FAILURE ANALYSIS ON SKUNK-ARM OF ELECTRICAL TOWER

Ahmad Rivai[1], Md Radzai Said[2]

[1,2]Faculty of Mechanical Engineering,
Universiti Teknikal Malaysia Melaka,
Karung Berkunci 1752, Pejabat Pos Durian Tunggal,
76109 Durian Tunggal, Melaka, Malaysia

Email: ahmadrivai@utem.edu.my

ABSTRACT

Un-proper procedure during installation of the wire-rope could introduce unusual loading to the skunk-arm of the high-voltage electrical tower. Understanding of structural behavior of this kind of structure under different loading should be helpful to take preventive measure again possible future accidental cases. Static and buckling analyses have been performed using Finite Elements Analysis (FEA). Skunk-arm structure modelized by one-dimensional elements, combination between bar and beam. Beam elements represent main structure that are flexural members and bar elements represent stiffener that can only be in tension or compression loading. Stress distribution due to maximum static loading did not show an excessive stress value, it means no yielding occurs in the structure. But buckling factor show the maximum loading is close to the critical loading that may provoke collapse of structure in buckling mode. The result shows the mechanism of structure failure, which begin with buckling of the lower members of the skunk. In normal condition, factor of safety of 2 is taken according to maximum loading of the structure.

KEYWORDS: *Skunk-arm, failure, buckling, bending, Finite Elements Analysis.*

1.0 INTRODUCTION

Despite the great design and analysis that have been made in a product development, failure continues to occur, often accompanied by human and economic loss.

The accident of a skunk cross arm of an electrical tower collapse when the worker was trying to lift-up the equipment for stringing the transmission line have several times happened.

To have a clear idea about what really happened and to prevent the event re-produce, an investigation has been conducted to determine the real cause of the collapse. In order to fulfill the research and lead a correct conclusion, fully understanding mode of operation of the component of the system involved, as well as knowledge of the possible failure mode must be acquired.

Based on shape of the structure, applied loading and material properties, structural analysis has been performed by Finite Elements Simulation, using PATRAN-NASTRAN software in static and buckling analysis to determine the most probably mechanism of the accident.

2.0 PROBLEM DESCRIPTION

This paper will discuss about possible failure of skunk-arm of electrical tower structure under critical loading.

The critical loading of skunk-arm of an electrical tower is at installation phase as shown in Figure 1. At that time one raises wire, insulators, running block and together with some tools using pulley system. If only one pulley is used, the total load experienced by the structure will be more than double of the total raising weight. This occurrence is due to the dynamic of rope tension and friction on the pulley. If the total raising weight is very near to the maximum allowable load, that situation is very dangerous because factor safety of the structure is generally taken equal to 2.

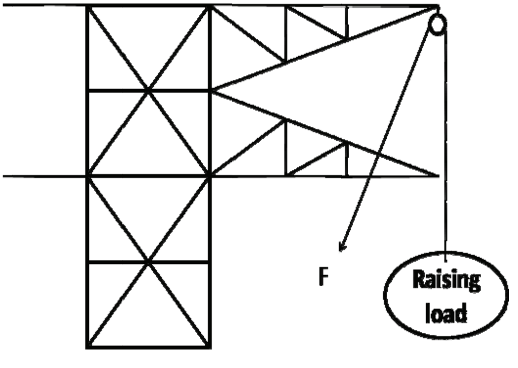

FIGURE 1
Loading of skunk-arm during installation

3.0 THEORETICAL REVIEW

Some important terms which will be used along this paper, will be reviewed:

3.1 Material properties

In our analysis we are more interested on mechanical properties, especially:

- Young's modulus: ratio of normal stress within the proportional limit to the corresponding normal strain
- Poisson's ratio: ratio between the strain of expansion in the direction of force and the strain of contraction perpendicular to that force $v = -\square_t/\square_1$
- Yield strength: highest stress a material can resist when exposed to a stretching load before yielding.

