USE TENSION-ONLY ELEMENTS TO MODEL AND ANALYZE THE STRUCTURE OF STEEL MONOPOLE WITH TIEBACK CABLES

Nhat Dung Tran[1,*]

[1]*Le Quy Don Technical University*

Abstract

Steel monopole with tieback cables is the commonly used structure in Vietnam as well as in the world. In classical analysis models, tieback cables were modeled by tension-compression bar elements with equivalent axial stiffness, considering the deflection of the cable caused by self-weight. The results of these models did not reflect the actual working properties of structures and the physical nature of the problem. This article proposes a bar element model (only tension without pretension) to model a tieback cable structure. Numerical experiment results for a steel monopole with non-pretension tieback cables show considerable differences of displacements and internal forces between the proposed model and the classical model.

Keywords: *Steel monopole; tieback cable; tension-only element; Tower software.*

1. Introduction

1.1. Steel monopole

Steel monopole has only 01 pole. The base of pole has 01 joint flange with existing holes to connect with the foundation. Hence, steel monopoles occupy small areas, are commonly used in crowded accommodation terrain, low-voltage electrical network, lighting poles, traffic lights, antenna poles… In case of suitable terrain, steel monopoles can be reinforced by a system of tieback cables, called steel monopoles with tieback cables.

A steel monopole frequently has a narrower cross section as it goes up, the cross section shape is usually circular, annulus, regular polygons, this pole is prefabricated in the factory. The base of pole is welded to a joint flange with existing holes to connect with the foundation through bolts. This design helps steel monopoles occupy very small areas. Hence, poles are suitable for antenna poles, power poles in the city terrain, residential areas, lighting poles, traffic lights…

Compared to ubiquitous poles that are used for high-voltage, medium-voltage and

---

[*] Email: trannhatdung01@gmail.com

https://doi.org/10.56651/lqdtu.jst.v5.n01.369.sce

low-voltage line such as concrete pole, lumber pole, steel pole with tie bars, steel monopole is a new technology product with a bunch of advantages. Especially, when it comes to medium and high-voltage grid, steel monopole is almost only inevitable product that satisfies the aesthetic aspect and has the smallest occupation [1].

a) Power pole *b) Lighting pole* *c) Antenna pole*

Figure 1. Application of steel monopole with/without tieback cables.

Steel monopole requires fast construction and erection, low cost of maintenance, good lightning protection… It is more and more widely used [1, 2].

1.2. The problem of designing a steel monopole with tieback cables

A steel monopole with tieback cables is not much different from a steel monopole without tieback cables. Arranging tieback cables around the pole is a basic reinforcement method, strengthening the pole stiffness. Tieback cables are typical steel cables, one of its ends is connected to the pole, another end is connected to an anchor (which was usually buried in the ground). To make tieback cables have tensile forces, people use turnbuckles.

In calculation, tieback cables are essentially tension-only structures (having no compression capacity). However, to determine whether internal forces in tieback cables are tension or compression, the structural problem must be resolved in order to get internal forces of elements. Besides, loads impacting on the structural system are frequently separated into groups (static load, live load, wind load…), each load group causes tensile or compressive forces on the same tieback cable element [2].

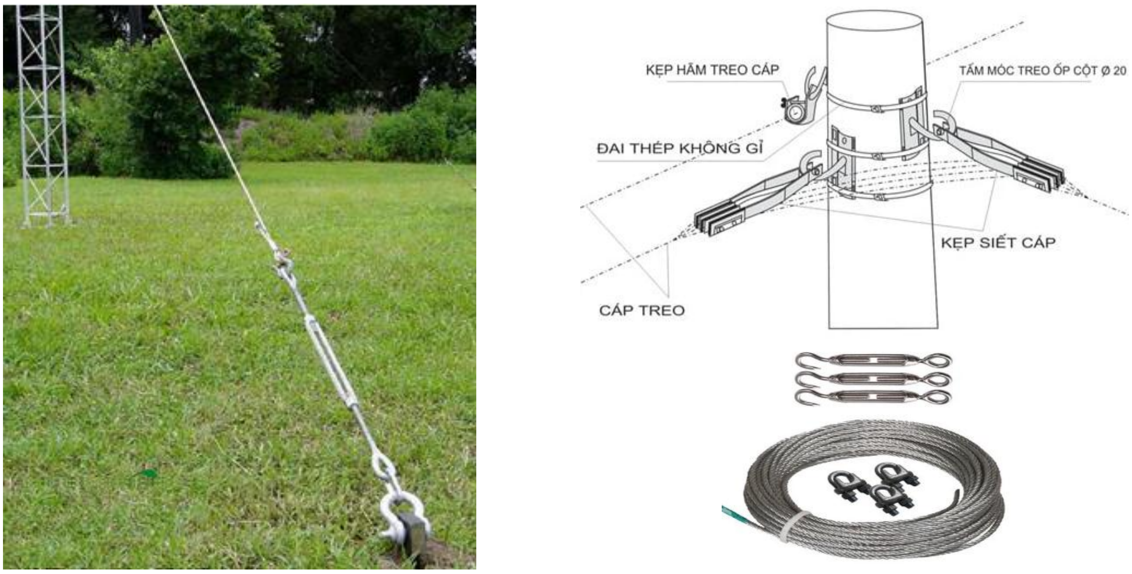


Figure 2. Tieback cable and its accessory.

