SET UP THE ALGORITHM AND CALCULATING PROGRAMME OF ZAMIL STEEL FRAME WITH PRESCRIBED DISPLACEMENT

Nhat Dung Tran[1,*]
[1]*Le Quy Don Technical University*

Abstract

This article presents the design introduction of the Zamil frame steel structure. Building algorithms and programming specialized software in order to solve the problem of prescribed displacement (or the differential settlement problem) of the Zamil frame type. Using specialized software to study and test numerically the design problems of the Zamil steel frame structure, including prescribed displacement at the foot of the columns. Therefore, compare the results of calculating the internal force and stress of the structure between the original and after taking into account this load. From there, the author suggests highly valuable comments to help design the type of this structure.

Keywords*: Zamil steels frame; column; rafter; prescribed displacement; Vn3DPro software.*

1. Introduction

Steel structures, especially Zamil-style steel frames, have gained more popularity around the world, including Vietnam. Steel structures have proven to be very suitable for factory buildings with large ground areas and large spans because they have convenient features such as quick installation, reasonable pricing, and high durability... In terms of design calculation, the Zamil frame steel structure is quite explicit and easy to design and verify, easy to fabricate, easy to erect and replace [1].

A Zamil frame steel building generally has two main parts: the foundation, and the steel frame. The foundation is usually made of reinforced concrete, processed and fabricated on-site; steel frame consists of many components (columns, rafters, braces...). These components are usually prefabricated at the factory, then transported to the construction site and erected in the designed position. Because Zamil steel frames often span large spans, few columns, column foundations are usually designed as single foundations, so after a period of use, non-uniform settlement between the foundations will appear (differential settlement). Differential settlement is disadvantage for the steel structure, it causes the structure to fall into an undesirable prestressed state. This is also one of the reasons why many Zamil frame structures have problems with shaking, even collapse, even though the external load is still quite small compared to the design load [2, 3].

* Email: trannhatdung01@gmail.com https://doi.org/10.56651/lqdtu.jst.v4.n02.348.sce

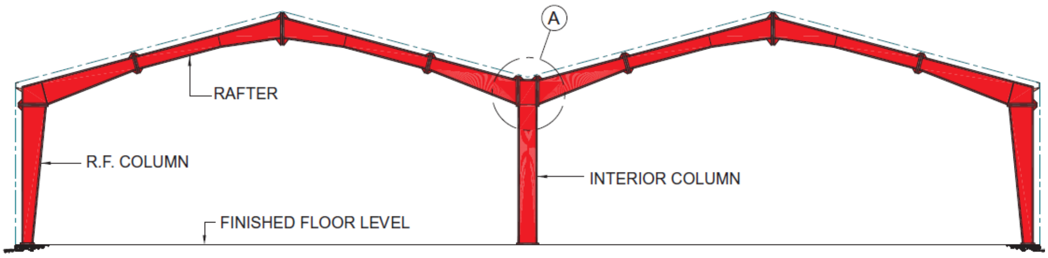


Figure 1. Typical Zamil steel frame.

2. Algorithm and programming problem of Zamil frame steel structure with prescribed displacement

2.1. Converting prescribed displacement to nodal load

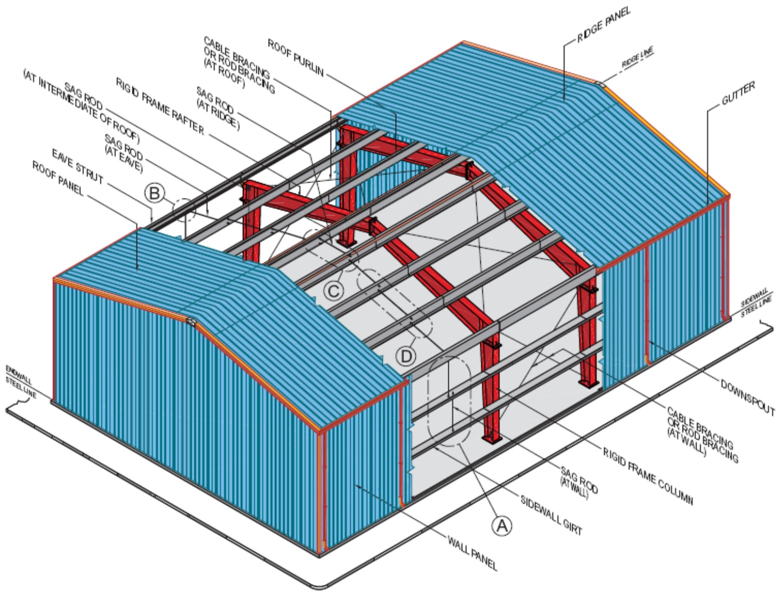


Figure 2. Typical workshop with Zamil steel frame.

2.1.1. Elemental stiffness matrix [k_m] and global stiffness matrix [K]

In fact, prescribed displacement is also a type of load (connection load). The value and direction of this load are determined through two parameters, the connection stiffness and the forced displacement value.

The essence of solving structural problems with finite element method (FEM) software is to solve a system of balanced equations of the elements in a global coordinate system.

$$[K].\{d\} = \{Q\} \quad (1)$$

where **[K]** is global stiffness matrix; **{d}** is connection displacement vector; **{Q}** is connection payload vector.

