Cable Elements in S-FRAME – Tips and Tricks

Cables are used in a variety of structures that employ their unique characteristics to create an efficient design. It is important to realize that, while similar in some ways, Cables and Beams do not behave alike. This Tips and Tricks will discuss the theory behind Cable Elements, and how you can properly implement them into your next model.

What is a Cable?

A Cable is a flexible structural member that is effective at resisting tensile forces. It helps to think of the Cable element as a piece of rope.   Its high flexibility makes it too unstable to provide compressive, bending or shear resistance. Common applications include: tension members in suspension bridges, guy-wires for tall, slender structures, and auxiliary elements in fabric structures. As Cables usually require large deformations in order to reach equilibrium, the use of linear analysis is generally ineffective and non-linear analysis must be performed. All Cables, except those completely vertical, develop a sagged shape dependent on their own self-weight, and prestress.

Modeling Considerations

As mentioned, Cables almost always have some sort of catenary sag, as well as some prestress force to avoid instability in the structure. Usually either the shape or the prestress is known, and the other must be found.

- If we begin with a known or assumed geometry (sag) we can iterate to converge on the initially unknown prestress while retaining the original geometry.
- If we start with a known or assumed stressed state, we can converge on an initially unknown geometry while retaining the original prestress.

Ultimately, we desire to model the equilibrium state of the Cable Structure where an analysis reveals no further deflections under the combined effects of the Cable's self-weight and prestress. We can speed up the iteration process by either making informed guesses of the unknown values, or use the "trick" method of adjusting the stiffness of the Cable Elements. These two alternatives are optional, but will reduce the number of steps required to reach equilibrium.

Demonstration of the "Trick" Method to Finding Initial Conditions

This demonstration assumes that you have already created your Cable Section, and Materials (ensure that the Cable's material has the Shear Modulus set to zero, as there are no shear effects in Cables). The following steps are performed assuming we know the initial pre-stress in the Cable, and want to find the geometry.

1. Select the Member Type tool, and choose Cable as the member type you want to use.
2. Set the initial prestress of the Cable to your desired amount (this is your known or assumed initial condition).
3. Create a Cable Element by clicking on two points in the Graphical Input Window.
4. Fix both end supports of the Cable.
5. Create a new load case and call it Self Weight. Ensure that the Gravitational Factor in the Z direction is set to -1.
6. At this point we will want to subdivide our Cable member into smaller elements, such that several straight elements can represent the curved shape reasonably well. For this example, the cable was modeled with 4 elements.
7. Perform a Non-linear Static Analysis
8. After the Analysis is complete, we can examine the Axial Force and Z-Deflection diagrams. You will notice that your Axial Force is not consistent with the prestress applied earlier. This is because the stress must increase to account for the self-weight load on the cable.

Axial Force (kN)


Z-Deflection (mm)

9. This is where the "trick" comes in. We will create a copy of our Cable material, and rename it "Low E". Much like its name suggests, we will reduce the Modulus of Elasticity of this new material by a factor of 1000. The magnitude of this reduction is just a rule of thumb. You can reduce it even more to further speed up the iteration process, however, reducing it too far will lead to instabilities. Keep this in mind if you run into errors during analysis.

