,}35$, in order to compare both FINELG and TOWER models;
- an elastic instability analysis with safety load factors equal to 1,0;

- a second-order linear elastic analysis with safety load factors equal to 1,0, to complement the instability analysis; will be performed with and one without initial imperfections, so as to investigate their influence;
- a second order plastic non-linear analysis with safety load factors equal to 1,0, without initial imperfections, to evaluate the impact of plasticity on the tower response.
- a full non-linear (second-order effects and plasticity) analysis with safety load factors equal to $\gamma_G=\gamma_W=1{,}35$ and with initial imperfections so as to validate the initial design of the tower made by means of the TOWER software.

6 SIMULATION WITH FINELG

Beam finite elements with 7 degrees of freedom (DOF) are used in the FINELG finite element software, as plate buckling phenomena in the angle legs are not to be contemplated. The model of the tower is represented in Figure 4. It is worth noting that FINELG has been already successfully used in the past to simulate a lattice tower [11].

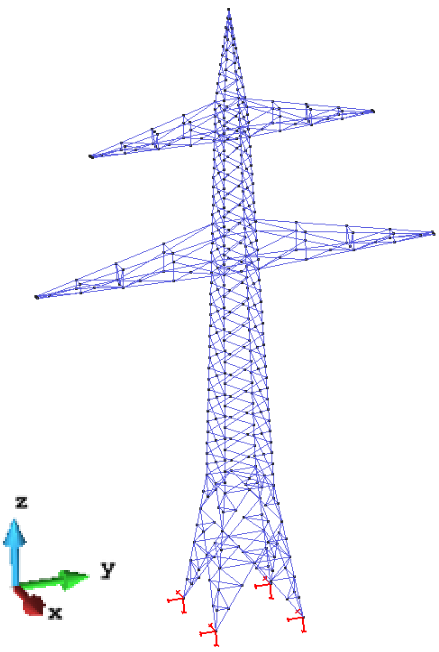

Figure 4: 3-D model of the tower with FINELG software

The bolted connections between the diagonals and tower legs as well as the splices in the tower legs are not considered in the model. However, their global response has been simulated through appropriate hinges/constraints at the ends of the elements.

In the members that are considered as continuous over their total length (main tower legs), the 7 DOF are blocked at the extremities of the finite beam elements. Besides that, the bracing members and horizontal members are considered as pinned at their ends. The secondary bracing elements are also considered as pinned at their ends. For those members, all the rotations are free, except the torsion about the beam axis which is blocked. All the other DOF are blocked too. The foundations are assumed pinned. Every element/bar is modelled with its appropriate eccentricity, rotation and orientation in order to simulate the reality as closely as possible. The wind loads on the conductors and the earth wire, as well as their self-weight, are calculated apart and entered in the model as point loads acting at the top of the insulators.

7 COMPARISON OF FINELG AND TOWER MODELS IN THE ELASTIC RANGE

Before the full non-linear analyses and the validation of the initial design of the tower, it is important to compare the model created by FINELG with the initial model built with TOWER. First of all, the self-weight of the structure has been compared to the one provided by TOWER. Then, the maximum displacements for three different load combinations have been evaluated.

As already said in paragraph 2, the total weight of the structure reported from TOWER is 16,996 tons=166,73 kN. It is reminded that the total weight includes the weight of the angle profiles, the weight of the insulators and the weight of the bolts and gussets which is estimated through a load adjustment factor of 1,2. The corresponding value in FINELG is 172,60 kN. The difference between the two models, of 3,40%, may be explained by the difference of the lengths of the members in TOWER (elements connected at the point of intersection of the member axes, at the centre of gravity) while, in FINELG, actual connection eccentricities are modelled.

