Textile Composites and Inflatable Structures
E. Oñate and B. Kröplin (Eds)

DESIGN AND ANALYSIS OF TENSION STRUCTURES USING GENERAL PURPOSE FINITE ELEMENT PROGRAMS

Ruy Marcelo de Oliveira Pauletti[*]
Departamento de Engenharia de Estruturas e Fundações
Escola Politécnica da Universidade de São Paulo
Caixa Postal 61548 – 05424-970 São Paulo, Brazil
E-mail: pauletti@usp.br

Reyolando M.L.R.F. Brasil
Departamento de Engenharia de Estruturas e Fundações
Escola Politécnica da Universidade de São Paulo
Caixa Postal 61548 – 05424-970 São Paulo, Brazil
E-mail: rmlrdfbr@usp.br

Key words: tension structures, design and analysis, form finding, structural analysis, cutting patterns, finite element method, physical models, Ansys finite element program.

Abstract. This work discusses the use of general purpose finite element programs for the analysis and design of tension structures, such as cable nets and membranes. Some examples are presented, starting from simple cases, aimed to validate analysis procedures, and then progressing towards more realistic situations, with physical models being actually constructed to verify the full design procedure. Results suggest that general purpose programs can undertake quite satisfactorily most of the procedures involved in the tension structures design process, and may indeed be indispensable, in cases in which special types of analyses or modeling options are required, not readily available in special purpose programs. Some future research steps are also briefly outlined.

1 INTRODUCTION

Tension structures are characterized by a particular profusion of solutions. If not too many restrictions are imposed to shape, it is possible –with a lot more of freedom, compared to other structural systems– to arbitrate the stress field and then proceed to form finding. This variability of solutions, allied to the difficulties related to geometric and material non-linear behavior, usually overrules the use of analytical solutions, letting numerical analysis as the only general approach to the design and analysis of tension structures[1,2]. Indeed, according to Campbel[3], "no other class of architectural structural systems is as dependent upon the use of digital computers as are tensile membrane structures". Besides complexity, the process of design and analysis of tension structures presents a series of specificities, that usually lead, according to Tabarrok[2], to the use of *special purpose programs*, instead of *general purpose programs*.

In a special purpose program (SPP), part of the design *know how* is embedded in program routines. It is very probable that this type of program will be the option of the final user, in design offices. However, a good deal of the task of modeling a membrane or cable structure can also be undertaken with the aid of a general purpose finite element program (GPP), aimed to structural analysis. By definition, GPPs offer a range of types of analyses and modeling options. Besides, traditional GPPs are constantly being tested by a large number of users, and are updated according to their actual demands. That pushes evolution of their features and capabilities, as well as of their reliability. GPPs can indeed be indispensable, in design cases in which special types of analysis or modeling options, not readily available in special purpose programs, are required.

However, contrary to special purpose programs, the use of GPP to the modeling and analysis of tension structures usually requires previous customization, demanding a relatively deeper knowledge, both of those design phases already embedded in the SPPs and of the syntax particularities and resources of the GPP actually employed. It is reasonable to suppose that the major part of the procedures tested in the present work with the Ansys[4] program is reproducible, with due adaptation, with the aid of other good FE programs available in the technical market. Nevertheless, the chosen program is acknowledged as one of the GPPs of broader application spectrum, larger versatility and reliability, therefore constituting a good choice, as far as the scope of the present work is considered.

2 SOME BASIC EXAMPLES

As a first investigation, consider the process of finding the form of a freely hanging cable, under self weight (assimilated to constant nodal forces, but gravity could as well be imposed). Geometry and analysis options are rather simple. The model is composed of 10 tridimensional truss elements (LINK8). A geometric nonlinear, large displacements analysis was undertaken. The use of the purely iterative Newton's method resulted to be more efficient than an incremental process. Once the initial equilibrium shape was found, natural modes of vibration were determined, by means of a linear modal analysis, around the deformed geometry.

