

[Elements Reference](#) | [Part I, Element Library](#) |

LINK10

Tension-only or Compression-only Spar

MP ME ST PR PRN <><><><><><> PP <>

[Product Restrictions](#)

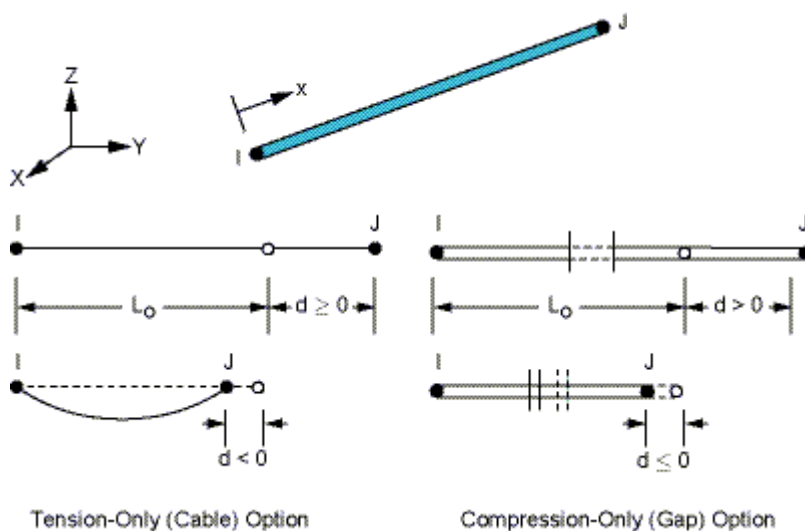
LINK10 Element Description

LINK10 is a 3-D spar element having the unique feature of a bilinear stiffness matrix resulting in a uniaxial tension-only (or compression-only) element. With the tension-only option, the stiffness is removed if the element goes into compression (simulating a slack cable or slack chain condition). This feature is useful for static guy-wire applications where the entire guy wire is modeled with one element. It may also be used in dynamic analyses (with inertia or damping effects) where slack element capability is desired but the motion of the slack elements is not of primary interest. This element is a line version of [SHELL41](#) with KEYOPT(1) = 2, the "cloth" option.

If the purpose of the analysis is to study the motion of the elements (with no slack elements), a similar element which cannot go slack, such as [LINK8](#) or [PIPE59](#), should be used instead. LINK10 should also not be used for static convergence applications where the final solution is known to be a taut structure but a slack condition is possible while iterating to a final converged solution. For this case either a different element should be used or the "slow dynamic" technique should be used if LINK10 is desired.

LINK10 has three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending stiffness is included in either the tension-only (cable) option or the compression-only (gap) option but may be added by superimposing a beam element with very small area on each LINK10 element. Stress stiffening and large deflection capabilities are available. See [LINK10](#) in the [Theory Reference for ANSYS and ANSYS Workbench](#) for more details about this element.

Figure 10.1 LINK10 Geometry



LINK10 Input Data

The geometry, node locations, and the coordinate system for this element are shown in [Figure 10.1: "LINK10 Geometry"](#). The element is defined by two nodes, the cross-sectional area, an initial strain or gap, and the isotropic material properties. The element x-axis is oriented along the length of the element from node I toward node J.

The initial strain in the element (ISTRN) is given by Δ/L , where Δ is the difference between the element length, L , (as defined by the I and J node locations) and the zero strain length, L_0 . For the cable option, a negative strain indicates a slack condition. For the gap option, a positive strain

indicates a gap condition (as shown in [Figure 10.1: "LINK10 Geometry"](#)). The gap must be input as a "per unit length" value.

Element loads are described in [Node and Element Loads](#). Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature T(J) defaults to T(I).

KEYOPT(2) is used to apply a small stiffness ($AE \times 10^{-6}/L$) across an open gap or to a slack cable to prevent unconstrained portions of the structure from "floating free" if the gap opens or the cable goes slack.

A summary of the element input is given in ["LINK10 Input Summary"](#). A general description of element input is given in [Element Input](#).

