

# Structural Analysis and Computer Modelling of Steel Frame Base on Stiffness Method

Zhengyang Wang \*

Faculty of Engineering, The University of Sydney, Sydney, Australia

\* Corresponding Author Email: zwan6519@uni.sydney.edu.au

**Abstract.** The main purpose of this paper is to analyze and calculate the structure using the stiffness method to obtain various structural data. At the same time, different loading conditions, including wind load, self-weight, and live load, are considered to assess their impact on the structure's behavior, with wind load being the most critical factor. In the paper, simulations are performed through calculations to evaluate these effects. Subsequently, the structure will be designed and optimized in accordance with relevant Australian standards. All the coefficients and calculation formulas mentioned in the paper can be referenced from the standards cited. All calculation and modeling processes will be carried out in MATLAB to simplify the design steps and quickly obtain design results, by calculate the capacity of each material, and facilitates the selection of appropriate materials for the structure's members. This enables automated analysis and design, meeting the safety and serviceability requirements of the structure.

**Keywords:** Stiffness method; Modeling; Member capacity; Steel structure; Wind pressure.

## 1. Introduction

The primary challenge in designing and analyzing steel structures lies in identifying the critical factors that affect floor design across various geometries and material properties, especially under diverse loading conditions. This requires in-depth research into how these structures respond to different stresses and how their components interact under a range of conditions.

Compared to the traditional force method, the stiffness method offers a more efficient approach for designing and analyzing structures. The force method has limitations when it comes to handling geometric nonlinearity and frames with complex deformation conditions, which are crucial factors in the analysis of truss and frame structures. In contrast, the stiffness method can more effectively address these complexities [1]. In comparison to the ground structure method, the stiffness method focuses on analyzing the deformation and internal forces of a given structure. Meanwhile, the ground structure method is advantageous for developing the optimal topology of a structure by systematically removing unnecessary elements from a predefined set of potential nodes with fixed locations, while retaining members that share similar properties. This approach helps in achieving an efficient structural design by refining the layout to use only essential elements. However, since the nodal locations are unable to vary during this procedure, it is necessary to work with as many nodes and members as possible, which would lead to large number of design variables, to deal with this situation with many unknown's values and a huge computational work, stiffness method can deal with this problem better by developing a transformed matrix for each element [2].

The structural design problem has been refined by introducing a compliance constraint, which means it now only considers axial forces in the members as the primary state variable. However, this simplification is not applicable to the compliance minimization problem in frame structures. This is because the stiffness matrix of a structure with fixed joints differs significantly from that of a structure with pin joints, making the transformation between these two types of joint conditions incompatible for such optimization problems [3]. Therefore, the stiffness method requires adjustments in different scenarios, needing further discussion to achieve the more accurate results, the following research

shows how to applied stiffness method in real design condition with assistance with computer modeling, and achieve a more efficient and accurate analyses method during design.

## 2. Structure Analyses

This section introduces the use of the stiffness method combined with integration techniques to analyze steel frames, aiming to evaluate the structural behavior of the steel frame under different loading conditions and provide data support for the subsequent structural design process. The stiffness method is chosen because of its high accuracy and efficiency in handling complex and statically indeterminate structures, allowing for rapid acquisition of the desired mechanical results. This method works by dividing the structure into several members connected by nodes, with each member and its connecting nodes at both ends considered as stiffness elements.

### 2.1. Stiffness Method

The calculation steps involve defining the geometry of steel frame, calculating the stiffness matrices for all elements, form transformation matrices, assembling the global stiffness matrix, then applying boundary conditions to solve all nodal displacements, and calculating internal forces and moments.

#### 2.1.1. Stiffness matrix.

The basis of the stiffness method relies on the relationship between displacement and load that applied along the member axis at its two end nodes, The relationship can be expressed as:

$$q = kd \quad (1)$$

The stiffness matrix shows in the following formula represents the stiffness of a single element with respect to both bending and axial deformations in its local coordinate system.

$$k = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \quad (2)$$

Each element in the matrix is referred to as a member stiffness influence coefficient, analogous to Hooke's law. The coefficient at the i-th row and j-th column represents the reaction force generated at node i along the member's axial direction when node j undergoes a unit displacement along the member's axial direction. This forms the basis for establishing the relationship between displacement and load.