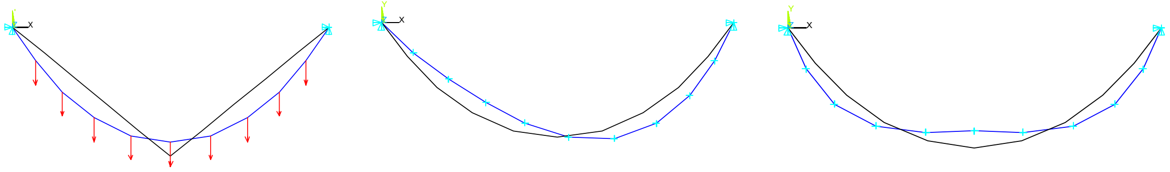

Figure 1: Freely hanging cable, under self-weight; equilibrium position,
starting from an arbitrary shape, and two modes of free vibrations

The initial geometry to start the form finding procedure of cables and membranes may be quite arbitrary. In some cases, however, Ansys pre-processing routines enable to begin analysis with forms that already constitute equilibrium configurations, under prestressing loads. These are the cases of the models shown in figure 2, for which the initial mesh geometries already conform to hyperbolic paraboloid surfaces. As a consequence, when imposing a uniform prestress field, (almost) zero displacements resulted. Also stresses showed very little variation, results being rather insensible to mesh updating. Figure 2(c) shows results for the rigid border paraboloid under wind pressure acting along the model's ridge direction (figure 2(b)). Realistic values were adopted to material properties. The structure displayed small displacements, about 5cm maximum (figure 2(c)) with a magnification of 100 times.

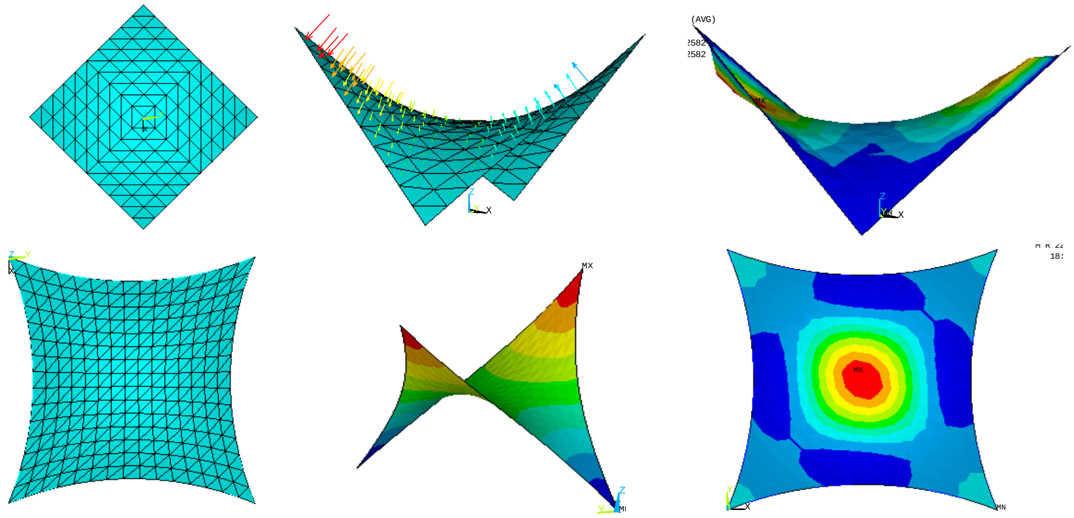

Figure 2: Rigid and flexible border hypars, under prestress and wind loads

Further complexity arises when an elastic boundary is added to the structure. That is the case of the membrane shown figure 2(d), connected to border cables anchored at their extremities. If the boundary is excessively flexible, the model tends to relax the prestressing field, making convergence harder to achieve. Figure 2(f) shows converged maximum principal stresses (ranging from 0,997MPa to 1,0MPa, after relaxing a 1MPa prestress field), figure 2(e) shows vertical level contours for the equilibrium surface.

In the previous examples, minimal equilibrium shapes were readily generated, both to stiff and elastic boundary paraboloids. In many cases, it can indeed be advantageous to work with minimal forms, since they correspond to the minimum area to a given contour, either stiff or flexible. On the other hand, since these surfaces have zero mean curvature, they sometimes present undesirable flat regions, leading to low transversal stiffness and insufficient slopes to cope with drain raining water. In order to briefly inspect non-minimal surfaces, the conoid shown if figure 3 was studied. Figure 3(a) shows the conical geometry adopted to start the form finding process. An uniform 1MPa prestress field was imposed to the model. Since the cone is not a double curvature surface, it obviously can not constitute an equilibrium shape. However, different equilibrium shapes can be obtained by varying the membrane elastic module, the more peaked the shape as more flexible the material. Thus, figure 3(b) shows deformed configuration and maximum principal stresses for a very stiff membrane, with very small displacements, but a considerable stress variation (radial tensions ranging from 3,2MPa at the top to 0,4MPa at base of the cone).