The global stiffness matrix **[K]**, built based on the calculation and accumulation of the stiffness matrix of the bar element **[$\mathbf{k_m}$]** with size 12 × 12, is determined by formula (2). The generalized spatial bar stiffness matrix is obtained by accumulating the stiffness matrix of the bar elements.

$$[K_m]=\begin{bmatrix} \frac{EF}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EF}{L} & 0 & 0 & 0 & 0 & 0 \\ & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} & 0 & \frac{-12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\ & & \frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 & 0 & 0 & \frac{-12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 \\ & & & \frac{GI_p}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GI_p}{L} & 0 & 0 \\ & & & & \frac{4EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 \\ & & & & & \frac{4EI_z}{L} & 0 & \frac{-6EI_z}{L^2} & 0 & 0 & 0 & \frac{4EI_z}{L} \\ & & & & & & \frac{EF}{L} & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\ & & S & Y & M & & & & \frac{12EI_y}{L^3} & 0 & \frac{-6EI_y}{L^2} & 0 \\ & & & & & & & & & \frac{GI_p}{L} & 0 & 0 \\ & & & & & & & & & & \frac{4EI_y}{L} & 0 \\ & & & & & & & & & & & \frac{4EI_z}{L} \end{bmatrix} \qquad (2)$$

2.1.2. Element's connection load vector {Q_m}

The load acting on the element can have many different forms such as a concentrated force or moment acting in a certain direction, the point of application of the force some distance from the left connection. The load can also be a length-distributed load acting on all or part of the element. These element loads are referred to as concentrated loads acting on the element connection.

Apply the general nodal load vector formula, when the surface force acts on an area S on the element surface:

$$\{Q_m\}=\int_S [N]^T\{P\}dS \qquad (3)$$

In case the load is a concentrated force at the connection (connection load), this load is added directly to the general load vector **{Q}**.

With the problem prescribed displacement, actually the load caused by prescribed displacement to the structure is also the concentrated load at the connection. There is a computational numerical treatment of the force ($\mathbf{P_i}$), which is determined only after the global stiffness matrix **[K]** has been built and calculated. The calculated value of $\mathbf{P_i}$, determined by formula (4), is as follows:

$$\mathbf{P_i = K_i \times \Delta_i} \qquad (4)$$

where $\mathbf{P_i}$ is equivalent connection load; $\mathbf{K_i}$ is stiffness of degrees of freedom corresponding to prescribed displacement; $\mathbf{\Delta_i}$ is prescribed displacement value.

2.1.3. Calculate internal force in element

The general formula for calculating the stress in an element according to the FEM has the form:

$$\{\sigma\} = [C][B]\{d\} \tag{5}$$

With the bar system, the program needs to give the internal force (or stress) at the connections of the element for different design purposes. The internal force components at each connection correspond to the connection displacements components in the element coordinate system.

2.1.4. The order of converting prescribed displacement into nodal load

The order of performing the calculation of the nodal load of the bearing forced displacement problem includes the following steps [1, 3]:

a) Declare the connection name, prescribed displacement direction (X, Y, Z according to the global coordinate system); the value of the prescribed displacement;

b) Calculate the stiffness matrix of each element $\mathbf{[k_m]}$ and the global stiffness matrix **[K]** for the structural system;

c) Calculate the connection load vector **{Q}** (according to the global coordinate system);

d) From the global stiffness matrix **[K]**, determine the stiffness value of the degrees of freedom corresponding to the current prescribed displacement;

e) Calculation and conversion of equivalent connection load value of displacement $\mathbf{\Delta_i}$;

f) Assign the value $\mathbf{P_i}$ to the nodal load vector **{Q}**;

g) Solve structural problems using FEM software to get results of displacement, internal force, stress [1, 3, 4]…

2.2. Programming the problem prescribed displacement of Zamil steel frame

Each element has its own stiffness and when calculating the global stiffness matrix, they contribute the stiffness to the global stiffness matrix (**[K]** matrix).

With 3D frame elements, each elements has 02 nodes, each node has 06 degrees of freedom (03 straight degrees and 03 degrees of rotation). In the FEM, when calculating the element stiffness matrix $\mathbf{[k_m]}$ (size 12×12), we can determine which index of the matrix belongs to node I (first node) and which index represents node J (end node) of element [4].

The essence of the problem of calculating the prescribed displacement is to determine the connection with the direction, and value of prescribed displacement;

convert to point loads (and assign to group 0: TLBT). Then proceed to calculate displacement, internal force, stress as a common problem [5-7].

• *Algorithm and block diagram of the program*

Algorithm to solve the prescribed displacement of Zamil frame is implemented according to the algorithm described in the following calculation diagram [8]:

2.3. Specialized software Vn3Dpro

• *Program organization*

Vn3DPro is built and programmed according to the following block diagram [4, 9]. Self-made Vn3DPro software has been used to guide and defend successfully on many master theses and doctoral theses. To check the accuracy of the software, the author has chosen to compare starting from simple problems, (problems with analytic solutions), and finallyc a few space frame problems, solved from SAP software. The results are almost identical. But because of the limited scope of the paper, this comparison could not be presented in the paper as it would increase the number of pages of the paper.