#### 2.1.2. Transformation matrix.

As mentioned above, the stiffness matrix is established based on the local coordinate system. To ensure consistency in the directions of forces and displacements during the calculation process, a transformation matrix will be used to convert all vectors and the stiffness matrix from the local coordinate system to the global coordinate system. This facilitates calculation and ensures uniformity and accuracy in the results.

$$T = \begin{bmatrix} l & m & 0 & 0 & 0 & 0 \\ -m & l & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & 0 \\ 0 & 0 & 0 & -m & l & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \quad (3)$$

For each element, a transformation matrix ( $T$ ) is established based on the angle in global system.  $\theta$  is the angle of the element with respect to the global coordinate system. Since the moment in the local system is the same as in the global system, the corresponding coefficients for moments in the transformation matrix is 1. The local stiffness matrix of each element is transferred to global stiffness matrix by the following equations:

$$D_e = T^T d \quad (4)$$

$$Q_e = T^T q \quad (5)$$

$$Q_e = K_e D_e \quad (6)$$

$$K_{\text{global}} = T^T k_{\text{local}} T \quad (7)$$

This transformation operation will align the element's stiffness characteristics with the global coordinate system of the entire structure.

## 2.2. Equivalent Nodal Load

In practical calculations, the loading conditions of a structure are often complex, and loads are not always applied directly at the nodes. Sometimes, uniform distributed loads are applied along the elements. Since the stiffness method calculates the response at the nodes, these distributed loads need to be converted into equivalent nodal loads for accurate analysis.

Equivalent nodal loads reproduce the same structural response as the original applied loads. This conversion is based on the assumption that the element is fixed at both ends. By solving the system under the applied loads, it can be determined that the reaction forces and fixed-end moments at these supports. The reaction forces at the fixed supports counteract the effects of the loads on the beam, ensuring the structure is in equilibrium. Thus, although the fixed-end moments and forces are in the opposite direction of the applied loads, they can be treated as equivalent loads during the analysis.

## 2.3. Euler–Bernoulli Beam Model

When a distributed load is applied to an element, it is sometimes not uniform. To handle various distributed load cases, the Euler–Bernoulli beam model will be used. The main principle is to analyze the relationship between load and deflection through integration. The following equations represent the derivation process starting from the distributed load.

$$v^{IV} = \frac{w}{EI} \quad (8)$$

$$v''' = \int \frac{w}{EI} dx + C_{v1} \quad (9)$$

$$v'' = \iint \frac{w}{EI} dx dx + C_{v1}x + C_{v2} \quad (10)$$

$$v' = \iiint \frac{w}{EI} dx dx dx + C_{v1} \frac{x^2}{2} + C_{v2}x + C_{v3} \quad (11)$$

$$v = \iiiii \frac{w}{EI} dx dx dx dx + C_{v1} \frac{x^3}{6} + C_{v2} \frac{x^2}{2} + C_{v3}x + C_{v4} \quad (12)$$

where  $w$  is the applied load, and  $EI$  represents the flexural rigidity of the member.  $v$  stands for the deflection in the vertical direction. The constants  $C_{v1}$ ,  $C_{v2}$ ,  $C_{v3}$ ,  $C_{v4}$  are the constants that arise during the integration process. These constants will be determined based on the boundary conditions of the element, which will allow the complete equation to be formed.

## 2.4. Boundary Conditions

There are four basically boundary conditions, fixed supports, Pin support, roller support and free edges. Each supports have different boundary conditions, for vertical displacement, at fixed supports,  $v = 0$ ,  $v' = 0$ , for pin supports  $v = 0$ ,  $v'' = 0$ , for roller supports  $v = 0$ ,  $v'' = 0$ , for free edges, no shear force and the moment at the free edge  $v'' = 0$ ,  $v''' = 0$ , use these boundary conditions can solve the constant  $C_{v1}$ ,  $C_{v2}$ ,  $C_{v3}$ ,  $C_{v4}$ .

