Finite Element Analysis of the Maximum Stress at the Joints of the Transmission Tower

Zarina Itam\textsuperscript{1*}, Salmia Beddu\textsuperscript{1}, Nur Liyana Mohd Kamal\textsuperscript{1}, Khaled H. Bamashmos\textsuperscript{1}

\textsuperscript{1}Department of Civil Engineering, Universiti Tenaga Nasional, Malaysia

E-mail: iZarina@uniten.edu.my

Abstract. Transmission towers are tall structures, usually a steel lattice tower, used to support an overhead power line. Usually, transmission towers are analyzed as frame-truss systems and the members are assumed to be pin-connected without explicitly considering the effects of joints on the tower behavior. In this research, an engineering example of joint will be analyzed with the consideration of the joint detailing to investigate how it will affect the tower analysis. A static analysis using STAAD Pro was conducted to indicate the joint with the maximum stress. This joint will then be explicitly analyzed in ANSYS using the Finite Element Method. Three approaches were used in the software which are the simple plate model, bonded contact with no bolts, and beam element bolts. Results from the joint analysis show that stress values increased with joint details consideration. This proves that joints and connections play an important role in the distribution of stress within the transmission tower.

1. Introduction

Joint in transmission towers have been studied for several decades now due to the effects they bring to the stress distribution within the structure. Knight and Santhakumar [4] conducted tests on a full-scale quadrant of the lowest panel of a transmission tower. They pointed out that the secondary stresses caused by bolted joint could be significant enough to cause failure of leg members even under normal working-load conditions. Chan et al. [3] compared the experimental failure loads of single angle struts with those predicted by design code equations and numerical analysis results, and concluded that more reasonable ultimate load predictions can be obtained by considering both the effects of joint fixity, which increase member capacity compared to the ideal pinned-joint conditions, and eccentricity which reduces capacity compared to ideal centric loading. Engineers normally use four common methods for modeling bolted connections. The simplest kind of models for a bolted joint does not include the bolts themselves. Instead, bonded contact is defined between the joint surfaces. This model however, would not allow individual bolt loads to be extracted. Beam Element Bolt is another method for modeling bolts. In this method, beam elements will be created attached to an edge or face as a beam connection, and is generally done with a fixed joint for line bodies. The third method is Solid Element Bolt, where it allows the model to account for contact between the washer/bolt and the rest of the joint. The last method is by modeling the threads. The thread geometry with frictional contact defined between the threads can be explicitly modeled. However, these models can be extremely computationally expensive. A very fine mesh is required in the threads. In this paper, a study on the effects of stress caused by the loads applied on the tower’s members will be conducted. This paper is mainly divided...
into two stages: The structural analysis of the tower, and the joint analysis. The structural analysis of the tower will determine the joint with the maximum stress. The joint analysis will focus on one joint to study its effects on the tower behavior.

2. Analysis of the transmission tower

2.1. Transmission tower geometry
The purpose of this stage is to analyze the whole tower to determine the stresses at the joints and the forces in each member. The effects of joint will be neglected in this stage. All the joints are assumed as pinned. STAAD Pro software will be used due to its ease of usage, enhanced features, and its efficiency in displaying results. The tower consists of 13 bays with a total height of 37.41 meter and two faces: transversal, and longitudinal face. The European Angle (L-section) was used for this model. The material used for all members is steel.

2.2. Results and discussion
In this stage, the stresses distribution in the tower under different loading condition was determined. Five loading conditions were considered; Normal Condition, Ground Wire Broken, Top Conductor + Middle Conductor Broken, Middle Conductor + Bottom Conductor Broken, and tower self-weight. The results from the analysis show that the maximum stress occurs at Node 18 (see Figure 1). Table 1 shows the stresses and internal forces of the members attached to Node 18. The force value of Members 291 and 270 are very small and can be neglected. The other forces will be considered in the joint analysis.

