

Tension-only elements in ETABS

Tension-only elements can be modeled using all versions of [ETABS](#). The process is as follows:

1. Select the elements to be specified as tension-only.
2. Specify the tension limit (usually a limit higher than expected maximum tension capacity) and a compression limit of zero (or negative) through Assign > Frame/Line > Tension/Compression Limits.
3. Define and Run a [nonlinear](#) static or nonlinear [time-history](#) analysis. This is necessary since tension-only assignment is a nonlinear feature. See below on how to convert any load case or combination to nonlinear.

Any [load case](#) or [combination](#) may be converted to a nonlinear analysis as follows:

1. Select the case(s) through Define > Convert Combos to Nonlinear Cases
2. Add the nonlinear case(s) to a user-defined combination with a scale factor of 1.0.
3. For design, you can select the created nonlinear combination as design load combinations.
4. Perform design.

Please note that [response-spectrum](#) analysis only applies to linear systems. Further, response-spectrum loads cannot be converted to nonlinear cases.

See Also

- [Tension and compression limits](#) article
- [Response-spectrum analysis and nonlinear properties](#) article