## 2.5. Establish Equilibrium

Based on the stiffness matrix, transformation matrix, and equivalent nodal load mentioned above, a complete equation can be established to solve the structure finally, as shown in the following equation.

$$q = kTD_e + q_F \quad (13)$$

After obtaining the vectors corresponding to each degree of freedom at every point, further analysis of the structure can be conducted. This includes plotting the Shear Force Diagram (SFD), Bending Moment Diagram (BMD), and deflection shape. For the BMD, the cut method can be applied. The SFD and deflection can be plotted by integrating the equation obtained from the area under the curve.

## 3. Member Capacity Calculation and Selection

### 3.1. Original Design

The analysis and calculation methods mentioned above will first be applied to the original design. The design process begins by simplifying each grid of a warehouse into a 2D frame with four main elements. The two feet are fixed points, and the connection points between each element are rigid, with initial dimensions including height, width, slope, and grid spacing. For each element, the above calculation methods are used to analyze the behavior of the entire structure under different loading conditions and different designs.

### 3.2. Wind Load Simulation

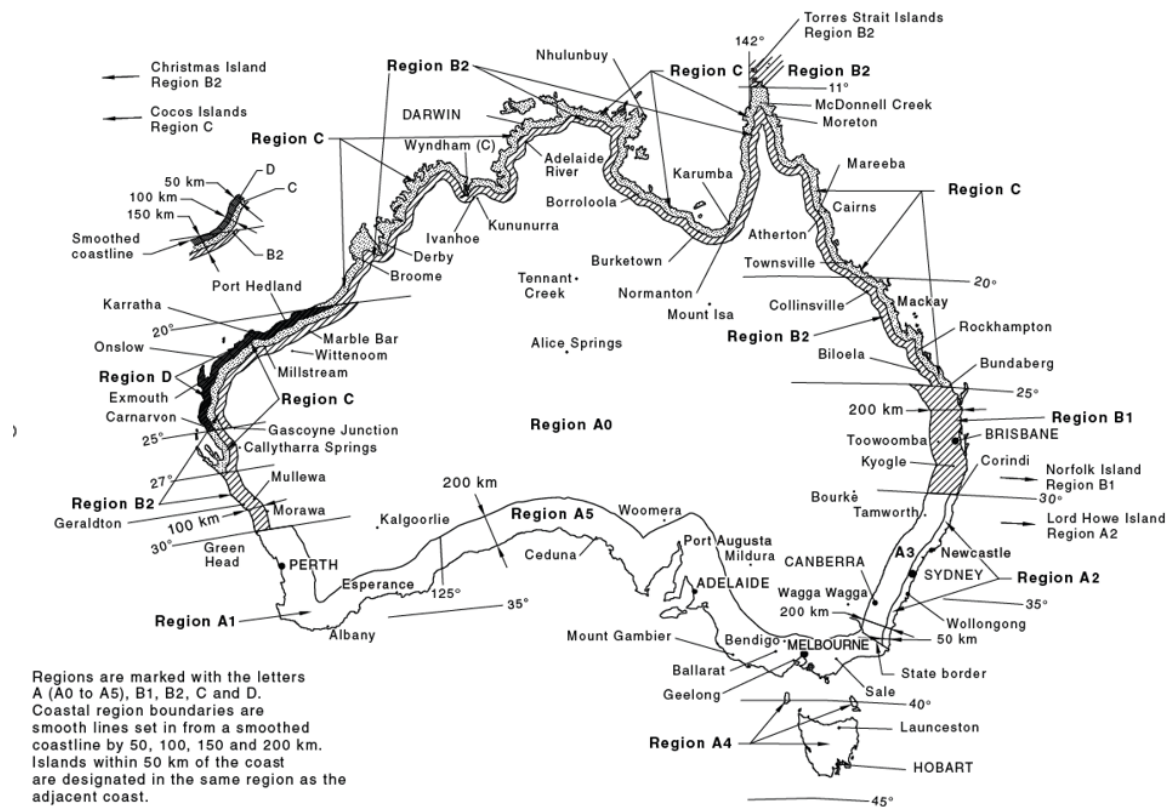
Wind load is an important factor in the design process, especially for structures with large surface areas, as wind pressure can have a significant impact on them. In the calculation and analysis process, wind load will be calculated according to AS1170.2. Based on factors such as the location, orientation, and dimensions of the building, appropriate wind load simulation will be conducted.