From above reasons, although the calculation of steel monopole with tieback cable is not complicated in terms of structural aspect, but it is fairly difficult to calculate properly, and sufficiently. Besides, for almost all design documents of steel poles using a finite element software, tieback cable elements are considered as bar elements with nodal hinge at both ends (tension and compression are available). The fact that a piece of tieback cable is a tension-only element, having no compression capacity. It is incorrect to consider a piece of tieback cable as a bar element with nodal hinge at both ends, making calculated results inclined to be unsafe [3].

2. Solving problem of steel monopole with tieback cables using finite element software

2.1. Tension-only element (tieback cable element)

A tieback cable element is a tension-only element, it means that only tension stiffness is available, having no compression stiffness ($[\boldsymbol{k_e}] = 0$). Moreover, one special thing of a tieback cable element is that its stiffness can be changed during calculation. When calculating with load cases or load combinations causing tensile forces in tieback cables, the tieback cable stiffness is calculated as other common bar element. Conversely, if load cases or load combinations cause compressive forces, the tieback cable stiffness will be inexistent.

In a structural problem, there are commonly basic load cases and load combinations. When solving with each load case to get internal forces, the values of axial forces in tieback cables are frequently sign-changing (from tension to compression and vice versa). Hence, element stiffness matrices [$\boldsymbol{k_e}$] and the general stiffness matrix [$\boldsymbol{K}$] are also changed continuously so they require specific treatments. It can be said that tension-only element is a special element which is specifically designed for the problems having tieback cables [2, 4].

2.2. Problem solving sequence of a steel monopole with tieback cables

The pole body is considered as consecutive bar elements in three-dimensional space. The cross-section of bar is defined as annulus shape (tube cross-section). The base of pole is a fully restrained connection (or hinge), connecting to the ground. Tieback cable elements connect to the ground by hinge connections [4, 5].

Problem solving sequence of a steel monopole with tieback cables consists of following steps:

a) Build the geometric diagram (nodal, element diagram);

b) Identify geometric, material parameters;

c) Identify connections and freedom connections;

d) Identify load cases and load combinations;

e) Calculate the general stiffness matrix [$\boldsymbol{K}$] (from distributions of partial stiffness matrices [$\boldsymbol{k_e}$]);

f) Solve to get displacement, internal forces of elements as each load case;

g) From calculated results of load cases ***nNhom***, check axial force of each cable, if the value of axial force Ne > 0 (compression), some treatments will be executed as follows:

- Assign the 0 value to stiffness matrix [$\boldsymbol{k_e}$] of tieback cable element $\boldsymbol{n_i}$;

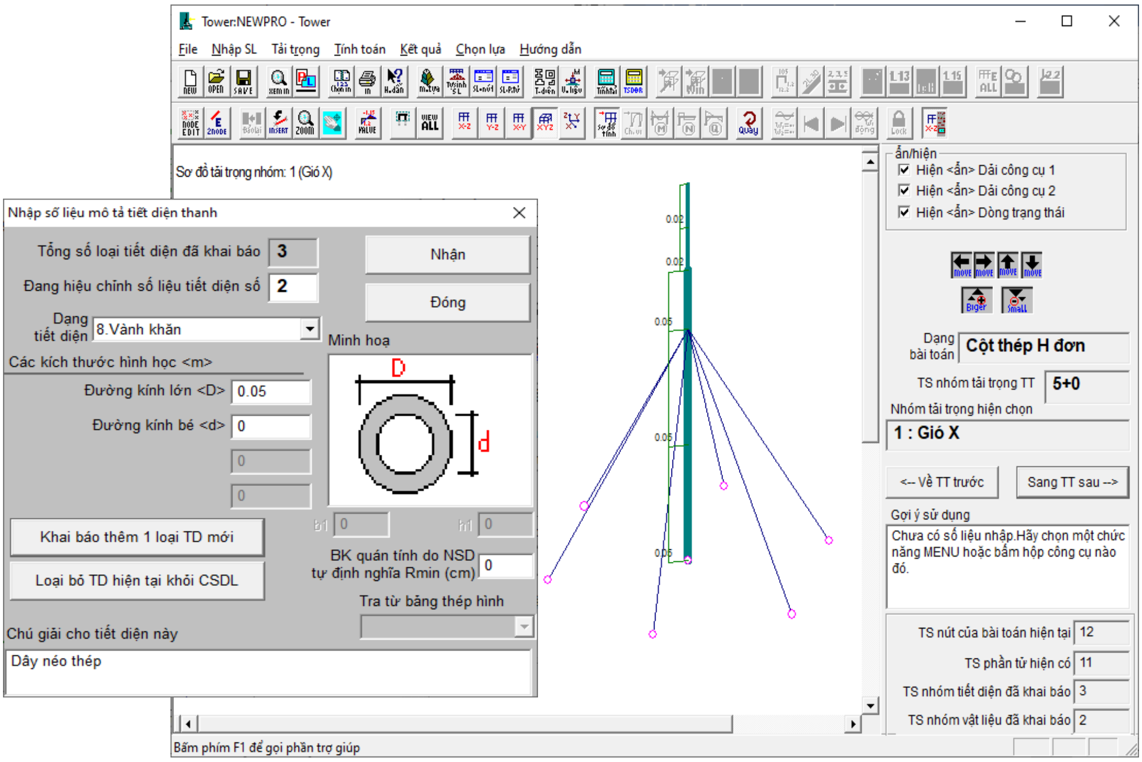
- Recalculate the general stiffness matrix (essentially remove the stiffness contribution of cable elements which have compression). Recalculate displacements and internal forces;

- Recheck axial forces of tieback cable elements, if axial forces of any tieback cable element Ne > 0 (compression) then go back to step e) - Calculate the stiffness general stiffness matrix ([**K**]).