Table 1: Maximum displacements for 1st order linear elastic analysis

Load comb.	Node with max displacement	Direction of displacement	Max displacement from TOWER [m]	Max displacement from FINELG [m]
1,35G	Edge of the lower arm	Z	$-8{,}14\cdot10^{-3}$	$-9{,}61\cdot10^{-3}$
1,35G+1,35W_x	Top of the tower	X	0,301	0,164
1,35G+1,35W_y	Top of the tower	Y	0,514	0,596

The maximum displacements are summarized in Table 1. It should be noted that:

- load combination 1,35G includes only the self-weight of the tower, bolts and gussets, without the conductors and insulators;
- the wind load calculations being based on different norms in TOWER and FINELG (EN 1993-3-1 and EN 50341-2-4 respectively), it is normal to see a difference between those displacements;
- the wind loads in FINELG are introduced as linear loads along the bars while in TOWER they are introduced as forces at the nodes;
- the wind loads on the body of the tower are bigger according to EN 50341-2-4 than EN 1993-3-1, what justifies the difference in load combination 1,35G+1,35W_X;
- the wind loads on the conductors are smaller according to EN 50341-2-4 than EN 1993-3-1, what explains why the difference in load combination 1,35G+1,35W_Y is smaller than in load combination 1,35G+1,35W_X.

Regarding those displacement values, one notices that they are high. However, the displacements are appearing at the failure limit state (applying loads with 1,35 load factors) and not at the service limit state (unfactored loads). This being, there is no special indication or limitation specified in the norms (EN 1993-3-1 or EN 50341-2-4) in terms of maximum displacement at service limit state. The only reason to provide displacements here is to compare the order of magnitude – not even the exact value – between the TOWER and FINELG software.

8 FURTHER INVESTIGATIONS ON THE TOWER RESPONSE

In order to investigate further the tower response, an elastic instability analysis is performed, and complemented by a second order linear elastic one. Furthermore, a second order plastic analysis is carried out to evaluate the importance of the plasticity effects. For the following analyses, load factors equal to 1,0 will be used, as explained before. Through those analyses, the influence of the initial imperfections is also investigated.

8.1 Instability analysis

It is important to notice that “instability analysis” means a first order linear elastic analysis. Specifically, the critical loads have been calculated for the load combinations $G+W_x$ and $G+W_y$. The results are summarised in Table 2 and in Figure 5 and Figure 6.

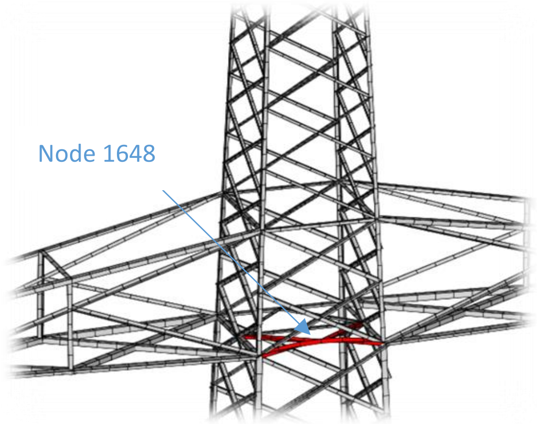

Figure 5: First member instability for load combination $G+W_x$

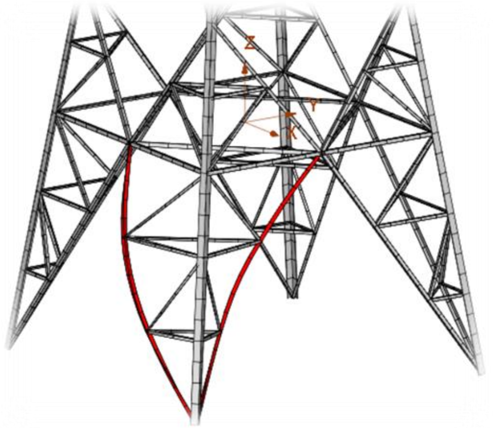

Figure 6: First segment instability for load combination $G+W_y$

Table 2: Results from elastic instability analysis

Load combination	$G+W_x$		$G+W_y$	
No of mode	Load factor λ	Type of instability	Load factor λ	Type of instability
1st	2,270	Member	1,371	Segment
2nd	2,862	Member	1,418	Member
3rd	4,256	Member	1,608	Member
4th	4,279	Segment	1,641	Member

8.2 Non-linear analyses for load combination G+W

Further to the instability analysis of the tower, three complementary analyses have been performed for the same load combination (G+W) as previously explained:

- a geometrically non-linear elastic analysis with elastic material law without initial imperfections;
- a geometrically non-linear elastic analysis with elastic material law considering initial imperfections;
- a geometrically and materially non-linear analysis with full plastic material law (no residual stresses) without initial imperfections.