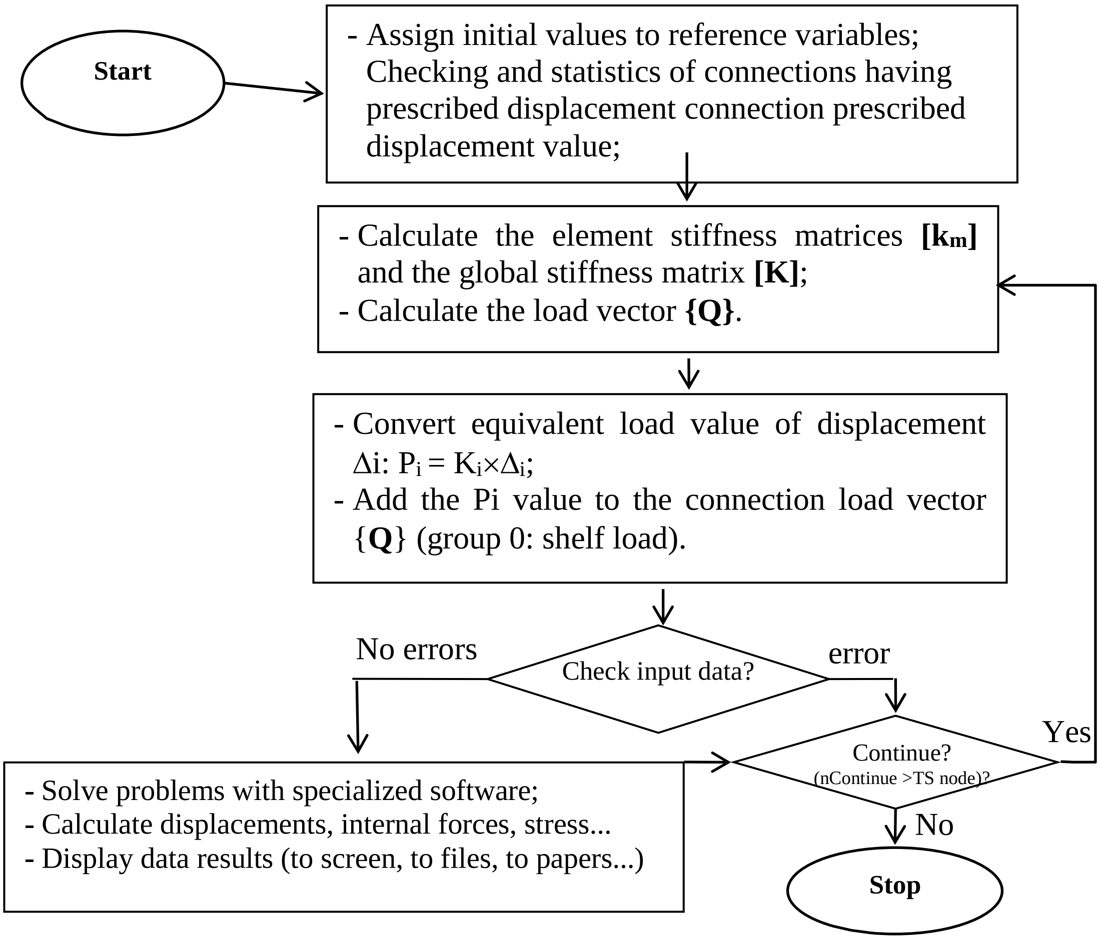


Figure 3. Block diagram for solving the prescribed displacement problem.

• ***Programming specialized software Vn3DPro***

From the above algorithm diagram, to build Vn3DPro software, choose the Microsoft Visual C^{++} programming language. This is an advanced programming language with high processing speed, flexible and diverse dynamic memory allocation capabilities. C^{++} is also a language with inheritance and multiple correspondence, these are outstanding features, helping programmers to develop applications in depth, easily in upgrade them, and add new features.

The entire source file of Vn3DPro software is quite large (about over 100000 command lines), cannot be presented in the framework of the article. Therefore, the article would like to quote an illustration of the source code that converts prescribed displacement into point loads and accumulates this value into the load vector of the problem [4, 9].