2.3. Specialized software Tower

Solving the problem of a steel monopole with tieback cables is performed as the algorithm demonstrated in the following block diagram (Figure 4) [6, 7]:

Tower is a software used for designing and calculating steel pole structures under the type of truss bar; steel monopole with tieback cables meets requirements of TCVN [1, 2, 8, 9]. This software is designed and programmed by authors: ***Tran Nhat Dung [MTA] – Lo Ba Tho [EVN].***



Sơ đồ tải trọng nhóm: 1 (Gió X)

Nhập số liệu mô tả tiết diện thanh

Tổng số loại tiết diện đã khai báo: 3

Đang hiệu chỉnh số liệu tiết diện số: 2

Dạng tiết diện: 8.Vành khăn

Các kích thước hình học <m>

Đường kính lớn <D>: 0.05

Đường kính bé <d>: 0

0

0

b1: 0

h1: 0

BK quán tính do NSD tự định nghĩa Rmin (cm): 0

Tra từ bảng thép hình

Chú giải cho tiết diện này: Dây néo thép

☑ Hiện <ẩn> Dải công cụ 1

☑ Hiện <ẩn> Dải công cụ 2

☑ Hiện <ẩn> Dòng trạng thái

Dạng bài toán: **Cột thép H đơn**

TS nhóm tải trọng TT: **5+0**

Nhóm tải trọng hiện chọn: **1 : Gió X**

Gợi ý sử dụng: Chưa có số liệu nhập.Hãy chọn một chức năng MENU hoặc bấm hộp công cụ nào đó.

TS nút của bài toán hiện tại: 12

TS phần tử hiện có: 11

TS nhóm tiết diện đã khai báo: 3

TS nhóm vật liệu đã khai báo: 2

Bấm phím F1 để gọi phần trợ giúp

Figure 3. Main screen interface of Tower.

Tower is a finite element software, dedicated for steel pole calculation as three-dimensional space. Tower has Vietnamese interface with favorable Menu + Toolbar system, it has the ability to generate data quickly, strongly, accurately with abundant and vivid graphics. All data and calculated results from Tower can be expressed by graphical forms, files are easy to be saved or printed. Tower is used for examining a numerical testing problem in the following item 3.