### 3.2.1. Wind speed calculation.

Wind speed directly affects wind pressure, and the corresponding wind speed can be calculated using Eq. 14.

$$V_{sit,\beta} = V_R M_d (M_{z,cat} M_s M_t) \quad (14)$$

The regional wind speed ( $V_R$ ) is selected from AS1170.2 [4]. Based on the wind occurrence frequency and the regional characteristics defined by the standard shows in Fig. 1. Based on the structure's location and the required wind speed occurrence frequency, the corresponding wind speed is selected.  $V_R$  means that the wind speed corresponding to R will occur once every R year.



**Figure 1.** Wind regions [4]

The wind Direction Multiplier ( $M_d$ ) is a coefficient used to account for the effect of wind direction. Terrain and Height Multiplier ( $M_{z,cat}$ ) adjusts the wind speed based on the surrounding terrain and the height of the structure. Shielding Multiplier ( $M_s$ ) is the shielding multiplier indicates the effect of surrounding buildings which may provide shadings or expose the structure to more wind. These parametes can be read from AS1170.2 [4]. Topographic Multiplier ( $M_t$ ) is based on the “Geoscience Australia report”, the topographic multiplier  $M_t$  is to account for the influence of topography on wind effects which is assumed as 1. Based on the wind speed calculated from the above content, wind pressure for side wall and raft can be calculated using Eq. 15.

$$\rho = (0.5\rho_{air})[V_{des,\theta}]^2 C_{shp} C_{dyn} \quad (15)$$

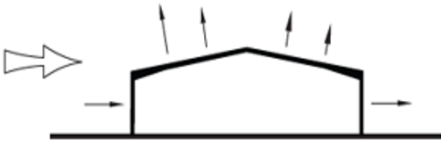
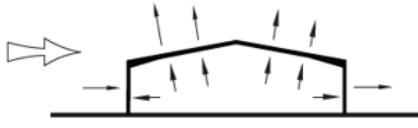

Air density ( $\rho_{air}$ ) is considered as  $1.2 \text{ kg/m}^3$  under standard environmental conditions. Design Wind Speed ( $V_{des,\theta}$ ) is obtained through calculation, and  $V_{sit,\beta}$  is the wind speed within the maximum range of  $45^\circ$  for  $V_{sit,\beta}$ . The factor that used to calculate wind load for side wall can be calculated with

Eq. 16. Dynamic Factor ( $C_{dyn}$ ) considers the influence of dynamic interaction between wind and the structure. Which is always set as 1. Aerodynamic shape factor ( $C_{shp}$ ) takes into account the effect of the structure's configuration on wind pressure and is specifically calculated using Eq. 16.

$$C_{shp} = C_{p,e}K_aK_{c,e}K_lK_p \quad (16)$$

External pressure coefficients ( $C_{p,e}$ ) represent the pressure effect directly on the windward side and leeward side building of the building and the external pressure coefficient. For leeward side, the external pressure coefficients can be also read from AS1170.2 [4]. Area Reduction Factor ( $K_a$ ) accounts for the impact of the tributary area and can be read from the AS1170.2 [4]. Action Combination Factor ( $K_{c,e}$ ) considers the effect of wind pressure acting on different structural elements and orientations which can be read from the Table1.