The initial imperfections have been chosen in accordance with the 1st instability mode, calibrated so as to reach an amplitude of L/1000 (L is the length of the member/segment where the instability occurs). The two geometrically non-linear elastic analyses have a double role: to verify the instability analysis and to investigate also the influence of the initial imperfections. Through the third analysis, it has been decided to minimize the parameters (initial imperfections, residual stresses) in order to observe the influence of the plasticity in the structure. In the analyses, the loads are increased proportionally: λ(G+W). The results are summarized, for each direction, in the next paragraphs.

8.2.1 Load combination G+W_x

Results are reported in Figure 7. For the three types of analyses, instability occurs in the same bar (vertical displacement u_z, see Figure 4, reported in Figure 7, at node 1648, see Figure 8).

By observing, in Figure 7, the two curves relative to the geometrically non-linear elastic analyses, one concludes that the influence of the initial imperfection on the instability is rather negligible. The load factor reached for both cases is about λ=1,62. Looking to the third curve in Figure 7, the maximum load factor obtained in this case is about λ=1,52.

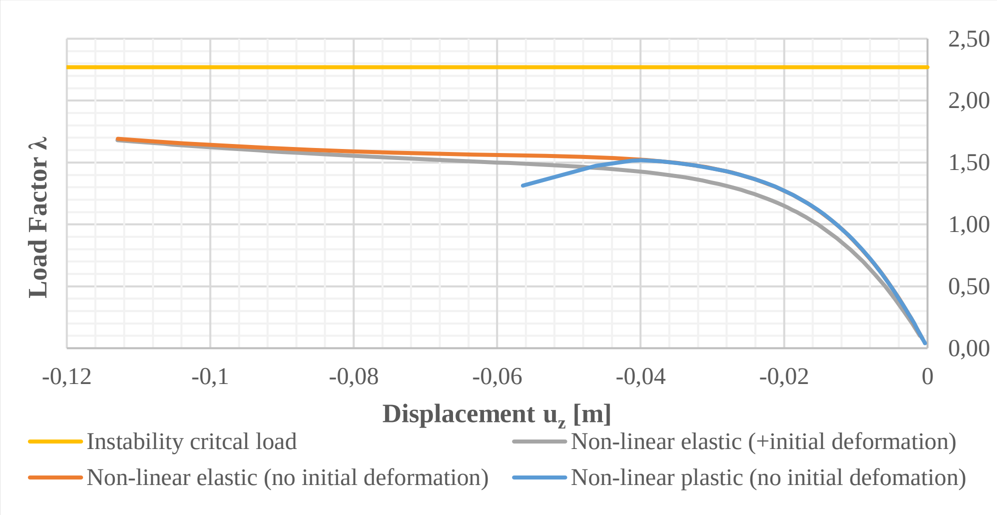

Figure 7: Displacement u_z versus load factor for different types of analyses - wind perpendicular to the arms (X direction)

It is a priori surprising to see that the critical load obtained by the instability analysis (λ=2,27) is significantly higher than the maximum load obtained by the geometrically non-linear elastic analysis (λ≈1,62). If someone checks the internal forces at node 1648 in both cases (see Table 3), realises that the failure occurs for two different triplets of relative axial force and bending moments (N, M_{by}, M_{bz}). Indeed, in the second order linear elastic analyses, the second order effects are significantly influencing the internal forces in the members. In a member with a double-symmetrical section, this would have no effect on the member critical resistance, but this is not the case for angle sections, what extra investigations on isolated members have shown. At the end, this explains that the “real” critical load is smaller than the load obtained through an instability analysis.