Reducing the material's elastic modulus, more uniform stress fields are obtained. However, prescription of lower elastic moduli allows larger geometric variations, possibly degrading numerical convergence. An alternative is to adopt a moderate elastic modulus, and then proceed to successive updates of the nodal coordinates, as shown in figure 3(e). Nevertheless, the process may still lead to excessive element distortion. Mesh adaptation is a last resource, not tested insofar. Figure 3(f) shows a quite slender geometry, with a stress field varying from 0,94MPa to 2,0MPa, still far from a minimal surface. However, cutting the conoid at half its height, a shape with a reasonably uniform stress field would be obtained.

Taking one of the intermediate equilibrium forms (figure 3(d)), it was investigated the capability of the program to flatten a previously defined cutting pattern by means of a structural analysis. A natural choice for a cutting pattern is that indicate in figure 3(g). After selection of this part of the model, convenient vertical displacements are imposed to nodes, dragging them to a single horizontal plane, as shown in figure 3(h). Figure 3(i) shows the flat pattern, also indicating vertical level contours.

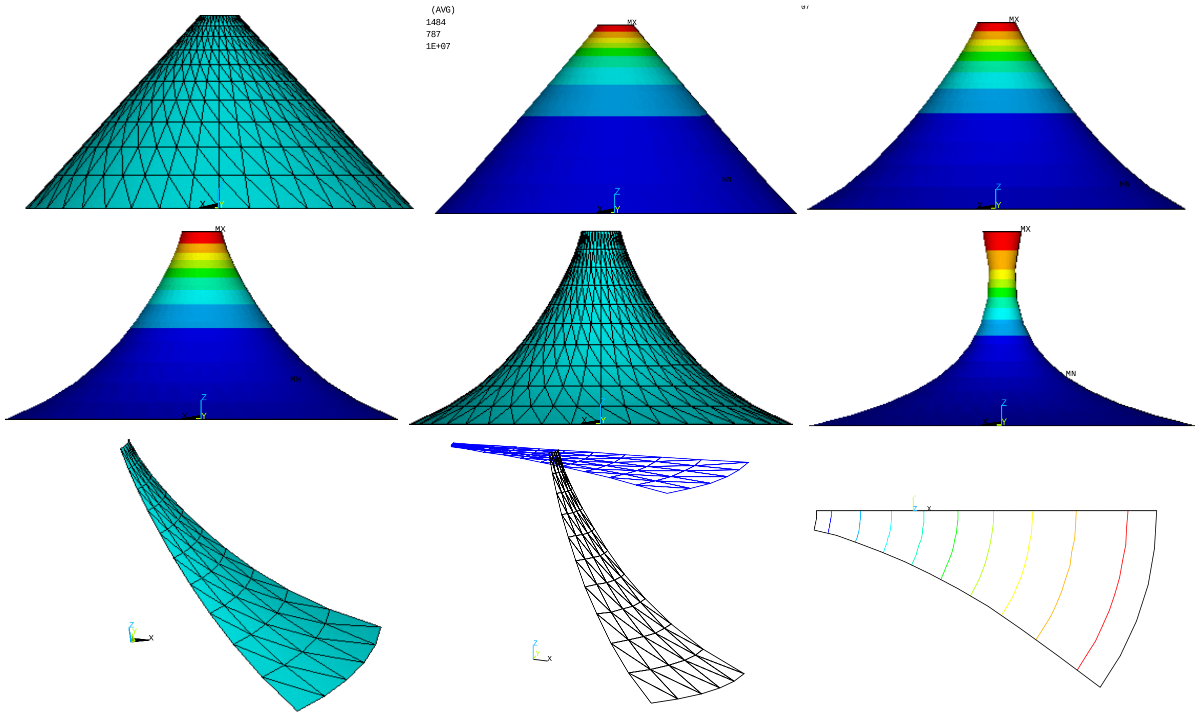

Figure 3: Non-minimal conoid; initial conical mesh, conoidal equilibrium configurations under prestressing, for different elastic modulus; mesh updating applied to an intermediate form; non-minimal conoidal cutting pattern; structural planification, flat pattern with vertical level contours.