10. Change your Cable's material to the new "Low E" material and run another analysis.

11. Re-examine your Axial Force and Z-Deflection, you'll notice your prestress assumption is much closer to your Axial Force results.
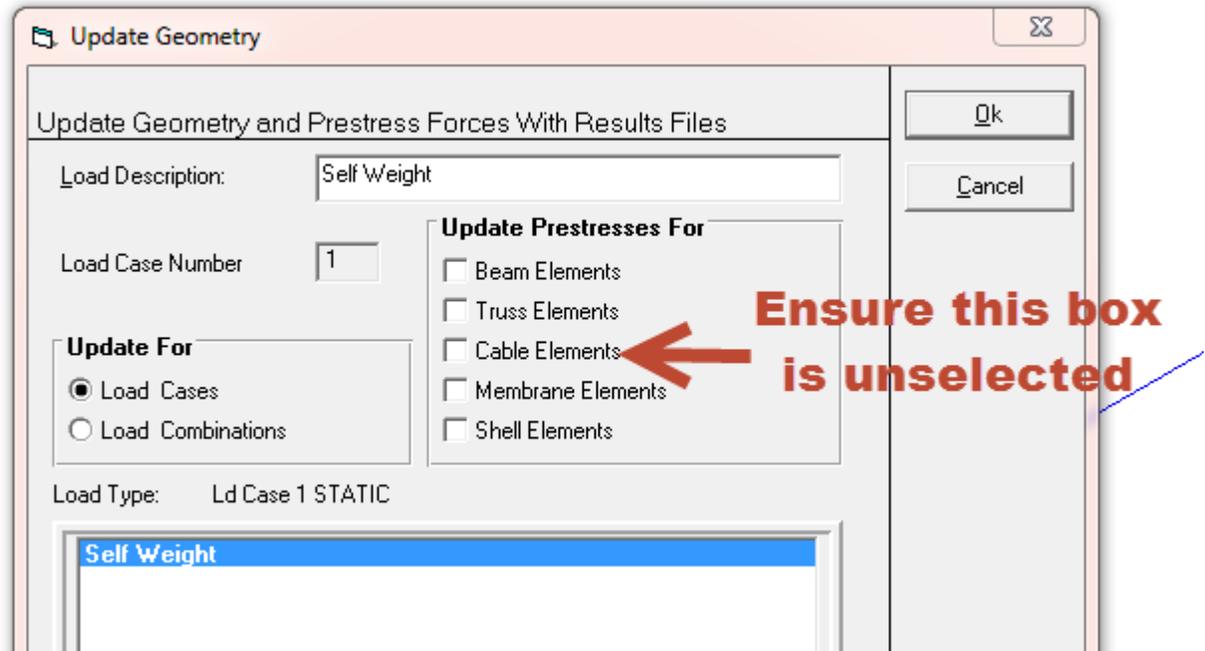
Axial Force (kN)

Z-Deflection (mm)
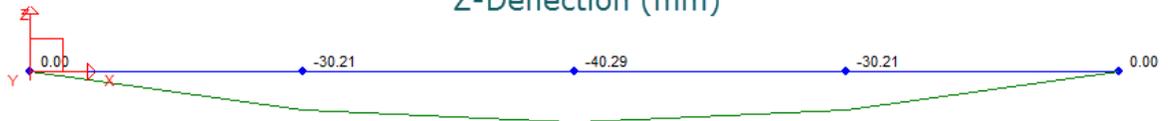
12. At this point, it is time to update the cable geometry with the deflected shape. In the Geometry Window, choose File > Update Geometry. In this dialog, we can choose which load case to use to replace our initial geometry. At this time, we also need to unselect "Update Prestress For" all element types.

13. After clicking ok, you may notice the cable sag slightly. We have successfully replaced our original geometry with deformed geometry and completed the first iteration. By clicking on any existing joint with the Joint Tool active, you can view its updated coordinates.
14. Next, this process should be repeated until the incremental deflection is very small. Check after each step that the prestress is preserved.
15. It may be tempting at this stage to consider the analysis complete, but there is one more step. We have been analyzing the cable with unrealistic material properties to help convergence. Once convergence is reached, however, we need to re-apply the original material and repeat the analysis one last time.
16. After the analysis, ensure your Axial Forces are as expected, and observe your deflection results; they should be very small as they represent the incremental deflection, not the total. For the total deflection, you will need to check the joint coordinates.

Axial Force (kN)

Z-Deflections (mm)

17. With the Cable shape **and** prestress now known, we can proceed to analyze the remainder of our structure, as well as add other loads to the Cables. Adding loads to the Cables will not require further iteration, as we are loading a structure in equilibrium.

