

Introduction

While I for one really enjoy the aspects of amateur radio operating, radios, and antennas associated with our hobby, I also enjoy other aspects, such as the mechanical and structural elements of towers. Because of this and my interest in steel fabrication to improve my station, I began looking for available FEM tools that were affordable and somewhat easy to use.

FEM refers to *finite element modeling*. FEM has applications to a diverse set of disciplines, to include, in part, mechanical and structural analysis, thermal, electromagnetic, acoustic, and even fluid analyses. Essentially in the use of this method, either in 2- or 3-dimensional space, one divides the structure up into much smaller physical elements such as squares, rectangles, or triangles, in 2-D space, and other more complex elements or polyhedrons in 3-D space. Figures 1 and 2 below provide examples for the two cases just mentioned.

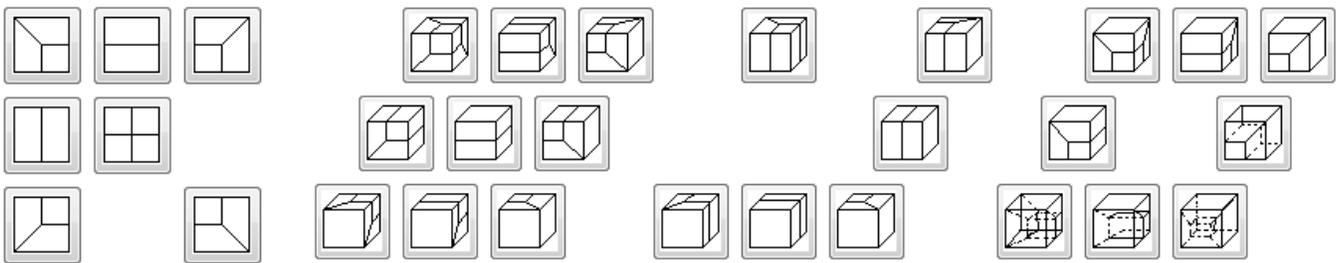


Figure 1
2-D

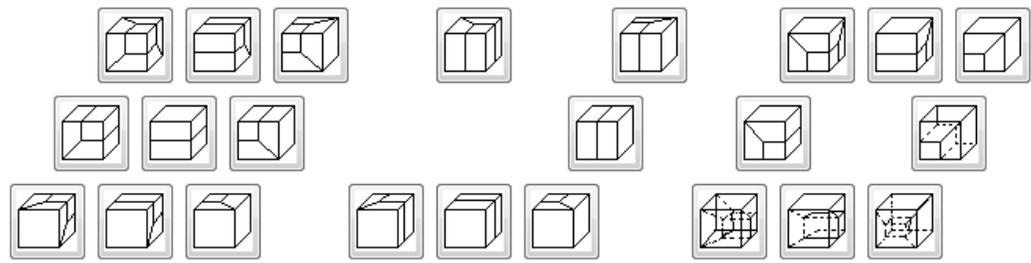


Figure 2
3-D

In the presence of forces and other “constraints”, FEM analysis for structures allows one to assess the compressive and tensile forces throughout the article of interest. A system of simultaneous equations, governed by specific constraints, is solved and in so doing, the distribution of forces and the accompanying outcome can be determined. The “constraints” refer to points on the structure that may freely rotate for example, or other points which are permanently fixed in both x, y, and z directions, such as the base of a tower sunk in concrete. The software tools which are available, some for free and others very expensive, allow you to construct your physical model in 2-D or 3-D space and visually see the distribution of forces, and specifically those areas where elements of your design may exceed the ultimate strength of your materials in use. Additionally, other aspects of the design, such as the amount of displacement, shear effects, or bending moments can be investigated.

The program I found, which is the primary focus for further discussions here, is LISA. It is available free off the internet at lisafea.com. The “free” version has a limitation of 1300 nodes. This may sound like a lot, but depending on your inventiveness, or not, in constructing your models, it may be very limiting. For \$99 it is possible to purchase a “Private Single User” license which, I believe, has no node limitations. Thus far I have found it unnecessary to purchase the single user license, but depending on my interests, that could change. If you are interested in analyzing full 3-D structures the free node count will probably be insufficient.