Table 2. Stresses and forces of members attached to Node 18.

| Member | Section (mm) | Load Condition | Stress (N/mm²) | Internal Force (kN) |
|--------|--------------|----------------|----------------|---------------------|
| 66     | L 120 × 120 ×10 | 3              | 196            | 454.871            |
| 14     | L 130 × 130 ×10 | 3              | 166.14         | 418.673            |
| 62     | L 80 × 80 × 6   | 3              | 146.847        | 137.302            |
| 77     | L 90 × 90 × 6   | 3              | 78.209         | 82.901             |
| 57     | L 75 × 75 × 6   | 3              | -32.685        | 28.600             |
| 299    | L 130 × 130 ×10 | 3              | 11.091         | 27.951             |
| 30     | L 130 130       | 3              | -7.232         | 18.225             |
| 74     | L 90 × 90 × 6   | 3              | -1.314         | 1.393              |
| 270    | L 75 × 75 × 6   | 3              | 0.692          | 0.692              |
| 291    | L 75 × 75 × 6   | 3              | -0.391         | 0.316              |

Figure 1. Node 18 and the associate members.
3. Analysis of joint

The purpose of this stage is to perform a detailed analysis for one joint of the transmission tower. Different parameters for the joint will be assumed. As given in the architectural drawing, all the connections are steel galvanised bolts and nuts as per BS4190. Finite element analysis software ANSYS was used. In this analysis, a steel plate with the thickness of 10mm and dimension of 500x500mm was used. Each member will be connected to this plate with two bolts of 16mm diameter. In this paper, three models will be considered (see Figure 2). Model 1 will consider only the plate without including the members in the design. This is the simplest design where the bolt will be represented by 16mm diameter holes. Members of the joint will be represented by its respective internal forces, as stated in Table 2. Model 2 includes a plate with the members attached to it. Similar to Model 1, bolts will be represented by 16mm diameter holes. Despite of this similarity, presence of the members attached to the plate makes it necessary to take into account the contact area between the plate and the members. Bonded contact is defined between the plate and the members. These two models are the simplest since they do not include a lot of details and there is no much setup needed for the design. At the same time, these models are considered as the least realistic models of the three models. Model 3 uses the Beam Element Bolts method. Five members will be designed and bolted to the plate. Bolts will be modeled by creating a line body and defining its cross-section in the ANSYS Design Modeler. The beam will be attached to the inner diameter of the bolt hole.

![Figure 2. Tower joint model approaches.](image)

3.1. Results and discussion

Figure 3 shows the stress distribution for the three models. It can be seen that the value of the stresses is the least at Model 1, and increases in Model 2 and Model 3. For example, in Bolt 1 the value of stresses are 141N/mm² for Model 1, 178.5N/mm² for Model 2, and 349.6N/mm² for Model 3. In Model 1, where the members of the joints are replaced by its forces and only the plate is considered, the maximum stress value produced at the bolts is 165N/mm². This result is much lesser than the value obtained by STAAD Pro where the maximum stress is 196N/mm². By only considering the plate and ignoring the members attached to the joint, this model becomes the least realistic between the three models. Model 2 takes into account the members attached to the plate and does not include bolts design. Thus, this model can be considered as the most similar model to that in the tower analysis. They are similar in terms of geometry and ignoring the bolts effects. For this reason, the results of this model is similar to that of STAAD Pro. The maximum bolt result in this model is 217.3N/mm² which is nearly 20N/mm² more than STAAD Pro results. This increment in the stress value is due to the presence of the holes in the members and the plate. Model 3 is the most realistic model. It considers the bolts effects in the design. The maximum stress for this model is 332.2N/mm².
4. Conclusion

The summary of the findings are presented below:

- The tower has maximum stress with value of 196 N/mm² at Node 18 of the tower.
- Model 3 shows that consideration of joint detailing in finite element simulation affects the analysis of the tower. The result of Model 3 shows a 78% stress increment compared to the tower analysis.
- As conclusion, the consideration of joint detailing in the analysis of transmission tower is effective, and can dramatically increase the stress produced at the members.

References

[1] Al-Bermani, F. G. A., Kitipomchai, S. 1992. Nonlinear Analysis of Transmission Towers. Engineering Structures, 14(3), 13 9-15.
[2] Bergstrom, R N., Arena, J. R., Kramer, J. M. 1960. Design of Self Supported Steel Transmission Towers. Journal of the Power Division, ASCE, 86(3), 55-88.
[3] Chan SL, Cho SH. Second-order analysis and design of angle trusses: elastic analysis and design. Eng Struct 2008;30(3):616–25.
[4] Knight GMS, Santhakumar AR. Joint effects on behavior of transmission towers. J Div, ASCE 1993;119(3):689–712.
[5] Lo, D. L. C., Morcos, A., Goel, S. K. 1975. Use of Computers in Transmission Tower Design. Journal of the Structural Division, ASCE, 101(7), 1443–1453.