Table 3: Internal forces at node 1648, for the two analyses

Internal forces/Type of analysis	Instability analysis	2nd order linear elastic analysis without initial imperfection
N [kN]	-266,49	-164,10
Torsion M_t [kNm]	0,050	0,368
Bending M_{by} [kNm]	3,557	13,300
Bending M_{bz} [kNm]	-0,24	-10,90
Load factor λ	2,27	1,62

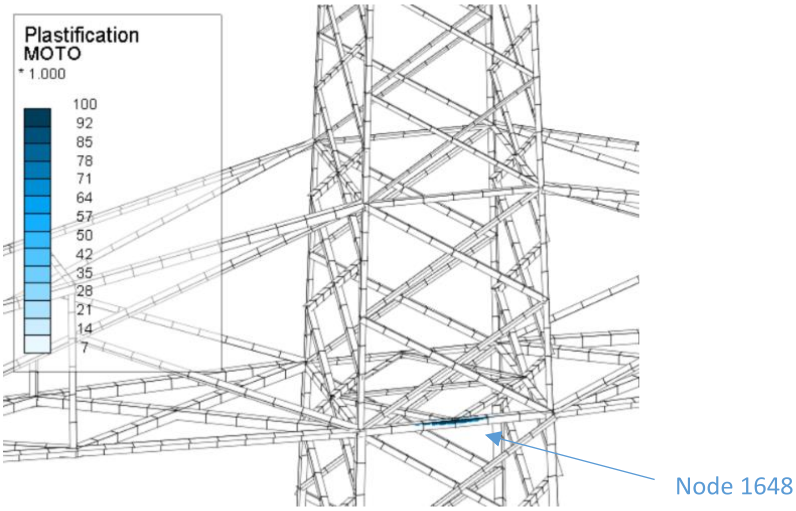

Figure 8: Results (plasticisation) from the 2nd order non-linear plastic analysis (X direction)

8.2.2 Load combination G+W_y

The behaviour is a bit different for this load combination. First, the failure does not occur only in a unique bar, but in a large number of bars. All these bars buckle in X direction, even if the applying wind loads are in Y direction (see Figure 4 for the axis). That actually happened due to the eccentricity of those bars, which creates significant bending moments. The plasticity starts from a number of lateral bars of the tower: two in the –X (bars 1 & 2) one in the +X (bar 3) global direction (front and back side in Figure 9). It is observed that the yielding in the backside bars starts simultaneously at the middle and the edges, while it starts at mid-span in the front bars.