Getting Started

What is needed going forward? There is a plethora of material on the internet as well as countless books on statics and dynamics of structures, some rather complex for sure. Most readers are not likely inclined nor have the time to get into this area as though you were in a mechanical engineering course, understandably. I will recommend two texts which, with some study, can give you gratifying results and understanding without going back to college. The two I would recommend include 1) “Understanding Structures, Analysis, Materials, Design” by David Seward, and 2) “Simplified Design of Structural Steel” by Parker and Ambrose. The former lays a broader and perhaps more simplified framework for your understanding of forces, vectors, and general terminology. If you are more hardcore you may be interested in a text like “Statics and Dynamics” by Merriam.

Some Essential Definitions

It is essential to delineate and describe *external forces* – such as applied forces, and *internal forces* being produced as the result of the applied external loads. If a weight of 10 kg (kilograms) were suspended by a rope, this load would cause a force F of

$$F = mg = 10 \times 9.81 = 98.1N$$

where g is the acceleration of gravity, 9.81 m/s^2
and m is the mass, 9.81 kg , with N being newtons

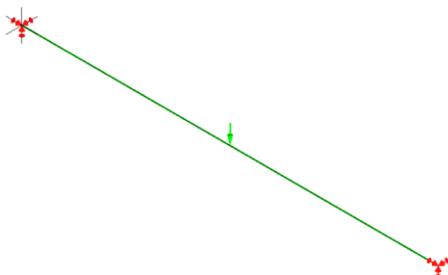
In such a scenario, for equilibrium, or in simple terms the weight stays in the same position, this force due to gravity must be opposed by a force in the rope of the same magnitude but opposite direction, “up”. In the vertical equilibrium case the rope has an internal force of 98.1 N , causing an internal force or *stress* in the rope.

$$\text{stress} = \text{force}/\text{area}$$

The unit of stress, symbolized by σ , is often in terms of N/mm^2 (newton per square millimeter) or Pascal (Pa), where one Pa is one N/m^2 . Many specifications in steel work will use units of Pa.

A Simple Example

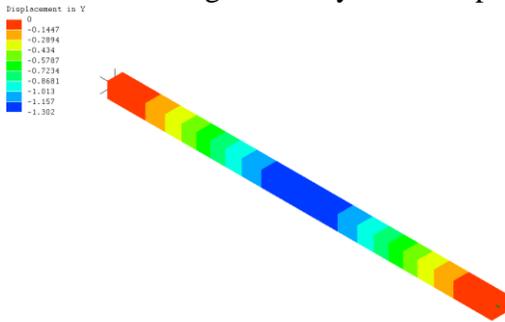
Let’s first consider a simple steel beam extending from $x = 0$ to $x = 10$, supported at both ends, and subjected to a downward force of 100 lbs at its center, i.e. $x = 5$. This simple scenario viewed within LISA is as shown below.



The red arrows at the two ends indicate a “fixed constraint”, in other words the ends of the beam are fixed in all three directions, x , y , and z . The green downward arrow depicts the placement of a downward force in the middle of the beam. The small three-arrowed element helps one keep track of which direction is which.

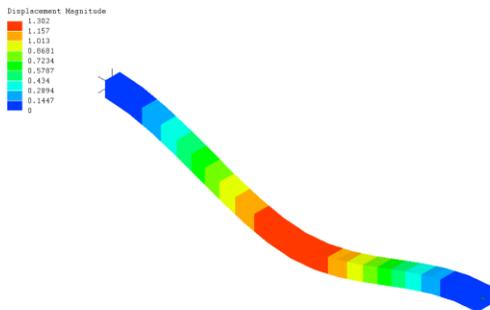


Performing one analysis for displacement in the y-direction gives the following:



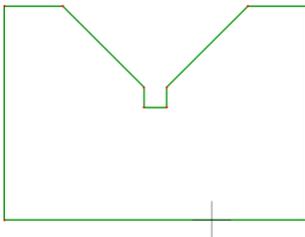
The amount of displacement is color coded. Since both ends of the beam are fixed and unable to rotate by definition, the red color indicates a displacement of zero. As one moves toward the center of the beam and the location of the 100 lbs downward force, the displacement downward increases, reaching a maximum of 1.302 at the center.