**Table 1.** Action Combination Factor [4]

Design case	Example diagram	External $K_{ae}$	Internal $K_{ai}$
(a) 3 effective surfaces Pressures from windward and leeward walls in combination with roof pressures		0.8	1.0 (not an effective surface)
(b) 4 effective surfaces Pressures from windward and leeward walls in combination with roof pressures and internal pressures		0.8	0.8
(c) 3 effective surfaces Pressures from side walls in combination with roof		0.8	1.0 (not an effective surface)

Local pressure factor ( $K_l$ ) is used to calculate wind pressure on supporting members. To simplify the calculation,  $K_l$  is set to 1.0 throughout the analysis. Permeable cladding reduction factor ( $K_p$ ) considers the effect of cladding on wind pressure for both roofs and sidewalls. For simplification,  $K_p$  is also set to be 1 during the calculation. The external wind pressure of roof can use the same equation to calculate the aerodynamic shape factor for the side wall can be applied for external wind pressure.

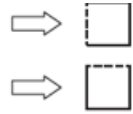
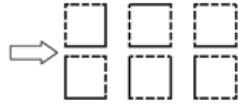


$$C_{shp} = C_{p,e}K_aK_{c,e}K_lK_p \quad (17)$$

The aerodynamic( $C_{p,e}$ ) shape factor needs to be determined based on different slopes and dimensions of the structure, which can be read from AS1170.2 [4]. The remaining coefficients  $K_a$ ,  $K_{c,e}$ ,  $K_l$ ,  $K_p$  are calculated in the same way as in the side wall's part. For internal pressure, the aerodynamic shape factor is given by Eq. 18.

$$C_{shp} = C_{p,i}K_{c,i}K_v \quad (18)$$

Internal Pressure coefficient ( $C_{p,i}$ ) is determined based on the open area conditions of the building's walls and the orientation of the structure relative to the wind direction. Which can be read from the Table 2.

**Table 2.** Internal Pressure coefficient [4]

Condition	$C_{p,i}$	Examples showing permeability and wind direction
One wall permeable, other walls impermeable:	$C_{pe}$ for the windward wall -0.3	
Two or three walls permeable, other walls impermeable: (i) permeable (ii) impermeable	-0.1, 0.2 -0.3	
All walls permeable	-0.3 or 0.0, whichever is more critical	
A building effectively sealed and having non-opening windows	-0.2 or 0.0, whichever is more critical	

Internal Action Combination Factor ( $K_{c,i}$ ) is same with the external factor  $K_{c,e}$ . Open Area/Volume Factor ( $K_v$ ) is based on the guidelines provided in AS1170.2 Clause 5.3.4. the value of  $K_v$  is set to be 1.

### 3.2.2. Wind pressure transfer to uniform distribute load.

After calculating the wind pressure, it needs to be converted into a uniform distributed load for analysis. For the windward side wall, leeward side wall, and roof, the corresponding load distribution can be determined. Then, using the method mentioned in Section 3.2, the obtained Uniform Distributed Load (UDL) can be converted into equivalent nodal loads for analysis and calculation. Dead load and live load also need to be considered. The dead load mainly comes from the self-weight of each element, which can be obtained from a steel table. For live load, based on AS1170.1 Clause 3.5.1 for roof actions, live load can be read from AS1170.2 [4].

$$UDL = (\rho_{\text{external}} + \rho_{\text{internal}}) * \text{tributary width} \quad (19)$$

### 3.3. Member Capacity Calculation and Selection

The calculations are based on AS4100 [5]. To determine the capacity of each member, the following key values need to be considered. The nominal member bending capacity for rafter and column can be calculated by Eq. 20.

$$M_{bx} = \alpha_m \alpha_s M_{sx} \quad (20)$$

where  $\alpha_m$  is the moment modification factor, and  $\alpha_s$  is the slenderness reduction factor. The coefficients are calculated using the following equations.

$$\alpha_s = 0.6 \times \left\{ \sqrt{\left(\frac{M_{sx}}{M_{oa}}\right)^2 + 3} - \frac{M_{sx}}{M_{oa}} \right\} \leq 1.0 \quad (21)$$

$$M_o = \frac{\pi}{L_e} \sqrt{EI_y GJ} \sqrt{1 + \frac{\pi^2 EI_w}{GJ L_e^2}} \quad (22)$$

$$L_e = k_t k_l k_r l \quad (23)$$

When top flange is in compression,  $L_e$  can be determined from Table 3. Where  $S_p$  is span of the purlin and  $S_f$  is span of fly bracing.