3.2 Linear Static Analysis

Evaluation of a structural behavior in the elastic region, under load assumed to be constant independently of the time. The results show how the structure will be deformed and how the stresses will be distributed. We look for un-event stress distribution or stress concentration, which could initiate the failure of the structure through comparison with the yield stress.

The failure in static commence with the yielding of material. The structure will deform plastically, if the load is not too high the deformation will occur slowly. If the load reaches the maximum load, a fracture of the structure could happen

3.3 Yield Criteria

Yield point of a material is defined in engineering and materials science as the stress at which a material begins to plastically deform. Prior to the yield point the material will deform elastically and will return to its original shape when the applied stress is removed. Once the yield point is passed some fraction of the deformation will be permanent and non-reversible.

In structural engineering, this is a soft failure mode which does not normally cause catastrophic failure unless it accelerates buckling.

3.4 Buckling Analysis

Buckling is defined as an instance of lateral bending or bowing of the beam shape due to a compressive load. There are three basic types of beam failures in compressive load:

- Compressive material failure, the material cracks or crumbles. This type of failure usually happens to beams that are very short and fat.
- Buckling failure, lateral bending or bowing of the column shape. This type of column failure usually happens to columns that are very long and skinny.
- A combination of both compressive and buckling failures. This type of column failure occurs when length and width of a column is in between a short and fat and long and skinny column.

Buckling is phenomenon of structure instability, alteration from stable state to instable state could happen suddenly. The maximum axial load a beam can support when it is on the verge of buckling is called the critical load.

Critical load for buckling usually is much lower than maximum load that will cause a failure in static. For this reason, we have always to evaluate the buckling behavior for the structure in compression.

4.0 SIMULATION

Analysis of the structure is done by using finite elements software, depth understanding of the physical problem must be acquired in order to build a good and right model and to reach the correct results of calculation.

Static and Buckling analysis will be performed to simulate different possibilities of loading and structure configuration in order to understand mechanism of the failure/collapse of the structure.

4.1 Finite elements model

Garbage in garbage out is a principle that must be remembered in development of the model. To have an accurate results, inputs or finite elements model must represent real physical phenomenon.

Based on three-view drawing and material list currently used, all of structural components made of L-shape beam but different size for principals and stiffeners beam.

To represent skunk-arm structure, coordinate of points of joints have been determined and then all of the points were linked by lines (curves).

4.1.1 Elements

According to geometry model abovementioned, one-dimensional type of elements has been chosen. Depend on joints, function and loading of the structure, two types of one-dimensional elements might be employed:

- Principal beams structure (upper and lower members) work on bending because the beams made of one single rigid beam that could be loaded by transverse force. To accommodate this property that allows the displacement in the direction perpendicular to the axial direction of the beam, beam elements or flexural elements have been applied.
- Stiffener beams work as supporting beam and jointed to the principal beams structure by bolt joints, which could be modeled by pin joint. According to truss structure theory, beam jointed by pin joints, stiffeners loaded only in tension or compression. Modeling this kind of the structural members is more suitable by rod (bar) elements, which allow only displacement in direction axial of the beam.

4.1.2 Boundary conditions

Boundary conditions defined as the environment to the system or interaction between the system and its external environment, in structural problem the boundary condition are usually constrains and loadings.

i) Constraint

In physical problem, constrain of the skunk cross arm structure is the attachment of the structure to the core (main tower structure), which is assumed very rigid, through two bolted joints in each beam. This kind of joint still permit the rotation of the member in the bolt axis, thus we consider as a pin joint.

ii) Loading

Neglect proper weight of structure, loading was the total raising weight and pulled-up force through the system of pulley.

According to currently practiced, pulley was fixed to structure by intermediary of a plate, and could freely rotate in the vertical axis. It means that directions of the load are in the plane of pulley (Figure 2).