Remember that the above procedure determined the geometry of a Cable for which we knew the desired axial force. Now let's say that we know the existing geometry of the Cable (our final geometry from above), and we wish to determine the prestress. We will continue with the model created at the end of the last example, except we will assume that we do not know the prestress at this point.

1. Rather than using the "Low E" material we created in the previous example, we will create a new material and call it "High E". We will adjust the order of magnitude of E accordingly, so let's make it 1000x stiffer than our original steel material.
2. Apply this new "High E" material to our Cable.
3. To avoid any analysis errors due to instability, we must apply an initial guess of our Cable's prestress. We will apply an initial value of 10kN in this case. If we do not apply a prestress our Cable Element will be unstable.
4. Run a Non-Linear Static Analysis , and view the Self Weight Load Case results for Axial Force. You should notice that the axial load after this step is very close to 50kN. As you probably remember this is the equilibrium prestress based on the geometry we have. You'll also see that our deflection results are very small, and are assumed negligible.

.

5. At this point we must update our Cable prestress based on our recent analysis result.
6. Use the Update Geometry tool again, and ensure that the option for Updating Prestresses for Cable Elements is selected.
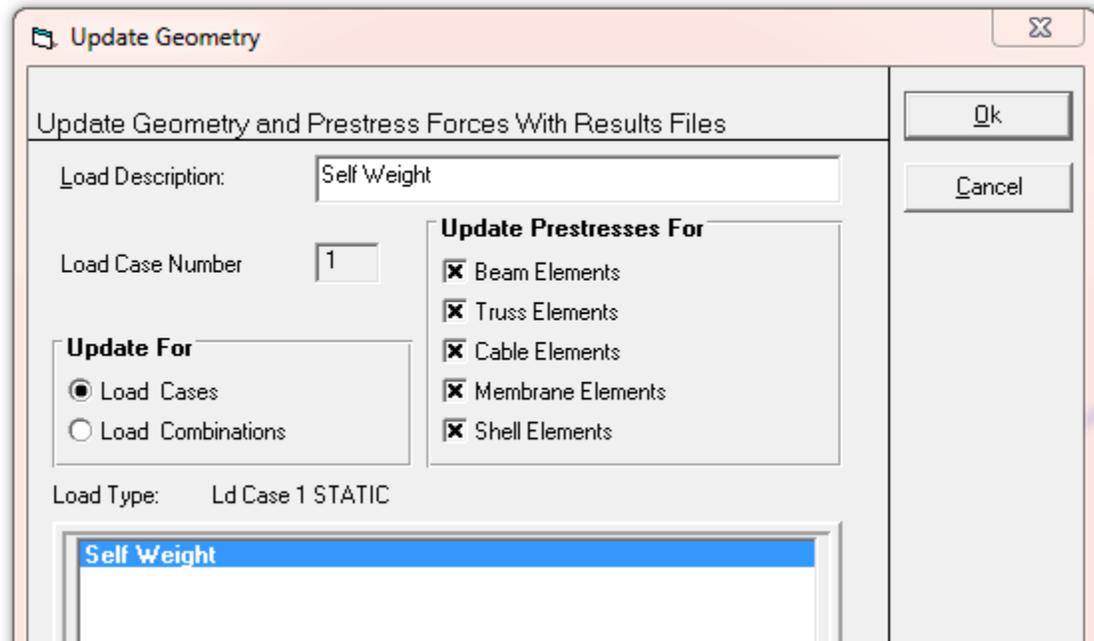
7. Reapply your original Cable Material, and run another Non-Linear Static Analysis. Your results should match those of the previous example.

For further explanation of Cable Elements please visit the Help System built into S-FRAME Software. Additional information can be found in the book Tension Structures: Behaviour & Analysis by John William Leonard.

6 of 6