

Tension and compression limits tips

On this page:

- [Modeling tension-only members](#)
- [Tension-only members in moving-load analysis](#)
- [Unattainable tension and compression limits](#)
- [See also](#)

Modeling tension-only members

[Tension-only](#) members may be modeled through the following process:

1. Select all [frame](#) objects which will only carry tension.
2. Set the compression limit to zero through Assign > Frame > Tension/Compression Limits.

Note that the assignment of tension/compression limits is a [nonlinear](#) analysis feature, and some applicable points, taken from [Context Help](#), include:

- Tension/compression limits will only be activated in a nonlinear static or nonlinear direct-integration [time-history](#) analysis [load case](#). In general, use a nonlinear analysis technique when tension/compression limits are present in the model.
- When linear analysis is performed using the initial stiffness of a member (stiffness at zero), computation will use the full axial stiffness (AE/L) as though no limits are assigned.
- When linear analysis uses the stiffness at the end of a nonlinear analysis [load case](#), axial frame behavior depends on whether or not the tension /compression limits are exceeded at the end of the case, conditions for which are described as follows:
 - **Limits exceeded.** Axial frame behavior is replaced with a sufficiently soft spring ($1e-6 \cdot EA/L$) which simulates zero stiffness.
 - **Limits not exceeded.** The software uses full axial stiffness (AE/L) as though no limits are assigned.

Tension-only members in moving-load analysis

The only method available to account for the nonlinear properties of members which are assigned [tension and compression limits](#) during [moving-load](#) analysis is step-by-step time-history moving-load analysis. Additional information is available in the [CSI Analysis Reference Manual](#) (Step-By-Step Analysis > Time-History Analysis, page 466).

Unattainable tension and compression limits

When tension and compression limits are specified, the program reduces axial stiffness of the element once the limits are reached as described above. If there is a path for load redistribution, the loads will be redistributed and the limits will be achieved. However, if there is no path for load redistribution (a simple example would be a single member loaded by axial tensile force with zero tension limit specified for the member), the limits may be exceeded. This is however consistent with implementation of this feature in the program.

See also

- [Tension-only elements in ETABS](#) article
- [Response-spectrum analysis and nonlinear properties](#) article