LISA allows you to look at this result in a “deformed state”, thus allowing you to see dimensionally the results rather than having to weigh just numbers. This same result is shown below.

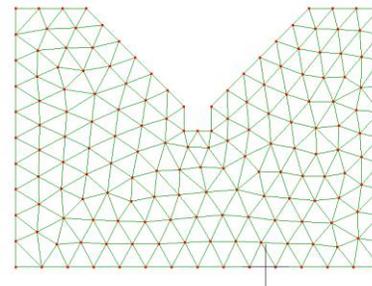


The results for the maximum deflection in the center of the beam match the 1.302 number in the figure above.

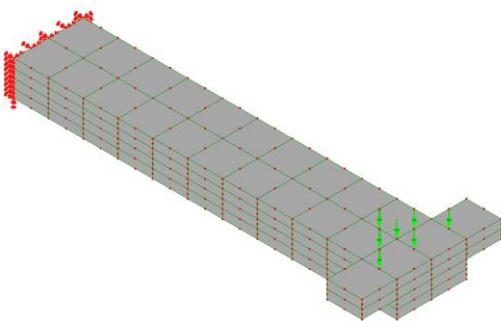
Another example is shown below in which a two-dimensional outline is the starting point.



With a simple command within LISA this two-dimensional structure can be “auto-meshed” on the way to performing a full finite element analysis.

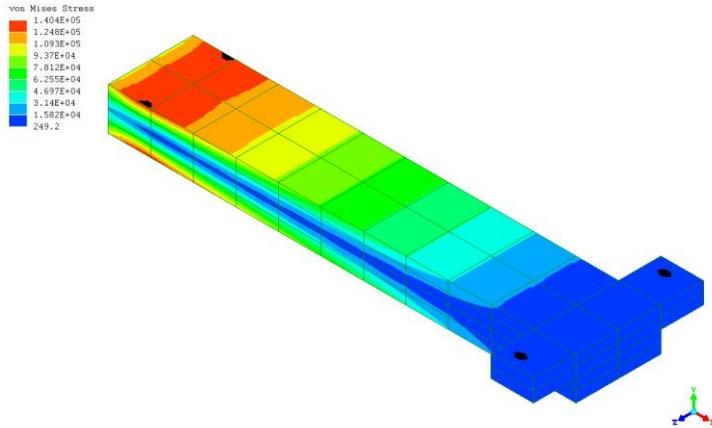


In yet another example a T-shaped beam is built, taking literally 1-2

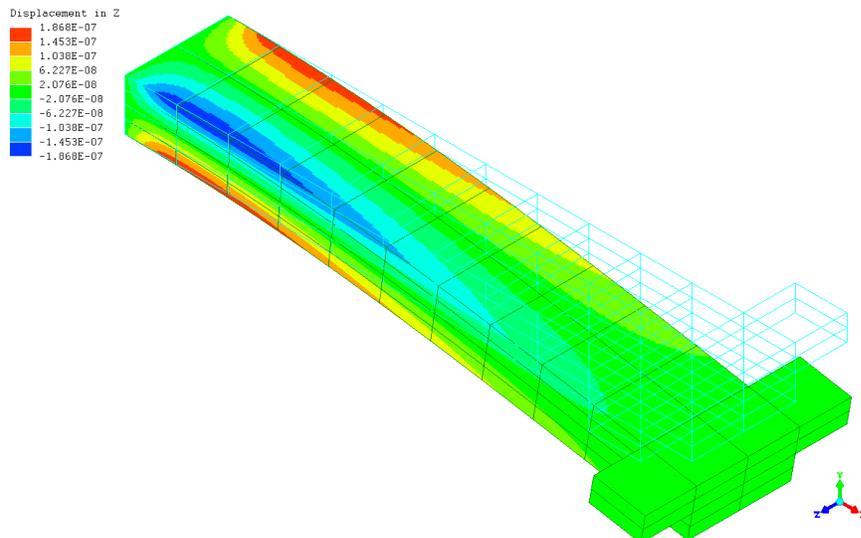


minutes, and 5000 lbs of force applied among 7 nodes at the end of the beam. The constructed model is shown to the left. The lines depict the different “elements” in the model; the 5 green downward arrows are the locations of the

downward force applied, and at the far end the red arrows indicate that end of the beam is “fixed”. Performing the analysis for “von Mises Stress” produces the results below.