**Table 3.** Calculation of  $L_e$

$L_e$ for Column and raft	
Top flange is in compression	$L_e = 1.0 \times 1.0 \times 0.85 \times S_p = 0.85S_p$
Bottom flange in compression	$L_e = 1.0 \times 1.0 \times 0.85 \times S_f = 0.85S_f$
$L_e$ for Column raft	
Inside flange in compression	$L_e = 1.0 \times 1.0 \times 0.85 \times S_f = 0.85S_f$
Outside flange in compression	$L_e = 1.0 \times 1.0 \times 0.85 \times S_p = 0.85S_p$

Combined actions include compression and bending, three bending moment capacity need to be calculated: in-plane member moment capacity (Mix), section moment capacity (Mrx), and out-of-plane member moment capacity (Mox). The following process shows how to calculate these three values.

$$N_{cx} = \alpha_{cx} N_{sx} \leq N_{sx} \quad (24)$$

$$\lambda_n = \left(\frac{L_{ex}}{r_x}\right) \sqrt{(kf)} \sqrt{\left(\frac{f_y}{250}\right)} \quad (25)$$

$$\alpha_a = \frac{2100(\lambda_n - 13.5)}{\lambda_n^2 - 15.3\lambda_n + 2050} \quad (26)$$

$$\lambda = \lambda_n + \alpha_a \alpha_b \quad (27)$$

$$L_{ex} = k_e L \quad (28)$$

where  $k_e$  is set to be 1, because L is the length of the column. According to AS4100,  $\alpha_b$  is set to be -0.5.

$$\eta = 0.00326(\lambda - 13.5) \geq 0 \quad (29)$$

$$\xi = \frac{\left(\frac{\lambda}{90}\right)^2 + 1 + \eta}{2\left(\frac{\lambda}{90}\right)^2} \quad (30)$$

$$\alpha_c = \xi \left[ 1 - \sqrt{1 - \left( \frac{90}{\xi \lambda} \right)^2} \right] \quad (31)$$

$$N_{sx} = k_f A_n f_y \quad (32)$$

where  $A_n$  is net cross-sectional area, which can be read from onesteel table. For minor axis compression capacity  $N_{cy}$ , The same calculation steps and methods used for  $N_{cx}$ , but with the values in the y-axis direction. Overall, these three value can be calculated,  $\Phi$  is 0.9 for all three values,  $N^*$  is the maximum axial force.  $M^*$  is the maximum bending moment. The result shows in Table 4.