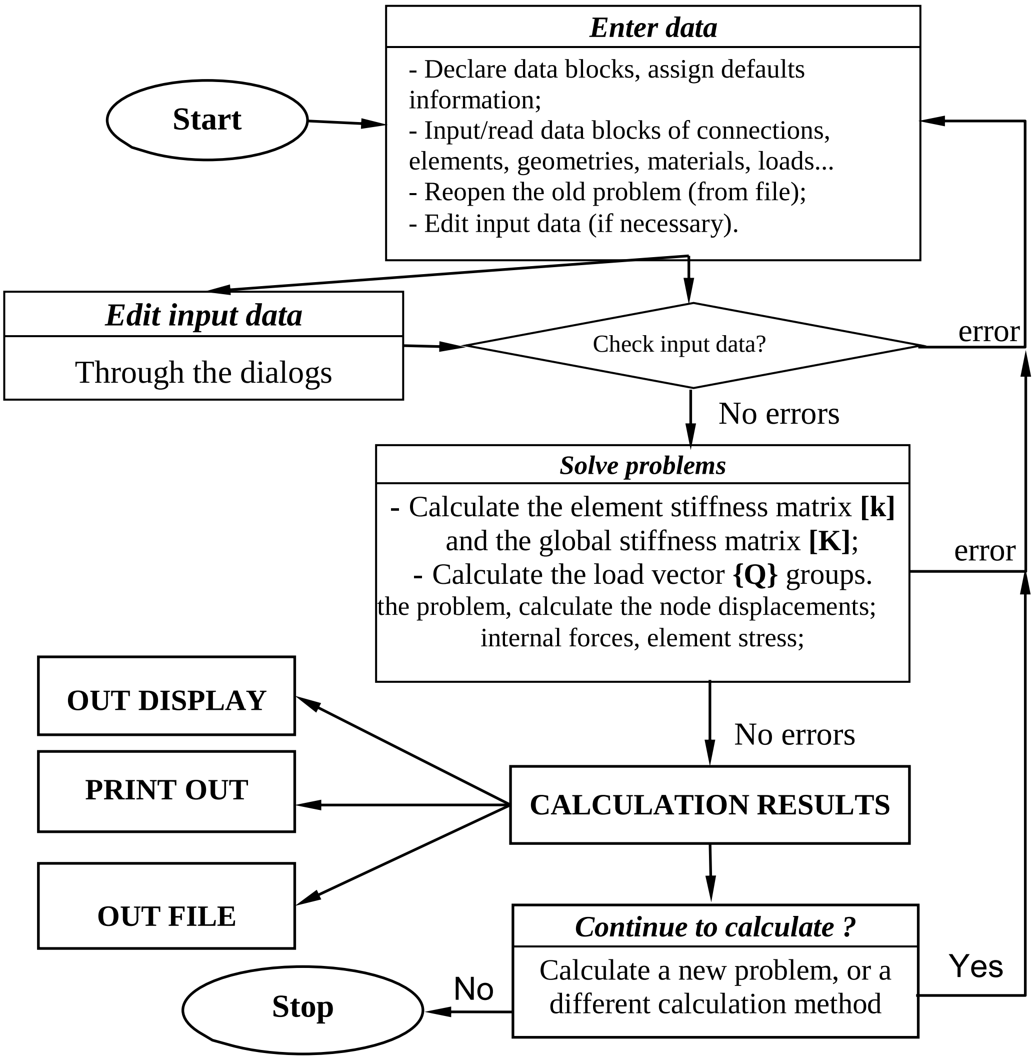


Figure 4. Block diagram of Vn3DPro software.

```
struct ConnectionDKB
{
  int Name;
  int MaCDich[6];
  double dCDCB[6];// prescribed displacement value corresponding to the degrees of freedom of the connection
};          // CONVERT PRESCRIBED DISPLACEMENT INTO CONNECTION LOAD AND ASSIGN TO LOAD VECTO.
void ChuyenDichCB()    // Calculate and integrate prescribed displacement into stiffness matrix
{ double dLoad;
  for (int i=0;i<nDKB;i++)
  { if(N_DKB[i].Name>0 && N_DKB[i].Name<=TsNut)
    {   int k = nOldToNew[N_DKB[i].Name-1];
        for (int j=0;j<6;j++)
        {   if(N_DKB[i].MaCDich[j]==1 && fabs(N_DKB[i].dCDCB[j])>0)
            {    dLoad=SK[NDS[NDF[k][j]]]*N_DKB[i].dCDCB[j];
                 if(NDF[k][j]>-1)   VecLoad.CAddPt(NDF[k][j],0,dLoad);
            }
            if(N_DKB[i].MaCDich[j]==2 && fabs(N_DKB[i].TaiTgDg[j])>0)
            {   dLoad=N_DKB[i].TaiTgDg[j];
                 if(NDF[k][j]>-1)  VecLoad.CAddPt(NDF[k][j],0,dLoad);
            } } } } }
```

Vn3DPro is a specialized software to solve structural problems by mathematical method. The software is directly designed and programmed by the author of the article. In this paper, Vn3DPro is also used to perform numerical test problems [3, 4, 5].

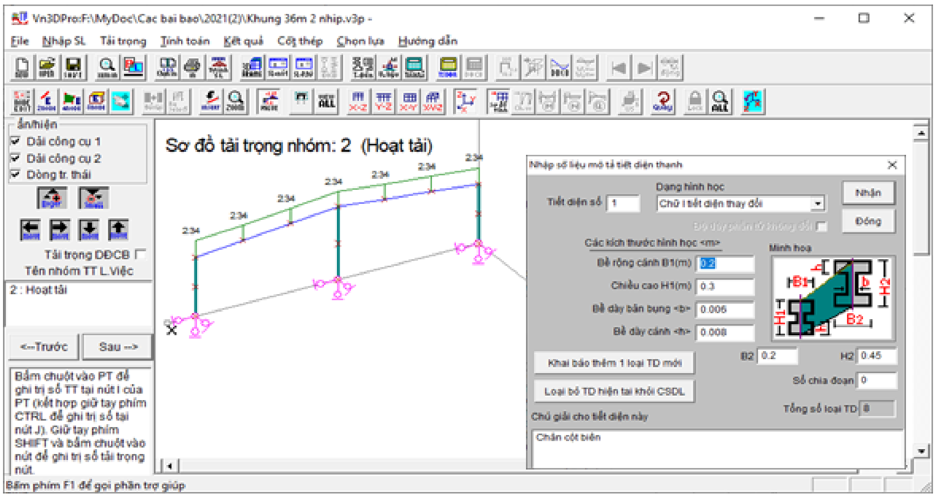


Figure 5. Main interface screen of Vn3DPro software.