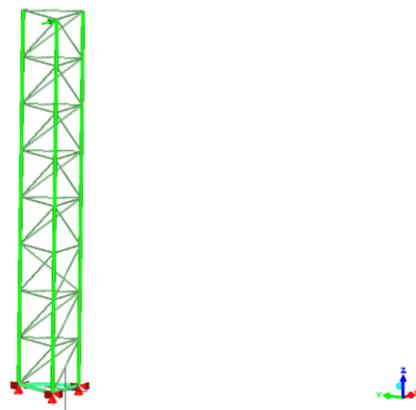


If you wish to see a magnified depiction of the resulting deformation under stress, this is possible too. The light blue lines in the figure below show the non-deformed state for easy comparison to the loaded case.



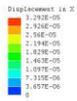
And finally, of interest to me, is the analysis of a tower section. This model depicts a ten foot section of tower which will be replicated and stacked to form a guyed tower of, perhaps 80 ft.

The main legs are of 1.25 OD steel tubing 0.120 in wall and the cross members are 1/2” steel rod.



In constructing this model I am not using the full meshing capability of the program, because in so doing I will almost instantly exceed the 1,300 node limit. Instead, each line segment is defined by two nodes and is classified as a “beam” with the proper steel properties.

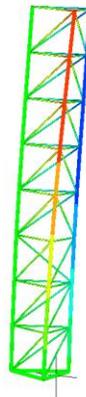
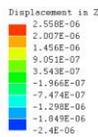
Applying a force of 500 pounds laterally along the x-direction gives the following results.



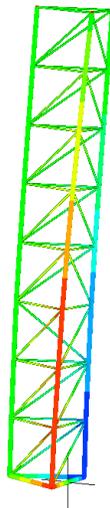
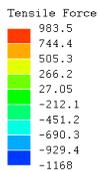
Exaggerated deformation along x



Exaggerated deformation along y



Deformation along z



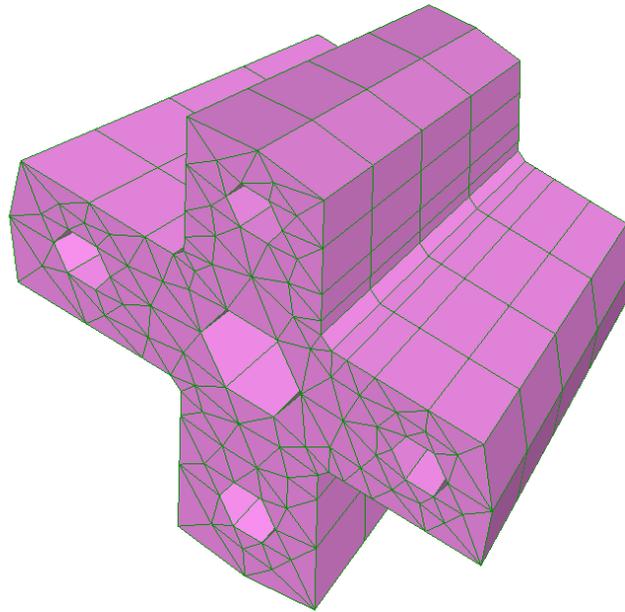
Of greatest interest to me, at least under this scenario, is the tensile force in each tower member. You must ensure that the yield strength, with a sufficient amount of safety margin, is not violated under expected worst case loading conditions.



To bring this analysis to completion I need to replicate this section to ~

80 ft and properly “guy the tower”, at which time I can begin assessing its strength and suitability for my antennas.

The types of structures and assessments one can make with this “free software” is impressive. Below is yet another example of a 3D structure “meshed out” ready for FEM analysis.



I hope this brief description of FEM has wetted your appetite to explore further.