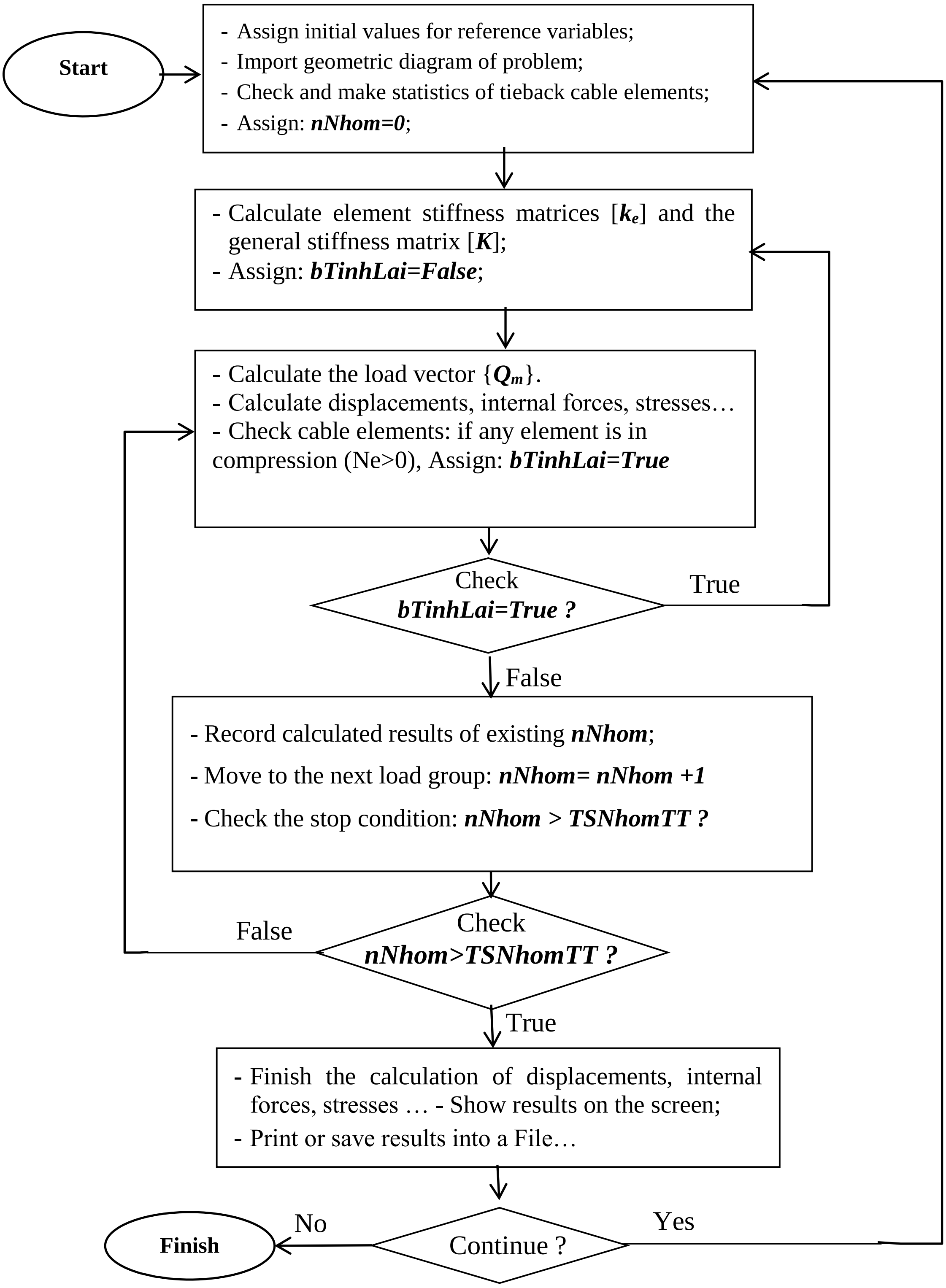


Figure 4. Block diagram of problem solving the tension-only element with specialized software.

3. Numerical testing of problem for steel monopole with tieback cables

3.1. Describe the testing construction

The testing construction is a power transmission pole under the type of steel monopole with tieback cables. This pole is 30 m high including 3 tieback cables. It is designed for installing and using in terrains of Hanoi city. Currently, Tower software has not programmed the pre-tensioned anchorage problem, so for the numerical test problem, the author has accepted to ignore the pre-tension in the anchor ropes. For numerical testing, **Tower** software is used for calculating as 2 plans:

- **Plan 1:** Designing a steel pole with tieback cables modelled by the bar elements with nodal hinge at both ends;

- **Plan 2:** Designing a steel pole with tieback cables modelled by tension-only elements;

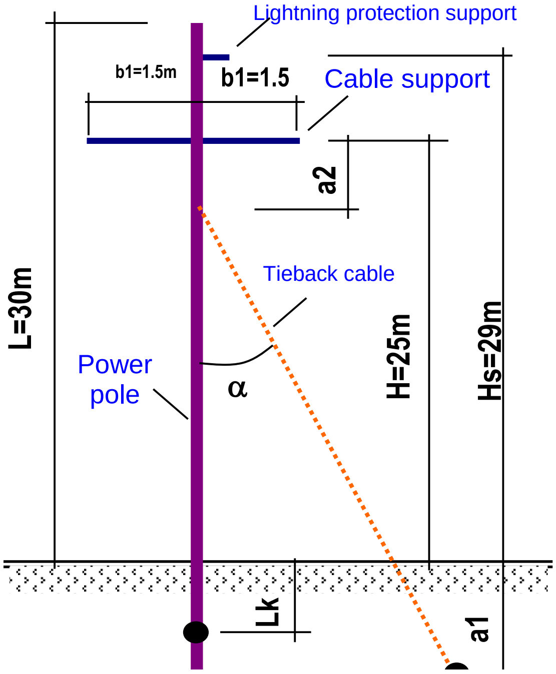
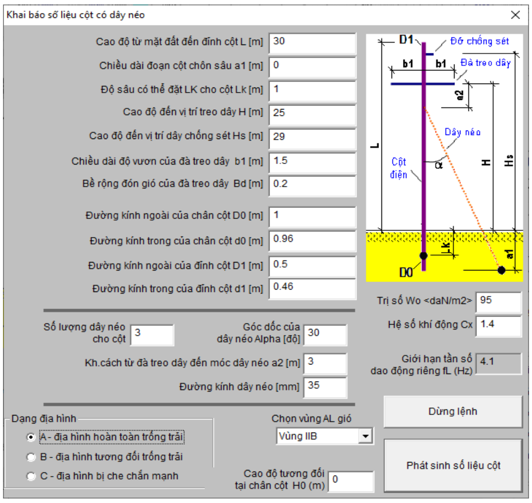


Figure 5. Main parameter of the testing contruction.

Calculated results between 2 plans are compared altogether, each plan is then evaluated and commented to drawn pros and cons [3, 6].


Khai báo số liệu cột có dây néo

Cao độ từ mặt đất đến đỉnh cột L [m] 30
Chiều dài đoạn cột chôn sâu a1 [m] 0
Độ sâu có thể đặt LK cho cột Lk [m] 1
Cao độ đến vị trí treo dây H [m] 25
Cao độ đến vị trí dây chống sét Hs [m] 29
Chiều dài độ vươn của đà treo dây b1 [m] 1.5
Bề rộng đón gió của đà treo dây Bd [m] 0.2
Đường kính ngoài của chân cột D0 [m] 1
Đường kính trong của chân cột d0 [m] 0.96
Đường kính ngoài của đỉnh cột D1 [m] 0.5
Đường kính trong của đỉnh cột d1 [m] 0.46
Số lượng dây néo cho cột 3
Góc dốc của dây néo Alpha [độ] 30
Kh.cách từ đà treo dây đến móc dây néo a2 [m] 3
Đường kính dây néo [mm] 35
Trị số Wo <daN/m2> 95
Hệ số khí động Cx 1.4
Giới hạn tần số dao động riêng fL (Hz) 4.1

Dạng địa hình
● A - địa hình hoàn toàn trống trải
○ B - địa hình tương đối trống trải
○ C - địa hình bị che chắn mạnh

Chọn vùng AL gió: Vùng IIB
Cao độ tương đối tại chân cột H0 (m) 0

Figure 6. Parameters used for generating pole and tieback cables.

