Element 134

Three-dimensional Four-node Tetrahedron

This element is a linear isoparametric three-dimensional tetrahedron (see Figure 3-209). As this element uses linear interpolation functions, the strains are constant throughout the element. The element is integrated numerically using one point at the centroid of the element. This results in a poor representation of shear behavior. A fine mesh is required to obtain an accurate solution. This element should only be used for linear elasticity. The higher-order element 127 is more accurate, especially for nonlinear problems.

This element gives poor results for incompressible materials. This includes rubber elasticity, large strain plasticity, and creep, For such problems, use element type 157.

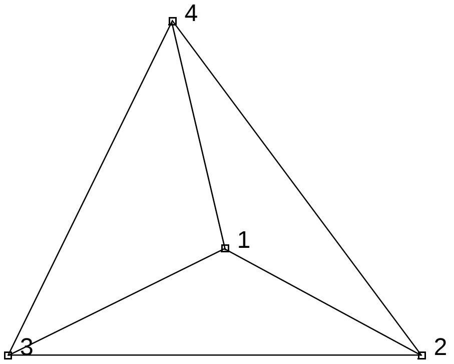

Figure 3-209 Form of Element 134

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of four nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

Quick Reference

Type 134

Four-nodes, isoparametric arbitrary distorted tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-209.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in the ROTATION A option.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in the ROTATION A option.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the one integration point.

For nonuniform surface pressure, force values need only be supplied for the one integration point on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to **Distributed Loads** in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{zx}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Notes: A large bandwidth results in a lengthy central processing time.

You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.