3 ASYMMETRIC CONOIDAL SURFACE

A more exacting example is given by a 70m long, 50m wide double asymmetric conoidal membrane, hanged by two masts, 20m tall, articulated at their lower ends, and connected to the membrane by two upper rings, 2m in diameter, 17m height. Border anchorages heights vary between 4m and 6m. The original finite element mesh is shown in figure 4(a). The membrane was modeled with SHELL41 elements working exclusively in tension. The elastic modulus of the membrane material was kept arbitrarily low. Cables and masts were modeled with LINK8 elements. Stresses obtained in the first analysis were physically meaningless. After updating mesh geometry (figure 4(b)), a realistic elastic modulus was defined, and analysis restarted, with an uniform 4kN/m prestress. Figure 4(c) shows displacement norms along the structure (maximum 17cm), whilst figure 4(e) shows the resulting first principal stress field. A maximum of 16.2kN/m (or 16.2MPa) was reached, around the upper supporting rings. A relatively low average stress of about 3,5kN/m characterizes most of the membrane surface, following the current trend to reduce membrane prestress, leaving service loads to be resisted by membrane displacements, rather than local changes in stress. Figure 4(h) shows tractions in the border cables due to prestressing (maximum traction about 120kN).

Response to wind loads were then verified. Basic wind velocity and pressure were determined according to the Brazilian national wind code. Pressure distribution along the membrane, for wind blowing in different directions, was roughly guessed, since the code does not make direct reference to this kind of shape. Displacements of the membrane for a wind acting along the positive Y direction are shown, exaggerated, in figure 4(g). Maximum displacements of about 1.6m were obtained, in the central region of the membrane. The model did not accuse any slackening under wind loads, with maximum stresses reaching about 30 kN/m, well bellow the rupture stresses of 100kN/m reported by the membrane manufacturer.

Convergence was more difficult than in previous analyses, because pressure loads applied in a structure with moving boundaries configure a non-conservative system, resulting in asymmetric tangent stiffness matrices. If this asymmetry was disregarded, the analyses would converge quite slowly, or not converge at all.

Choosing a mapped finite element mesh allowed to easily determine suitable cutting patterns over the surface, by simple selection of elements. The 64 different strips composing half the membrane surface are shown in figure 4(d). Some of these strips are more than 30m long, with a maximum width of 2.45m, and minimum width of 10cm. A structural procedure to flatten each one of these strips was deemed too cumbersome and unnecessary, since each strip had only one triangular element across width. Alternatively, a geometric procedure was developed, using Ansys APDL language, to rotate each triangular element into a single plane, allowing for compensation of the element side lengths, according to their stress level. Figure 4(f) shows the flattened set.