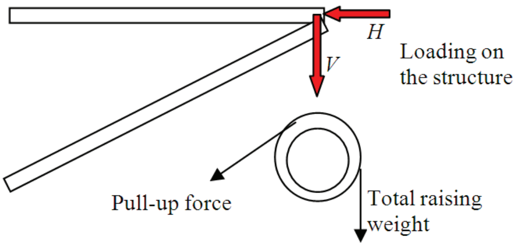

FIGURE 2
Loading of structure through a pulley

To simplify calculation of the load, we neglect acceleration of the raising weight and energy lost in friction between rope and pulley. We just consider the static equilibrium between raising weight and pull-up force, which means that pull-up force equals to raising weight.
For simulation the loads considered are:

V = 11,000 N
H = 0 N

4.1.3 Material properties

The skunk-arm made of high-grade galvanized steel. The shape of beam's cross section is L-shape with different size for principal and stiffener beams.

Mechanical properties of material for calculation:

- Yield stress : 390 MPa
- Poisson's ratio : 0.3
- Young's modulus : 203 GPa

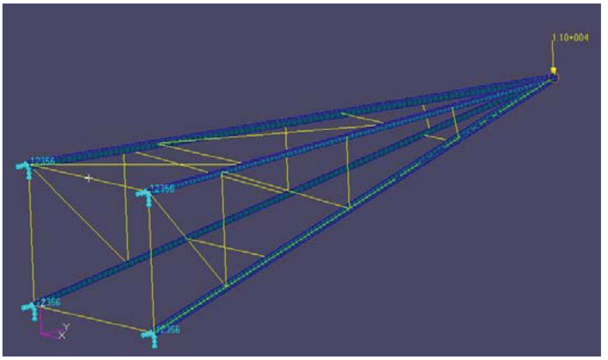

FIGURE 3
Complete finite elements model

4.2 Finite elements simulation

Different configuration and loading will be simulated and analysed in static and buckling.

4.2.1 Static Analysis

Static analysis performed for 2 cases: principal configuration and complete configuration.

4.2.2 Buckling Analysis

In this section buckling will also be analysed for principal members as well as complete structure. Simulation was done to have the first mode of buckling which is most critical and the results will be analysed for the load factor of buckling

5.0 RESULTS AND DISCUSSION

5.1 Static Analysis

The result of static simulation shows in Figure 4, for principal structure and Figure 5, for complete structure.

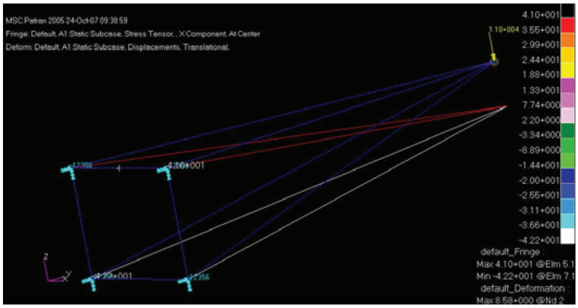

FIGURE 4
Stress and displacement of principal beams

For principal beams, results of simulation show that in static analysis, the maximum stress is 41 MPa occurs on the upper members (tension) and its minimum is -42.2 MPa on the lower members (compression) with maximum displacement is 8.58 mm. These values are still less than the yield stress of the material which means statically this structure is safe.

Regarding the above results for principal members, complete structure must be more safe because a part from principal beam, the stiffener were there to reinforce the structure by distributing and carrying the load. It is in accordance with the results in figure 5 stresses in the complete structure are distributed to each component in the way that the stress level is not too high, average of the stress is about +30 MPa but there is still stress concentration in some area, maximum stress is 81.2 MPa at the middle of upper beams and minimum of -70.9 MPa at the middle of lower beams. And maximum displacement is 10.6 mm at the tip of skunk cross arm.

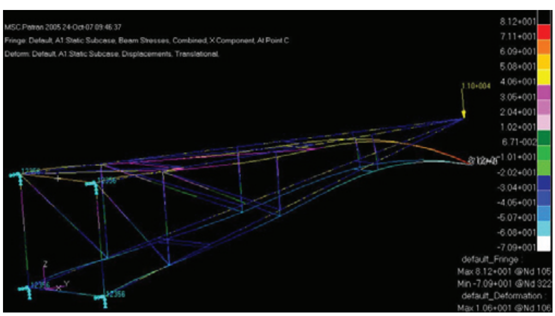

FIGURE 5
Stress and displacement of complete structure

5.2 Buckling Analysis

For principal beams, buckling phenomenon was characterized by huge bowing of the lower members up and outward of the structure as shown in Figure 6. Upper members experience little bit deformation which is insignificant compare to lower members.

