Element 127

Three-dimensional Ten-node Tetrahedron

This element is a second-order isoparametric three-dimensional tetrahedron. Each edge forms a parabola so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (Figure 3-195). This allows for an accurate representation of the strain field in elastic analyses.

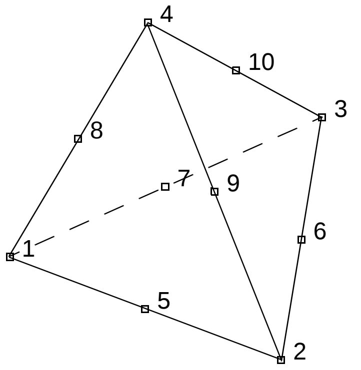

Figure 3-195 Form of Element 127

The stiffness of this element is formed using four-point integration. The mass matrix of this element is formed using sixteen-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 130. Element type 130 is also preferable for small strain incompressible elasticity.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten 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. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat (Mentat)) so that you need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element with the first point closest to the first node of the element (see Figure 3-196).

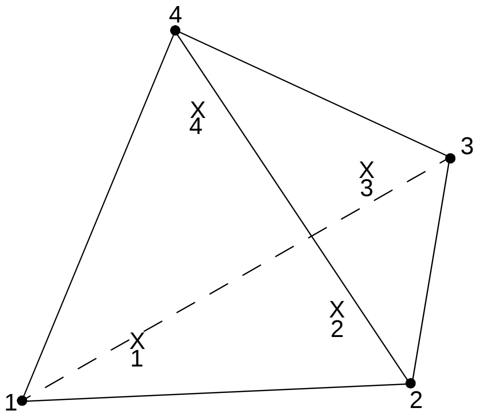

Figure 3-196 Element 127 Integration Plane

Quick Reference

Type 127

Ten nodes, isoparametric arbitrary distorted tetrahedron.

ConnectivitY

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-195.

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.

Load Type	Description
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 four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points 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 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = ε_{xy}
5 = ε_{yz}
6 = ε_{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 or four Gaussian integration points (see Figure 3-196).

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

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

Updated Lagrange Procedure and Finite Strain Plasticity

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

Note: Distortion of element during analysis can cause bad solutions. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used or entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.