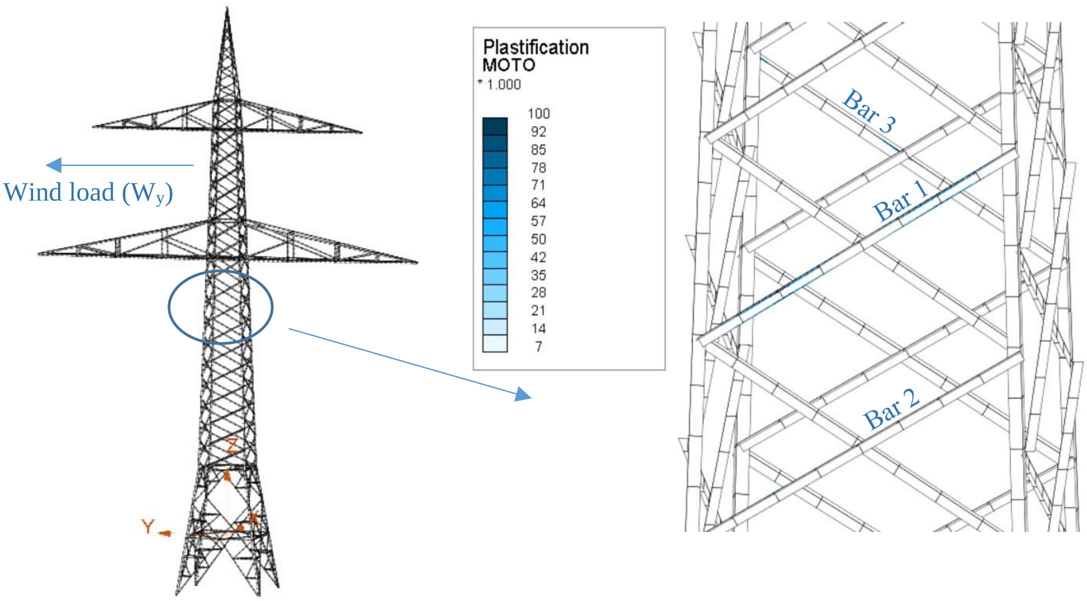

Figure 9: Results (yielding) from the 2nd order material non-linear plastic analysis (Y direction)

The graph in Figure 10 shows the horizontal displacement u_x (direction of global X axis - Figure 4) versus the load factor at the top of the tower. This node has been selected because it represents rather well the global response of the tower. It is again clear, from the two curves referring in Figure 10 to the non-linear elastic analyses, that the influence of the initial imperfection is negligible. The load factor for those cases is about λ=0,87. In addition, when material non-linearities are integrated in the analysis, the yielding starts from a number of bars and the load factor obtained in this case is about λ=0,85. Plasticity effects are therefore almost inexistent here. The difference between the critical instability load and the maximum load reached through a 2nd order linear elastic analysis could again be explained by the different loading situations (relative axial force and bending moments) in the critical bars.

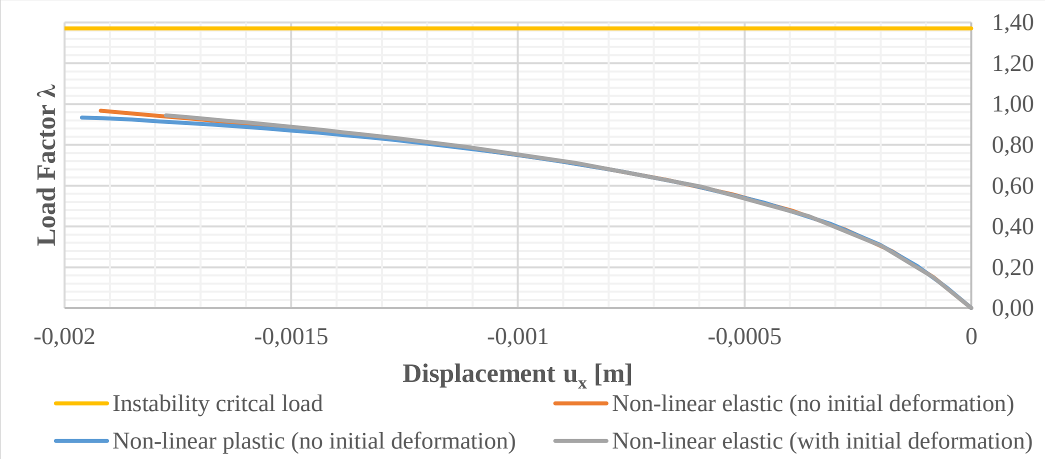

Figure 10: Displacement u_x versus load factor for different types of analyses - wind parallel to the arms (Y direction)

9 VALIDATION OF THE INITIAL DESIGN MADE THROUGH THE TOWER SOFTWARE

The validation of the initial design requires a full non-linear analysis, considering an elastic-perfectly plastic material, distributions of residual stresses (Figure 2) and an initial imperfection of the structure in accordance with the 1st instability mode (see 8.2).

The gravity loads are first applied and then wind loads are increased [1,35G+λ(1,35W)] until failure of the tower occurs. This load sequence simulation is closer to the reality. Just for sake of comparison, a 2nd order linear elastic analysis, with the same initial imperfections, is performed for this load sequence.