LINK10 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ

Real Constants

AREA - Cross-sectional area

ISTRN - Initial strain

If KEYOPT(3) = 0 and ISTRN is less than zero, the cable is initially slack.

If KEYOPT(3) = 1 and ISTRN is greater than zero, the gap is initially open.

Material Properties

EX, ALPX (or CTEX or THSX), DENS, DAMP

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Special Features

Nonlinear

Stress stiffening

Large deflection

Birth and death

KEYOPT(2)

Stiffness for slack cable:

0 --

No stiffness associated with slack cable

1 --

Small stiffness assigned to slack cable for longitudinal motion

2 --

Small stiffness assigned to slack cable for both longitudinal and perpendicular motions (applicable only with stress stiffening)

KEYOPT(3)

Tension / compression option:

0 --

Tension-only (cable) option

1 --

Compression-only (gap) option

LINK10 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in [Table 10.1: "LINK10 Element Output Definitions"](#).

The axial force, stress, and strain in the element are printed. Only positive values are obtained with the cable option and negative values with the gap option. The element condition (tension or slack, compression or gap) at the end of this substep is indicated by the value of STAT. A general description of solution output is given in [Solution Output](#). See the [Basic Analysis Guide](#) for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [[ETABLE](#), [ESOL](#)]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 10.1 LINK10 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	<u>2</u>
STAT	Element status	<u>1</u>	<u>1</u>
TEMP	Temperatures T(I), T(J)	Y	Y
MFORX	Member force in the element coordinate system	Y	Y
SAXL	Axial stress	Y	Y
EPELAXL	Axial elastic strain	Y	Y
EPHAXL	Axial thermal strain	Y	Y
EPINAXL	Axial initial strain	Y	Y

1. Element status values:
 - 1 - cable in tension or gap in compression
 - 2 - cable slack or gap open
2. Available only at centroid as a ***GET** item.

[Table 10.2: "LINK10 Item and Sequence Numbers"](#) lists output available through the [ETABLE](#) command using the Sequence Number method. See [The General Postprocessor \(POST1\)](#) in the [Basic Analysis Guide](#) and [The Item and Sequence Number Table](#) in this manual for more information. The following notation is used in [Table 10.2: "LINK10 Item and Sequence Numbers"](#):

Name

output quantity as defined in the [Table 10.1: "LINK10 Element Output Definitions"](#)

Item

predetermined Item label for [ETABLE](#) command

E

sequence number for single-valued or constant element data

I,J

sequence number for data at nodes I and J

Table 10.2 LINK10 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SAXL	LS	1	-	-
EPELAXL	LEPEL	1	-	-
EPHAXL	LEPTH	1	-	-
EPINAXL	LEPTH	3	-	-
MFORX	SMISC	1	-	-
STAT	NMISC	1	-	-
OLDST	NMISC	2	-	-
TEMP	LBFE	-	1	2

LINK10 Assumptions and Restrictions

- The element length must be greater than zero, therefore nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The temperature is assumed to vary linearly along the length of the element.
- The element is nonlinear and requires an iterative solution.
- If ISTRN is 0.0, the element stiffness is included in the first substep.
- With the gap (compression-only) option, a positive axial displacement (in the element coordinate system) of node J relative to node I tends to open the gap.
- The solution procedure is as follows: The element condition at the beginning of the first substep is determined from the initial strain or gap input. If this value is less than zero for the cable option or greater than zero for the gap option, the element stiffness is taken as zero for this substep. If at the end of the substep STAT = 2, an element stiffness of zero is used for the next substep. If STAT = 1, the element stiffness is included in the next substep. No significant stiffness is associated with the cable option having a negative relative displacement or with the gap option having a positive relative displacement.
- If the element status changes within a substep, the effect of the changed status is included in the next substep.
- Nonconverged substeps are not in equilibrium.
- The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.
- Stress stiffening should always be used for sagging cable problems to provide numerical stability. Stress stiffening and large deflection effects may be used together for some cable problems (see the [Theory Reference for ANSYS and ANSYS Workbench](#)).

LINK10 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element

in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional.

- The DAMP material property is not allowed.
- The birth and death special feature is not allowed.

Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.