3.2. Input data

Geometric data and loads of the numerical testing problem are defined under parameters (Figure 6), and then used for generating nodes, elements and load data by Tower software.

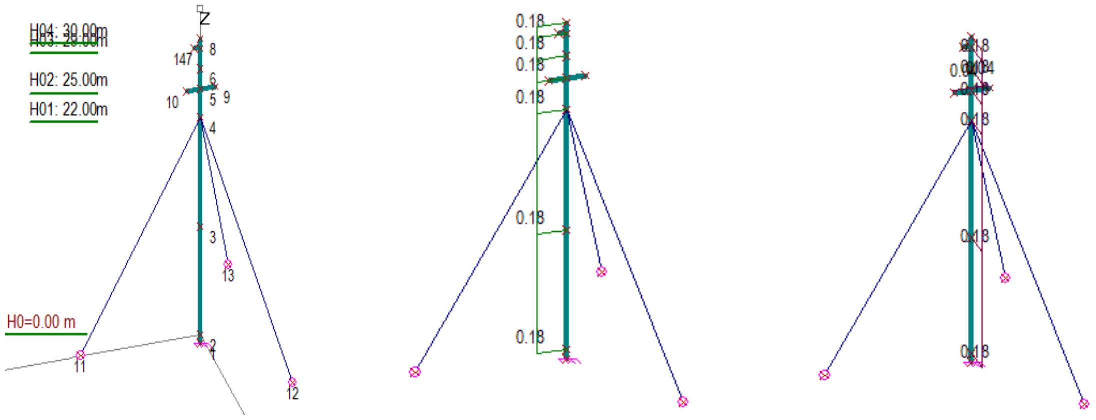
Loads and load combinations [2, 4, 7]

Basic loads of this problem are separated into 5 load cases as follows (Figure 7):

- Load case 0: Static load (self-weight of the structure);
- Load case 2: Wind load in Y direction: as TCVN 2737:1995;
- Load case 3: Sling load;
- Load case 4: Cable breaking load.

From these basic loads, 4 load combinations are defined as follows:

- TH01: 0(1.00)+1(1.00)
- TH02: 0(1.00)+2(1.00)
- TH03: 0(1.00)+3(1.00)
- TH04: 0(1.00)+4(1.00)



Nodel numbering diagram *Load case 1: Wind load X* *Load case 2: Wind load Y*

Load case 3: Sling load *Load case 4: Cable breaking load*

Figure 7. Diagram of basic load cases.

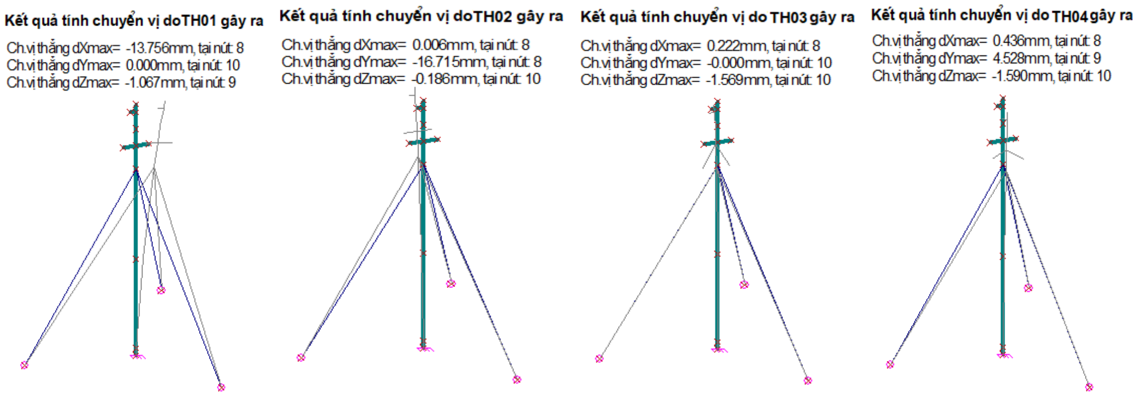
Load cases are defined in compliance with TCVN 2737:1995. Load case 0 (self weight) will be automatically calculated by the software; other loads are defined in compliance with TCVN 2737:1995 and related design standards. For simplicity, in the numerical testing problem, load combination is for illustrative purpose only, including the combinations of static load (load case 0) and load cases 1, 2, 3, 4 [7, 8].

3.3. Calculation results

When it comes to structural problems, in general, and steel monopole problems, in particular, the entire calculation document will be about several dozen or hundred pages if it is fully presented. Hence, with limited space, this article only prints brief displacements and internal forces results under following regulations:

- Only print results of load combinations;
- Only print results in brief forms;
- Combine graphical interface and tabular results.