3. Numerical test to evaluate the influence of prescribed displacement on internal force and stress of Zamil steel frame

3.1. Description of experimental works

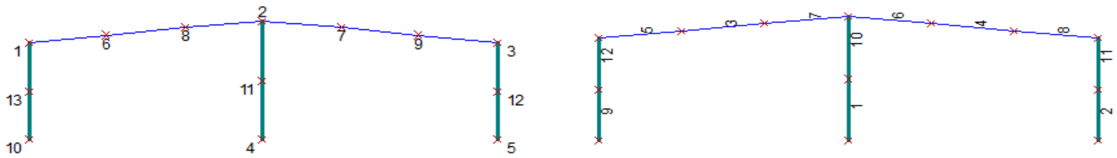
The numerical test in the article is a Zamil steel frame with three columns, a span of 36 m, a column height of 8.50 m, and a roof slope of 10%. The project was built in the North Thang Long industrial park. Frame columns and trusses are composite steel, made from construction steel. The column connects to the reinforced concrete foundation through 04 bolts ϕ 24, when making the calculation diagram, the bolts connecting the foot of the column to the foundation can be considered as a fixed bearing connection. The numerical test content will calculate the steel frame structure with the following 02 problems:

- **Problem 1:** Compute the frame without including the prescribed displacement problem;

- **Problem 2:** Calculating the frame when the prescribed displacement problem is included;

The calculation results of the two problems will be compared, evaluated and made necessary recommendations for similar works [4, 6, 10].

3.2. Problem 1: Compute the frame without including the prescribed displacement problem



a) Connection numbering diagram *b) Element numbering scheme*

Figure 6. Diagram of node numbering, number of elements.

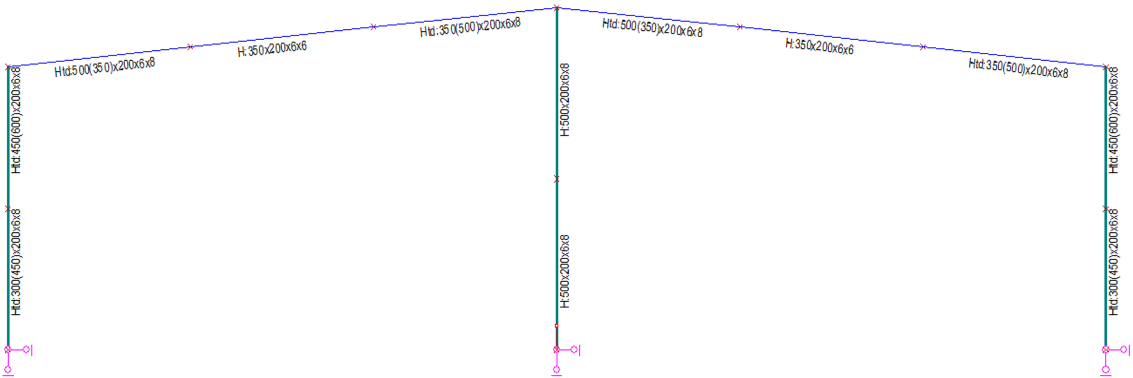


Figure 7. Cross section of columns and trusses.

3.2.1. Determine the load and make a calculation diagram

a) Groups of basic loads acting on columns: include 05 groups:

▪ Group 0: Self-weight (self-calculated software);

▪ Group 1: Static loads;

▪ Group 2: Live loads;

▪ Group 3: Wind pushing left;

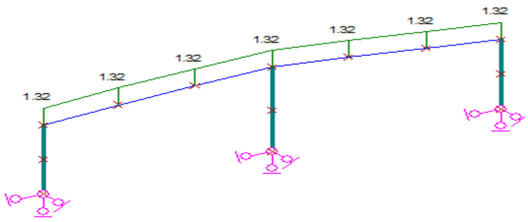
▪ Group 4: Wind pushed right.

The calculated values of the loads are determined according to TCVN 2737:1995, table 1 below is an illustration of the wind load calculation for the test work [4, 10]:

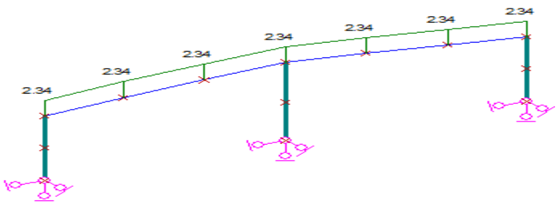
Table 1. Calculation of wind load according to TCVN 2737:1995