9.1 Load combination 1,35G+1,50W$_x$

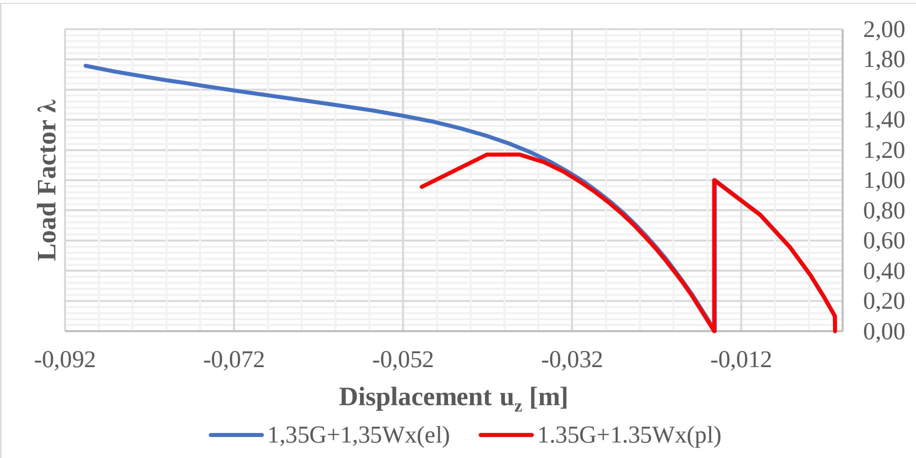

Figure 11: Displacement versus load factor for different types of analysis – 1,35G+1,35W$_x$ (X direction)

The load factor for the 2nd order linear elastic analysis is about λ≈1,62 while, for the full non-linear analysis, it is equal to λ=1,17. The failure occurs in the same bar (node 1648) than in section 8.2.1. The graph in Figure 11 represents the vertical displacement u_z (direction of global Z axis) at the node 1648 versus the load factor for the design loads for each sequence.

It is important to notice that the load factor for this load combination is bigger than 1,0 with comparison to the design factored loads. As a result, may be concluded the adequacy of the initial TOWER design. Furthermore, it is observed that the tower remains elastic for load factors λ≤1,0, so confirming the TOWER design assumptions.

9.2 Load combination 1,35G+1,50W$_y$

Figure 12 shows the horizontal displacement u_x (direction of global X axis) at the top of the tower versus the load factor. The load factor for the 2nd order linear elastic analysis is about λ≈0,64 while for the full plastic analysis it amounts λ≈0,62. Here again plasticity effects are quite negligible.

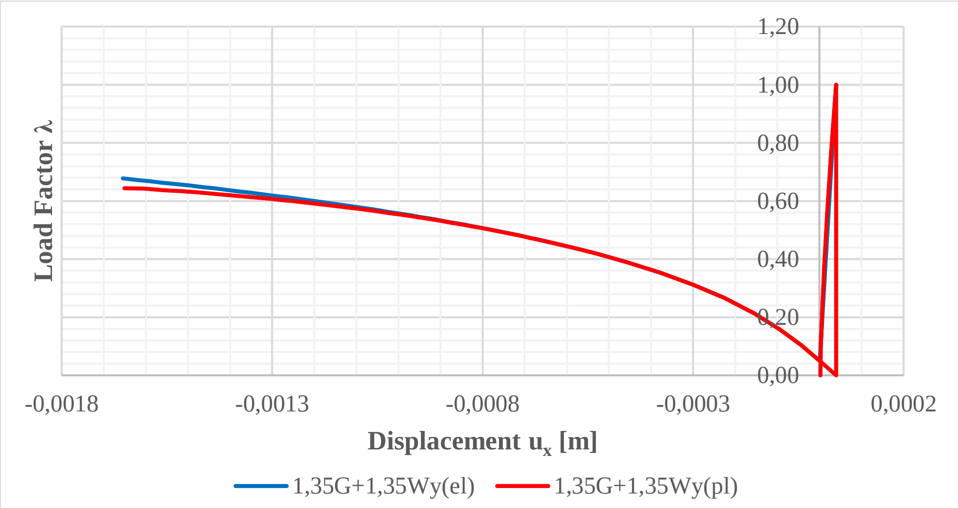

Figure 12: Displacement versus load factor for different types of analysis – 1,35G+1,35W$_y$ (Y direction)