3.3.1. Calculated results of nodal displacements in Plan 1



Kết quả tính chuyển vị doTH01 gây ra

Ch.vị thẳng dXmax= -13.756mm, tại nút 8
Ch.vị thẳng dYmax= 0.000mm, tại nút 10
Ch.vị thẳng dZmax= -1.067mm, tại nút 9

Kết quả tính chuyển vị doTH02 gây ra

Ch.vị thẳng dXmax= 0.006mm, tại nút 8
Ch.vị thẳng dYmax= -16.715mm, tại nút 8
Ch.vị thẳng dZmax= -0.186mm, tại nút 10

Kết quả tính chuyển vị do TH03 gây ra

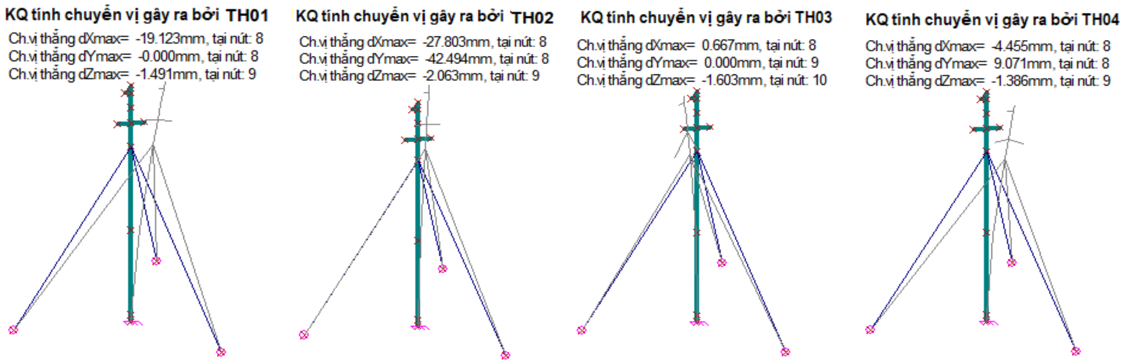
Ch.vị thẳng dXmax= 0.222mm, tại nút 8
Ch.vị thẳng dYmax= -0.000mm, tại nút 10
Ch.vị thẳng dZmax= -1.569mm, tại nút 10

Kết quả tính chuyển vị do TH04 gây ra

Ch.vị thẳng dXmax= 0.436mm, tại nút 8
Ch.vị thẳng dYmax= 4.528mm, tại nút 9
Ch.vị thẳng dZmax= -1.590mm, tại nút 10

Figure 8. Calculated results of displacements (Plan 1).

3.3.2. Calculated results of nodal displacements in Plan 2



KQ tính chuyển vị gây ra bởi TH01

Ch.vị thẳng dXmax= -19.123mm, tại nút: 8
Ch.vị thẳng dYmax= -0.000mm, tại nút: 8
Ch.vị thẳng dZmax= -1.491mm, tại nút: 9

KQ tính chuyển vị gây ra bởi TH02

Ch.vị thẳng dXmax= -27.803mm, tại nút: 8
Ch.vị thẳng dYmax= -42.494mm, tại nút: 8
Ch.vị thẳng dZmax= -2.063mm, tại nút: 9

KQ tính chuyển vị gây ra bởi TH03

Ch.vị thẳng dXmax= 0.667mm, tại nút: 8
Ch.vị thẳng dYmax= 0.000mm, tại nút: 9
Ch.vị thẳng dZmax= -1.603mm, tại nút: 10

KQ tính chuyển vị gây ra bởi TH04

Ch.vị thẳng dXmax= -4.455mm, tại nút: 8
Ch.vị thẳng dYmax= 9.071mm, tại nút: 8
Ch.vị thẳng dZmax= -1.386mm, tại nút: 9

Figure 9. Calculated results of displacements (Plan 2).

3.3.3. Calculated results of moment in Plan 1

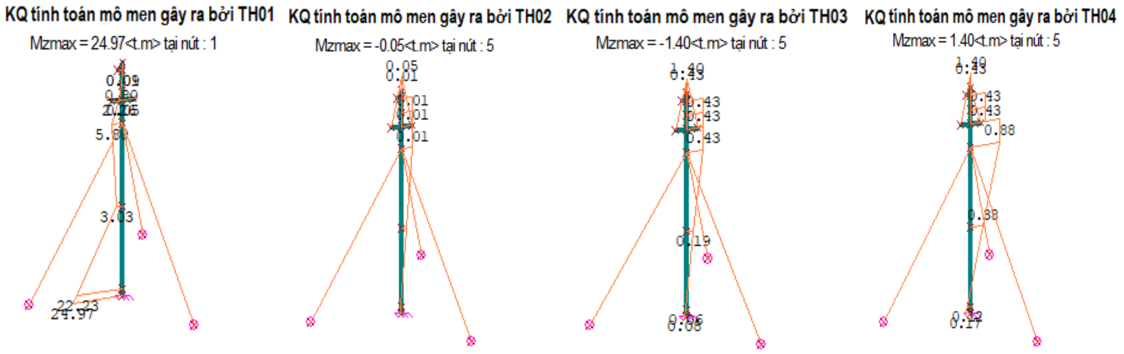


Figure 10. Calculated results of moment (Plan 1).