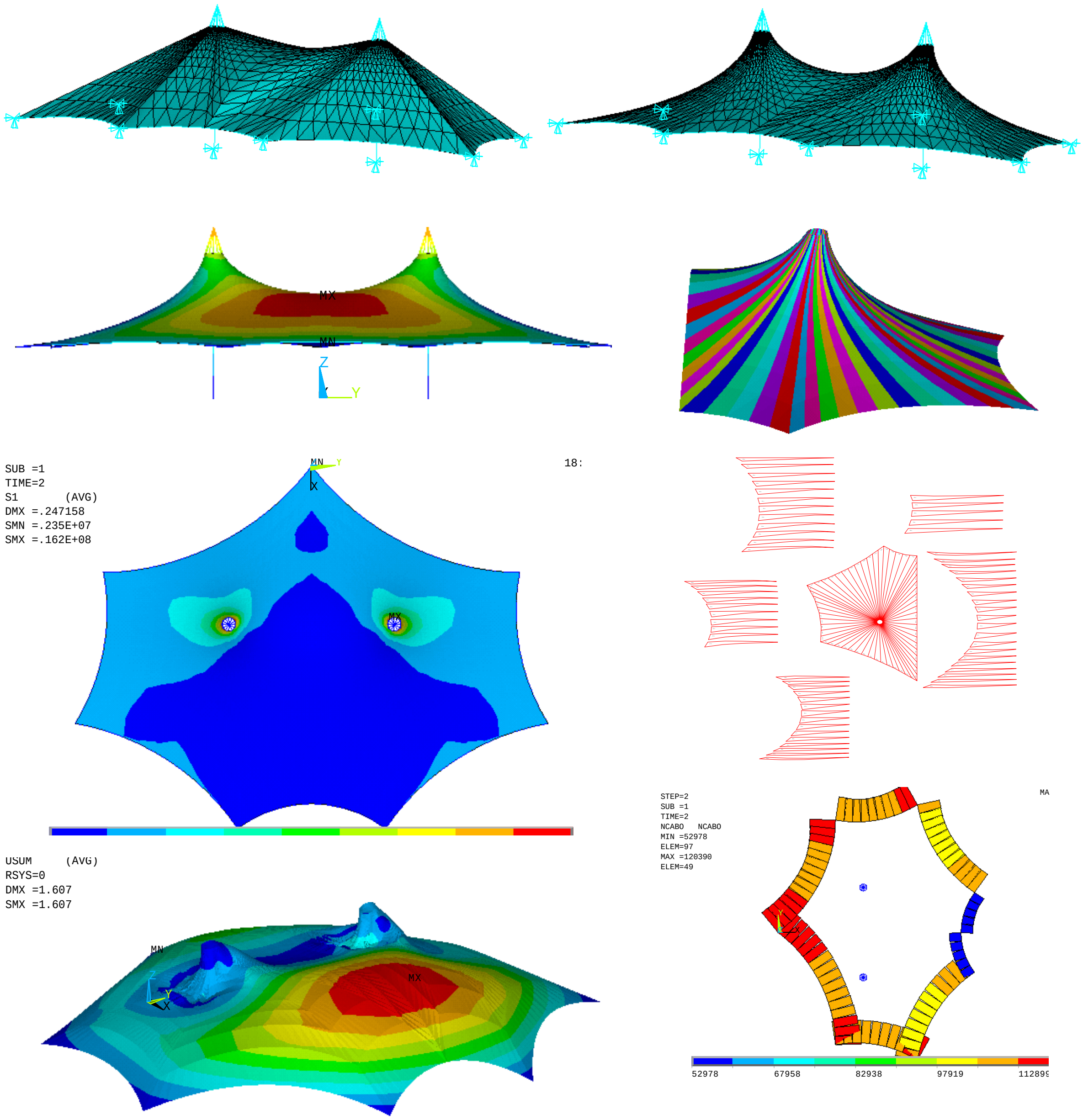

Figure 4: Analysis and design of a double asymmetric conoidal membrane

Figure 5(a) shows a reconstructed, 1:50 scaled paper model. Gluing of adjacent strips started always from the border end, progressing to the upper ring end. An odd “torsion” of the gluing lines was observed, close to the top rings. To circumvent this problem, it would be necessary to refine the strip discretization in the width direction, and to proceed with structural flattening, but that was again deemed unnecessary, since this torsion of the cutting lines did not jeopardize membrane functionality or aesthetics.

Finally, a more realistic, 1:10 model was built, in order to verify manufacturing, erection and prestressing procedures. The membrane was built by sewing strips of a light PVC fabric. Masts and border cables were made of steel. Figures 5(b,c,d) show some photographs of this last model.

As a next step in the analysis of this structure, the 1:50 paper model will be stiffened with acrylic resin and tested in a wind tunnel. Uniaxial and biaxial tests with the membrane actual fabric material, considering weldments, have also been conducted, and full scale models of the connection details are under way. Results of these experimental works will be reported in the near future.

Figure 5: Paper (1:50 scale) and PVC (1:10 scale) models of the double asymmetric conoidal membrane

4 FINAL CONSIDERATIONS

In a preliminary inspection, it was found that general purpose finite element programs are capable of carrying out the main phases of the design and analysis of tension structures, like cable nets and membranes. This paper presents different examples, starting from very simple cases, aimed to validate analysis procedures, progressing towards more realistic design situations, with models being actually constructed to verify the full design procedure. Overall results were deemed quite satisfactory. Nevertheless, there is a lot yet to be further explored: optimization routines, contact elements to find geodesic cutting lines, determination of wind pressures through fluid flow analyses, the study of flutter induced by fluid-structure interactions, or of the thermal behavior of the membrane micro-environment, are some among many interesting topics on the tension structures design, towards which the GPP wave promisingly.

REFERENCES

[1] W.C. Knudson. “Recent advances in the field of long span tension structures.” *Eng. Struct.* **13** 164–177 (1991).

[2] B. Tabarrok & Z. Qin. “Dynamic analysis of tension structures”, *Computers & Structures* **62** (3) 467–474 (1997).

[3] D. Campbel et al. “The Unique Role of Computing in the Design and Construction of Tensile Membrane Structures”. *American Society of Civil Engineers Second Civil Engineering Automation Conference.* Proceedings. New York, NY, 1991.

[4] Ansys Rev. 6.0. Ansys Inc. Global Headquarters. Southpointe, 275 Technology Drive, Canonsburg, PA, 15317, USA.