In term of load factor, results of calculation show that for this configuration the load factor is 0.465, it means that structure will reach instability or buckling at the load of 0.465 x 11,000 = 5,115 N. In this case structure already failed under instability or buckling mode.

This result could be verified easily by calculation of Euler buckling in one lower beam in the direction of the weakest moment of inertia and Pcr found compared to the compressive force at the lower beams due to load of 5115 N. Different value between the two calculations, = 9.25 %, comes from the assembly of the beams that will increase the stiffness of the whole structure.

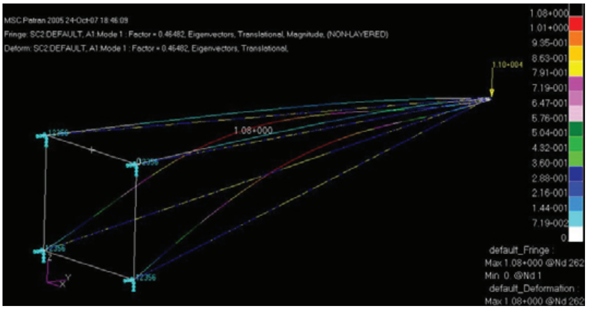

FIGURE 6
Buckling of principal beams

One of favorable effects of stiffeners in the complete structure was improvement of structural stiffness or rigidity, which in return will improve the behavior of structure under buckling loading. This fact is shown in the results of complete structure buckling.

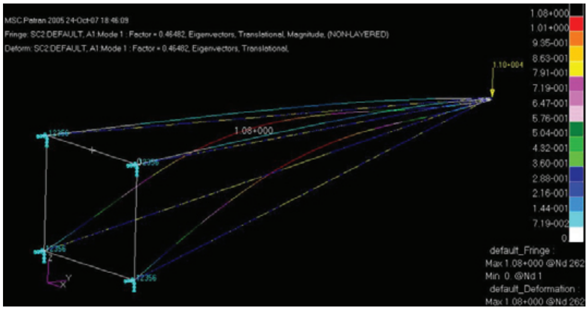

FIGURE 7
Complete structure buckling

Load factor for complete structure increase by about 2.5 times compare to the principal beams structure. Value of load factor is now 1.249, which means that the structure is still safe but already relatively close to the critical load of buckling that is about 13.7 kN.

Result of simulation shows that buckling for complete structure occurs especially at the lower members. But bowing is different from principal beams structure, lower members deflected into the left of skunk structure in the plane of lower beams. This occurrence due to the way how structure stiffeners have been arranged. From the model, we could see that at the top and two sides of skunk cross arm structure have been well stiffened by several beams arranged as triangular trusses in the plane that is the simplest stable structure. But at the bottom of the structure there were only two cross beams as stiffener members arranged in parallel that is unstable. It means that this region is the weakest stiffness, thus buckling will occur in this area.

6.0 CONCLUSION

According to simulation, failure of skunk-arm is more likely due to buckling or instability in lower members of the structure rather than to static loading. The results of static show that the stress on each member of structure is less than yield stress. But load factor for buckling case is too close to the critical load origin of buckling.

Regarding load factor resulting from simulation, structure was still safe but it was in the point of uncertainty. Small variation of loading could provoke change the stable structure to instable structure. Loading is close to critical buckling load.

In our calculation some assumptions have been used, but in reality there are many more factors that could increase the load of the structure. Factors that increasing load are: proper weight of the rope, wind loading, and most important factors are friction between rope and pulley and acceleration done by pulled-up force to raise the load.

7.0 REFERENCES

F.P. Beer,E.R Johnston, J.T. DeWolf. 2006 Mechanics of Material. Singapore: Mc Graw Hill.

Manual of PATRAN-NASTRAN

R.C. Hibbeler. 2008. Mechanics of Materials. Singapore: Pearson.