3.3.4. Calculated results of moment in Plan 2

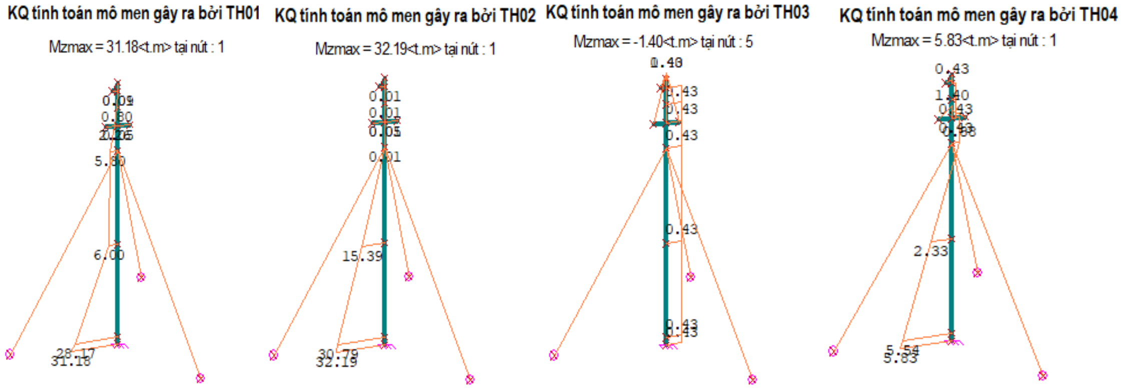


Figure 11. Calculated results of moment (Plan 2).

Table 1. Statistical calculation results of Plan 1, Plan 2 relating maximum values

Calculated results as Plan 1	**Calculated results as Plan 2**
Displacements and internal forces caused by combination TH01:(1.00)+1(1.00) dX$_{max}$= **-13.76 mm (node: 8);** dY$_{max}$= 0.00 mm (node: 10); dZ$_{max}$= -1.07 mm (node: 9) NMax= 12.038<t> at node: 1 (Element:1<1,2>) QyMax= **2.743<t> at node: 1 (Element:1<1,2>)** MzMax= **24.972<t.m> at node: 1 (Element:1<1,2>)** StressMax = 758.5<daN/cm2> at node: 4 (Element:11<4,12>) [Compressive stress]	**Displacements and internal forces caused by combination TH01:(1.00)+1(1.00)** dX$_{max}$= -19.12 mm (node: 8); dY$_{max}$= 0.0 mm (node: 8); dZ$_{max}$= -1.49 mm (node :9) NMax= 16.477<t> at node: 1 (Element:1<1,2>) QyMax**= 3.014<t> at node: 1 (Element:1<1,2>)** MzMax= 31.184<t.m> at node: 1 (Element:1<1,2>) StressMax = 599.6<daN/cm2> at node: 4 (Element:11<4,12>) [Compressive stress]

Calculated results as Plan 1	Calculated results as Plan 2
Displacements and internal forces caused by combination TH02:0(1.00)+2(1.00) dX_{max}=0.01 mm (node: 8); dY_{max}= **-16.72 mm (node: 8);** dZ_{max}=-0.19 mm (node :10) NMax= 12.038<t> at node: 1 (Element:1<1,2>) QyMax= 0.036<t> at node: 5 (Element:9<5,10>) MzMax= 0.054<t.m> at node: 5 (Element:8<9,5>) StressMax = 1460.5<daN/cm2> at node: 4 (Element:12<4,13>) [Compressive stress]	**Displacements and internal forces caused by combination: TH02:0(1.00)+2(1.00)** dX_{max}= **-27.80 mm (node: 8);** dY_{max}= **-42.49 mm (node: 8);**dZ_{max}=**-2.06 mm (node:9)** NMax= **17.116<t> at node: 1 (Element:1<1,2>)** QyMax= 1.400<t> at node: 1 (Element:1<1,2>) MzMax= **32.192<t.m> at node: 1 (Element:1<1,2>)** StressMax = 599.6<daN/cm2> at node: 4 (Element:12<4,13>) [Compressive stress]
Displacements and internal forces caused by combination TH03:0(1.00)+3(1.00) dX_{max}=0.22 mm (node: 8); dY_{max}= 0.0 mm (node: 10); dZ_{max}= -1.57 mm (node:10) NMax= **14.364<t> at node: 1 (Element:1<1,2>)** QyMax= 0.936<t> at node: 5 (Element:9<5,10>) MzMax= 1.404<t.m> at node: 5 (Element:8<9,5>) StressMax = 390.9<daN/cm2> at node: 9 (Element:8<9,5>) [Compressive stress]	**Displacements and internal forces caused by combination TH03:0(1.00)+3(1.00)** dX_{max}=-0.67 mm (node: 8); dY_{max}= 0.00 mm (node: 8); dZ_{max}=-1.60 mm (node: 10) NMax= 14.665<t> at node: 1 (Element:1<1,2>) QyMax= 0.936<t> at node: 5 (Element:9<5,10>) MzMax= 1.404<t.m> at node: 5 (Element:8<9,5>) StressMax = 599.6<daN/cm2> at node: 4 (Element:10<4,11>) [Compressive stress]
Displacements and internal forces caused by combination TH04:0(1.00)+4(1.00) dX_{max}=0.44 mm (node: 8); dY_{max}= 4.53 mm (node: 9); dZ_{max}**= -1.59 mm (node:10)** NMax= 14.074<t> at node: 1 (Element:1<1,2>) QyMax= 0.936<t> at node: 5 (Element:9<5,10>) MzMax= 1.404<t.m> at node: 5 (Element:9<5,10>) StressMax = **1563.1<daN/cm2> at node: 9 (Element:8<9,5>) [Compressive stress]**	**Displacements and internal forces caused by combination TH04:0(1.00)+4(1.00)** dX_{max}= -4.46 mm (node: 8); dY_{max}=-9.07 mm (node: 8); dZ_{max}=-1.39 mm (node :9) NMax= 15.377<t> at node: 1 (Element:1<1,2>) QyMax= 0.936<t> at node: 5 (Element:9<5,10>) MzMax= 5.832<t.m> at node: 1 (Element:1<1,2>) StressMax = **1563.1<daN/cm2> at node: 9 (Element:8<9,5>) [Compressive stress]**
Note: dX_{max} - maximum displacement in X direction; dY_{max} - maximum displacement in Y direction; dZ_{max} - maximum displacement in Z direction; N_{Max} - maximum axial force; Qy_{Max} - maximum shear force in 0y direction; Mz_{Max} - maximum moment in 0z direction; $Stress_{Max}$ - maximum stress	