However, contrary to what is seen under 1,35G+1,50W$_x$, the maximum load factor remains here far lower than 1,0. The initial design of the tower by TOWER software for this direction is therefore seen as insufficient and unconservative. This may be explained by the development, in reality, of an instability mode in one of the main tower legs (see Figure 6), called there “segment instability” and is not covered by TOWER, but more importantly, also not addressed by the reference normative documents to which it is referred in the introduction. Proposals for amendment of these codes are so being prepared within the ANGELHY RFCS project so to fill this gap.

10 CONCLUSIONS

From the present study, the following conclusions may be drawn:

- Two European norms exist for the design of transmission towers: EN 1993-3-1 & EN 50341-1.
- Both norms provide different recommendations. In the present report, EN 1993-3-1 has been used for evaluation of the wind loads.

- There is no special indication or limitation in the norms about the maximum displacements of the tower at serviceability limit states. This factor is so not checked, probably because of the lack of specific needs in this regard.
- A reasonable agreement is seen between FINELG and TOWER elastic analyses. The differences may be explained by modelling aspects.
- The "real" critical load obtained by a 2^{nd} order elastic analysis is smaller than the critical one obtained by an elastic instability analysis. The reason is that the forces acting on the members in both cases differ, so affecting the member buckling load in the case of non-symmetrical cross-sections. Moreover, these effects are amplified with regard to the actual member support conditions (eccentricities for instance).
- Full non-linear analyses have confirmed the lack of influence of the imperfections on the results.
- The initial design of the tower achieved by means of the software TOWER appears to be rather good in the case of application of the wind loads according to X, but quite unconservative for the application of wind loads in the Y direction. The reason is due to the development, in the second case, of a segment instability mode which is not recommended to be checked by the norms, and therefore by TOWER. Besides that, in TOWER, all the bars are simulated by truss elements, without any loading eccentricities; and the latter are considered through the use of reduced buckling length factors for members. And finally, in TOWER, the wind loads are applied at the nodes. The influence of all these parameters will be further investigated in the ongoing ANGELHY RFCS-funded research project.

REFERENCES

[1] CEN: EN 1993-3-1, Design of steel structures – Part 3-1: Towers, masts and chimneys – Tower and masts, 2006.

[2] CEN: EN 50341-1, Overhead electrical lines exceeding AC 1 kV - Part 1: General requirements - Common specifications, 2012.

[3] CEN: EN 50341-2-4, Overhead electrical lines exceeding AC 1 kV - Part 2-4: National Normative Aspects (NNA) for Germany (based on EN 50341-1:2012), 2016.

[4] TOWER User's Manual – Version1 15.0, ©Power Line Systems Inc, 2017.

[5] FINELG: Non-linear finite element analysis program, User's manual, Version 9.0, Greisch Ingeniere, 2003.

[6] CEN: EN 1991-1-4, Actions on structures - Part 1-4: General actions, Wind actions, 2005.

[7] Reinhard Fischer and Friedrich Kießling: Freileitungen: Planung, Berechnung, Ausführung. 4 Auflage, Springer Verlag, 2013.

[8] CEN: EN 50182, Conductors for overhead lines - Round wire concentric lay, 2001, 2018.

[9] CEN: EN 1991-1-4, Actions on structures - Part 1-4: General actions, Wind actions, National annex of Germany 2005.

[10] Zhang L, Jaspart JP, Stability of members in compression made of large hot-rolled and welded angles, Université de Liège, 2013.

[11] Vincent de Ville de Goyet, L'analyse statique non linéaire par la méthode des éléments finis des structures spatiales formées de poutres à section non symétrique, PhD, Université de Liège, 1989.