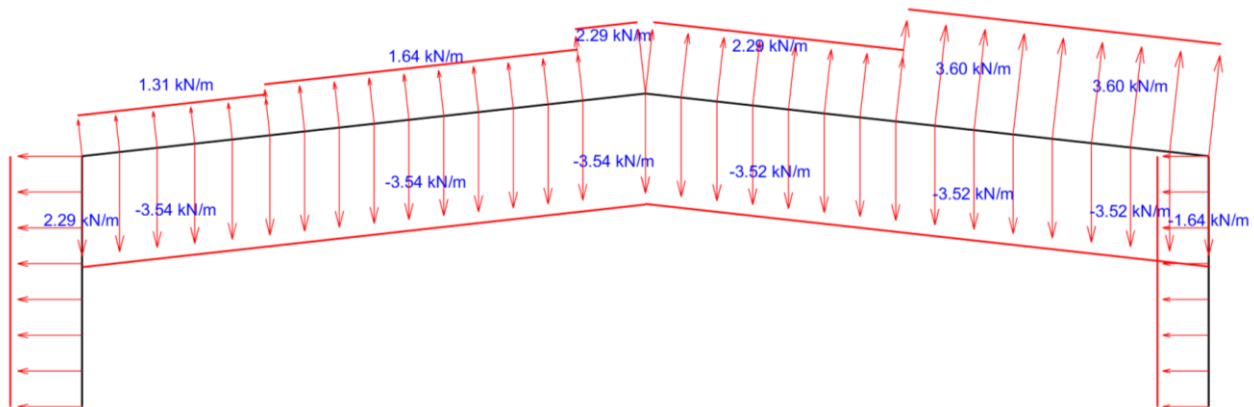
**Table 4.** Bending moment check

Capacity	Moment check	Formula location
In-plane member moment capacity (Mix)	$M^* \leq \phi M_{ix} = \phi M_{sx} \left( 1 - \frac{N^*}{\phi N_{cx}} \right)$	AS4100, Clause 8.4.2.2
Out-of-plane member moment capacity (Mox)	$M^* \leq \phi M_{ox} = \phi M_{bx} \left( 1 - \frac{N^*}{\phi N_{cy}} \right)$	AS4100, Clause 8.4.4.1
In-plane section moment capacity (Mrx)	$M^* \leq \phi M_{rx} = \phi M_{sx} \left( 1 - \frac{N^*}{\phi N_s} \right)$	AS4100, Clause 8.3.2

Based on the aforementioned calculation method, the capacity of each member in the OneSteel table is calculated. Then, considering the actual loading conditions of the structure, the most suitable member material is selected for each element.

### 3.4. Result and Discussion

In the case where the shape of the structure is as follows, with the height fixed at 4m and the width at 18m, the above calculation method and MATLAB model can be used to quickly select materials for each element of the structure while ensuring the serviceability and safety of the structure. Under the loading conditions as shown in Fig. 2, analysis and calculation are performed using the aforementioned model.



**Figure 2.** Applied load condition

Design result for all these four elements can be found by the model. The suitable material for left column is "150UC23.4". The suitable material for left raft is "250UB31.4". The suitable material for right column is "200UB29.8". The suitable material for right raft is "150UC23.4".

The results demonstrate that the model established based on the calculation steps in the paper can operate smoothly and efficiently, significantly simplifying the calculation process and accelerating the design progress. It quickly provides detailed information on the forces acting on each element of the structure and the corresponding material specifications that should be selected, ensuring structural stability. Additionally, the model automatically and efficiently selects the most suitable member material under different loading conditions, while ensuring the versatility and general applicability of the model.

#### **4. Conclusion**

The main analysis method in this paper is based on the stiffness method, combined with the Euler–Bernoulli beam model, to analyze and calculate the steel structure. This approach determines the moment, shear force, displacement, and reaction force at each node of the structure, ultimately producing the bending moment diagram and shear force diagram. Considering real-world conditions, wind loads are simulated and applied to the structure, alongside the self-weight of the structure, to simulate all uniform distributed loads it may experience.

Additionally, the bending moment capacity for each member listed in the Onesteel table is calculated. By integrating the calculated results under specific loading conditions, the appropriate material specifications are selected to ensure the safety and stability of the structure. All these processes are performed in MATLAB, with the aim of developing an automated design model. The model successfully automates structural analysis, design, and material selection, offering an efficient approach for design and calculation.

#### **References**

- [1] S.D. Guest, The stiffness of prestressed frameworks: a unifying approach, *International Journal of Solids and Structures* 43. 3-4 (2006) 842-854.
- [2] W. Shen, and M. Ohsaki & M. Yamakawa, Robust geometry and topology optimization of plane frames using order statistics and force density method with global stability constraint. *International Journal for Numerical Methods in Engineering* 122.14 (2021) 3653-3677.
- [3] W. Shen, and M. Ohsaki, Geometry and topology optimization of plane frames for compliance minimization using force density method for geometry model *Engineering with Computers* 37 (2020) 2029-2046.
- [4] AS 1170.2 Structural design actions - Wind actions, Part 2: Commentary (Supplement to AS/NZS 1170.2:2002), 2002.
- [5] AS4100 Steel structures - Commentary (Supplement to AS 4100-1998), 1999.