	Calculate the wind load according to the formula: **W=Wo.K.C.γ**					
1)	***W_0- look up the wind partition table (Hanoi [IIB])***	**95.00**	***kG/m²***			
	Geometric Dimensions (h1; L; b)	h1(m)	L(m)	b (m)	h1/L	b/L
		8.50	**66.00**	**36.00**	***0.13***	***0.55***
	Roof slope	**10%**	=	***5.71***	***degree***	
2)	***K - altitude coefficient***					
	Terrain: Terrain A (empty)					
	At the base of the column	***1.00***				
	At the top of the column	***1.15***				
3)	***C - Aerodynamic coefficient***					
	Put on the face to catch the wind on the wall (Ce)	***0.80***				
	Place it on the wall air intake (Ce3)	***-0.40***				
4)	***γ- Reliability factor***	***1.20***				
5)	***Loading distance (m) [frame span]***	**6.00**	**m**			
6)	***Calculated wind load value placed on the frame***					
	Column bottom (windward face)	**547.20**	**daN/m**	Average value		
	Column top (windward side)	**627.64**	**daN/m**	**587.42**		
	Column bottom (air intake side)	**-273.60**	**daN/m**			
	Column top (air intake side)	**-313.82**	**daN/m**	**-293.71**		
	Roof truss (windward side)	**-76.45**	**daN/m**			
	Roof truss (air intake side)	**-313.82**	**daN/m**			

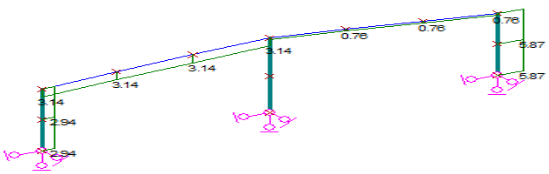
Load group 0: Self-weight will be automatically calculated by the software; other groups are determined according to TCVN 2737:1995 and entered directly. Combined load is calculated including combination of dead load (group 0) with group 1, group 2, group 3, group 4.



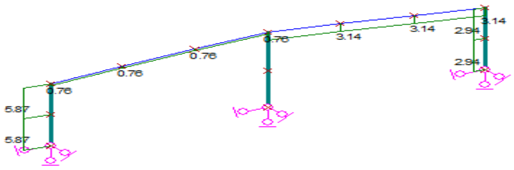
a) Group 1: Static loads



b) Group 2: Live loads



c) Group 3: Wind pushing left



d) Group 4: Wind pushed right

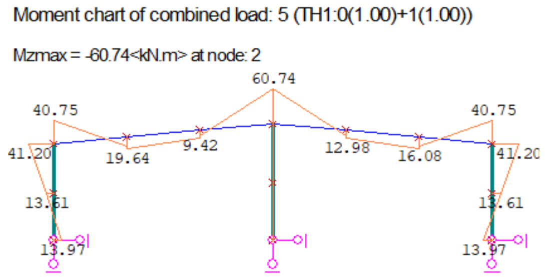
Figure 8. Calculation diagram of basic load groups.

b) Declare the combined loads

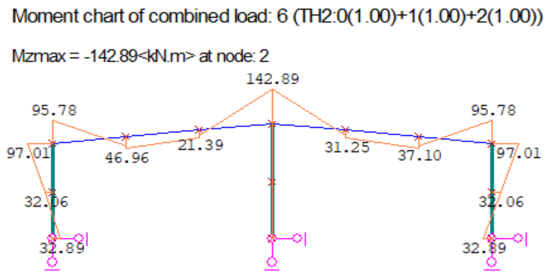
Problem 1 and problem 2 have the same declaration, with 04 groups of Combined Loads as follows [6, 8, 9]:

- TH1: 0(1.00)+1(1.00);
- TH2: 0(1.00)+1(1.00)+2(1.00);
- TH3: 0(1.00)+1(1.00)+3(1.00);
- TH4: 0(1.00)+1(1.00)+4(1.00).

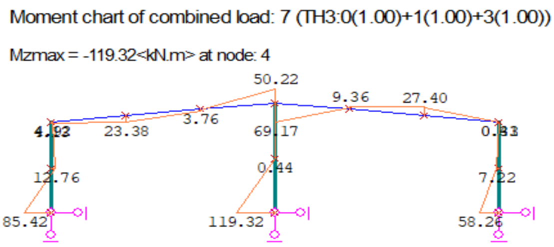
3.2.2. Calculation results



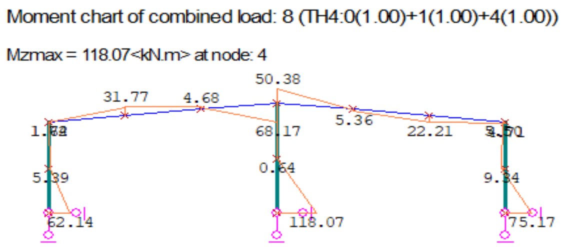
a) Moment diagrams caused by TH1



b) Moment diagrams caused by TH2

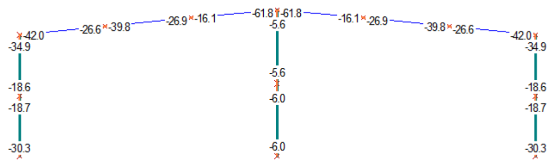


c) Moment diagrams caused by TH3

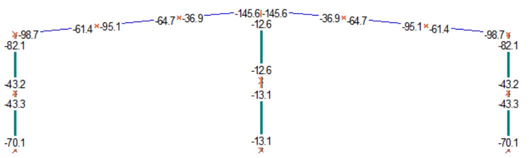


d) Moment diagrams caused by TH4

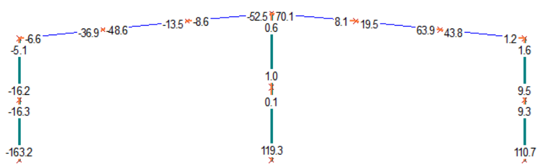
Figure 9. Moment diagrams caused by Combined loads (Problem 1).