Table 2. Comparison of calculated results between Plan 1 and Plan 2 at several typical nodes

No.	Internal forces, stresses element	Node, element	Unit	Plan 1	Load comb	Plan 2	Load comb	Differ (%)
1	Displacement dX_{max}	Node 8	mm	-13.76	TH01	-27.80	TH02	202.03
2	Displacement dY_{max}	Node 8	mm	-16.72	TH02	-42.49	TH02	254.13
3	Displacement dZ_{max}	Node 10	mm	-1.59	TH04	- 2.06	TH02	129.56
4	Bending moment M_z	Node 1, Element 1	T.m	24.97	TH01	32.19	TH02	28.91
5	Axial force N_e	Node 1, Element 1	T	14.36	TH03	17.12	TH02	19.22
6	Shear force Q_y	Node 1, Element 1	T	2.743	TH01	3.014	TH01	9.88
7	Stress σ_e	Node 9, Element 8	daN/cm^2	1563.1	TH04	1563.1	TH04	0

4. Comment on test results

- Displacements and internal forces in concerned positions indicate increasing changes when calculated as Plan 2, especially displacement of elements. This is suitable when it comes to the regulation of structural mechanics.

- Comparing the calculated results of Plan 1 to that of Plan 2 shows that maximum displacements and maximum internal forces computed by Plan 2 (with tension-only elements) are larger. The most significant rise is the horizontal displacement dY_{max} (up to 254%); internal forces of element (M_z, N_e, Q_y) change less than displacements (from 9.88% to 28.91%); only stress σ_e is a constant (same value of 1563.1 daN/cm^2). The reason is that σ_e is the result of TH04, it is the load combination of static load + cable breaking load (Figure 7), they are all vertical loads causing no compression in tieback cables. There is no compression in tieback cables so the tieback cable element is similar to the bar element with nodal hinge at both ends.

5. Conclusion

- Establishing the algorithm and programming by use of tension-only element for modelling tieback cable as demonstrated above is reasonable, it draws acceptable results of internal forces and stresses.

- The numerical testing problem with the use of tension-only elements shows that the model of tieback cable element is reasonable and essential for designing pole structures with tieback cables.

References

[1] Phạm Văn Hội, *Kết cấu thép công trình dân dụng và công nghiệp*, Nxb Khoa học và Kỹ thuật, 2005.

[2] Nguyễn Việt Hùng… "Nguyên lý làm việc cột trụ dây co áp dụng trong tính toàn thiết kế phân tích độ ổn định công trình cột lắp máy gió," *Tạp chí Khí tượng thủy văn*, 02/2018.

[3] TCVN 5575-2012: Kết cấu thép - Tiêu chuẩn Thiết kế.

[4] Nguyễn Quốc Bảo, Trần Nhất Dũng, *Phương pháp phần tử hữu hạn lý thuyết và lập trình*, Nxb Khoa học và Kỹ thuật, 2012.

[5] David J. Kruglinski, *Inside Visual* C^{++}, Microsoft Press, 1995.

[6] Đoàn Định Kiến, *Thiết kế kết cấu thép*, Nxb Xây dựng, 2004.

[7] TCVN, Tải trọng và tác động - Tiêu chuẩn thiết kế TCVN 2737-1995, Nxb Xây dựng, 2005.

[8] K. Bathe, E. Wilson, *Numerical Methods in Finite Element Analysis*, New Jersey, 1976.

SỬ DỤNG MÔ HÌNH PHẦN TỬ CHỈ CHỊU KÉO ĐỂ MÔ HÌNH HÓA VÀ PHÂN TÍCH KẾT CẤU CỘT THÉP ĐỘC LẬP CÓ DÂY NÉO

Trần Nhất Dũng

Tóm tắt: *Cột thép độc lập có dây néo là dạng kết cấu được sử dụng khá phổ biến ở Việt Nam và trên thế giới. Trong các mô hình cổ điển kết dây được mô hình hóa bằng phần tử thanh chịu kéo nén có độ cứng EF tương đương để xét đến độ võng của dây do trọng lượng bản thân, kết quả của mô hình này chưa phản ánh đúng thực tế làm việc của kết cấu và bản chất vật lý của bài toán. Bài báo đề xuất mô hình phần tử thanh (chỉ chịu kéo bỏ qua lực căng trước) để mô hình kết cấu dây néo. Các kết quả thử nghiệm số cho cột thép đơn có dây néo không căng trước cho thấy sự khác biệt đáng kể về chuyển vị và nội lực của mô hình đề xuất so với mô hình cổ điển.*

Từ khóa: Cột thép đơn thân; dây néo; phần tử chỉ chịu kéo; phần mềm Tower.

Received: 02/04/2022; *Revised:* 08/06/2022; *Accepted for publication:* 20/06/2022

❑