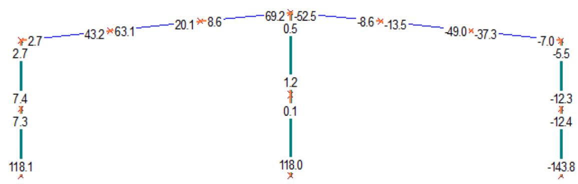
a) Stress caused by TH1



b) Stress caused by TH2



c) Stress caused by TH3



d) Stress caused by TH4

Figure 10. Stress graphs caused by Combined loads (Problem 1).

3.3. Problem 2: calculating the frame when the prescribed displacement problem is included

3.3.1. Determine the load and make a calculation diagram (similar to problem 1)

Problem 2 has essentially the same input data as Problem 1. The loads and load combinations are essentially the same. The only difference is the group 0 (Self-weight) load of Problem 2, which is added to the converted load component caused by prescribed displacement. With the distance between the boundary column and the middle column is 18 m, the maximum allowable differential settlement is (0.002 ÷ 0.003)×L = 0.036 m ÷ 0.054 m. For problem 2, we choose the middle column with differential settlement (prescribed displacement) = 3 cm is an acceptable value [3, 4]:

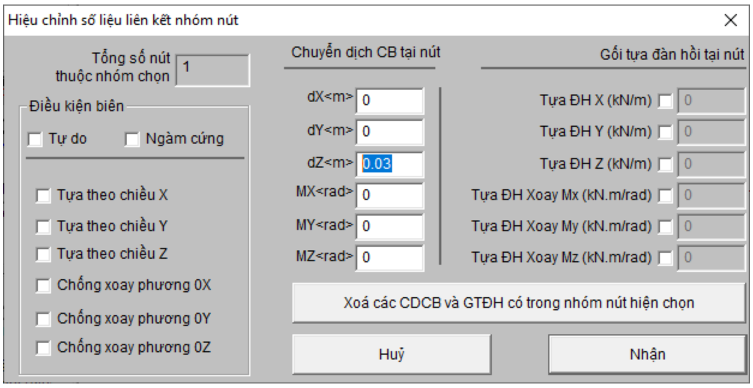
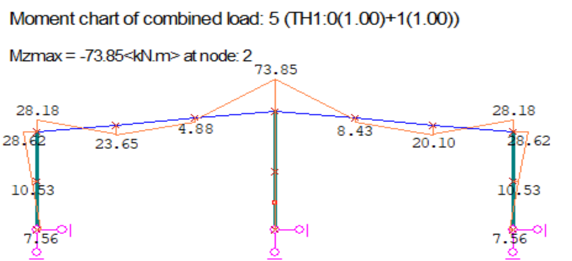
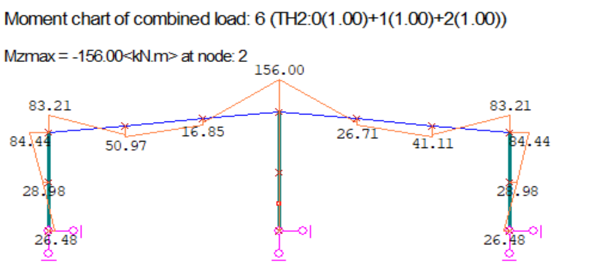


Figure 11. Prescribed displacement declaration (Δi =3 cm) for the middle column.

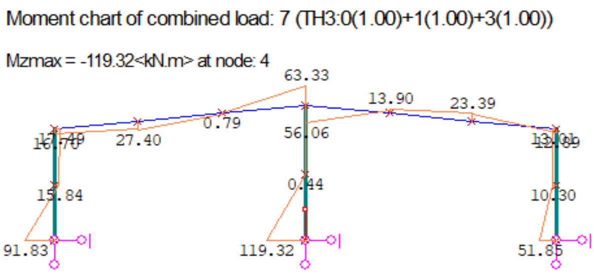
3.3.2. Calculation results



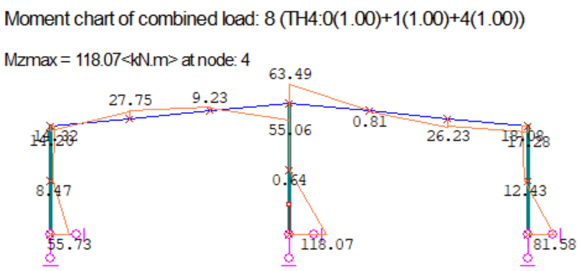
a) Moment diagrams caused by TH1



b) Moment diagrams caused by TH2

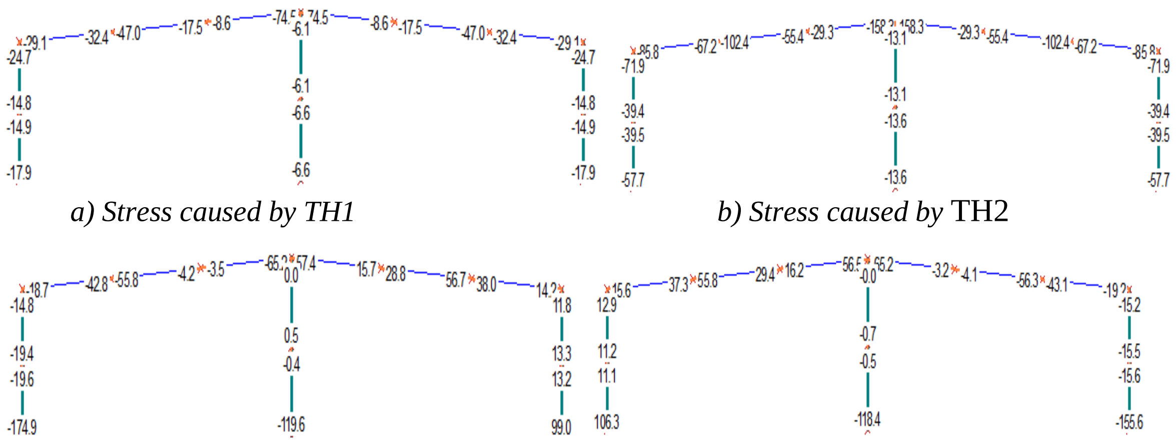


c) Moment diagrams caused by TH3



d) Moment diagrams caused by TH4

Figure 12. Moment diagrams caused by Combined loads (Problem 2).



a) Stress caused by TH1 *b) Stress caused by* TH2

c) Stress caused by TH3 *d) Stress caused by TH4*

Figure 13. Stress graphs caused by Combined loads (Problem 2).

Table 2. Comparison of calculation results at some typical locations

No	Internal force, and stress	At the node, element	Unit	Problem 1	Load group	Problem 2	Load group	Difference (%)
1	Moment M_z	node 2, ele. 6	kN.m	142.89	TH3	156.00	TH2	9.17
2	Axial force N_e	node 4, ele. 1	kN	79.66	TH2	82.96	TH2	4.14
3	Shear force Q_y	node 2, ele. 7	kN	24.04	TH4	36.99	TH2	53.87
4	Stress σ_e	node 10, ele. 9	MPa	163.2	TH3	174.90	TH3	7.17

4. Comments

- Internal force and stress at the comparison positions both increase when prescribed displacement is included (Problem 2). This is in accordance with the rules of structural mechanics.

- When taking into account the effect of prescribed displacement, the maximum value of internal force and stresses change by different levels. The shear force Qy witnesses a maximum increase of 53.87% while the axial force Ne grows slightly by 4.14%.

5. Conclusion

- Prescribed displacement problem of Zamil steel frame buildings is a scientific and practical research content. The construction of algorithms and programming for the prescribed displacement bearing problem as presented in the article is reasonable and the results of internal force and stress calculation are acceptable [2, 5].

- Numerical experiment with the problem with and without the supporting prescribed displacement shows that in the design of Zamil frame steel structures, the calculation including the supporting prescribed displacement is necessary.

- Due to the limited scope of the article, the test only presents the case with 1 footer connection (middle column) with prescribed displacement, in reality it is quite possible to appear cases where many columns have the same prescribed displacement as support much more complex problem. Then the degree of deviation of internal force and stress will be even higher.

References

[1] Phạm Văn Hội, "Kết cấu thép công trình dân dụng và công nghiệp", *Nxb KH&KT*, 2005.

[2] Technical Manual, "Zamil steels, Saudi Arabia Pre-Engineered Buildings Division", 1999.

[3] TCVN "Thiết kế, gia công lắp ráp và nghiệm thu kết cấu thép", *Nxb Xây dựng*, 2000.

[4] Nguyễn Quốc Bảo, Trần Nhất Dũng, "Phương pháp phần tử hữu hạn lý thuyết và lập trình", *Nxb KH&KT*, 2012.

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

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

[7] Robert Sedgewick, "Cẩm nang thuật toán", *Nxb Hà Nội*, 1996.

[8] Zienkiewicz O. C. and Taylor R. T. "The Finite Element Method", *Butterworth – Heinemann*, 2000.

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

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

XÂY DỰNG THUẬT TOÁN VÀ LẬP TRÌNH BÀI TOÁN CHUYỂN DỊCH CƯỠNG BỨC GỐI TỰA KẾT CẤU THÉP KHUNG ZAMIL

Trần Nhất Dũng

Tóm tắt: *Bài báo trình bày bài toán thiết kế kết cấu thép khung Zamil. Dựng thuật toán và lập trình phần mềm chuyên dụng để giải bài toán chuyển dịch cưỡng bức gối tựa (hoặc bài toán lún không đều của kết cấu nhà xưởng), kiểu khung Zamil. Sử dụng phần mềm chuyên dụng để nghiên cứu, thử nghiệm số với bài toán thiết kế kết cấu khung thép Zamil có kể đến chuyển dịch cưỡng bức tại chân cột. Qua đó, so sánh kết quả tính nội lực và ứng suất của cùng một cấu kiện khi có hoặc không tính đến chuyển dịch cưỡng bức tại chân cột. Từ đó có các nhận xét, đánh giá cho các kết cấu công trình tương tự.*

Từ khóa: Khung thép Zamil; cột; kèo; chuyển dịch cưỡng bức; phần mềm Vn3DPro.

Received: 01/11/2021; *Revised:* 11/12/2021; *Accepted for publication:* 28/12/2021

❑