Element 11

Arbitrary Quadrilateral Plane-strain

Element type 11 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, you need more of these lower-order elements than the higher-order elements such as types 27 or 54. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

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

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

For rubber materials with total Lagrange procedure, element type 80 can be used. This is slightly more expensive because of the extra pressure degrees of freedom associated with element type 80.

Quick Reference

Type 11

Plane-strain quadrilateral.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-13).

Geometry

The thickness is entered in the first data field (`EGEOM1`). Default thickness is one.

If a nonzero value is entered in the second data field (`EGEOM2`), the volume strain is constant throughout the element. That is particularly useful for analysis of approximately incompressible materials and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady-state solution.

If a one is entered in the third field, the assumed strain formulation is used.

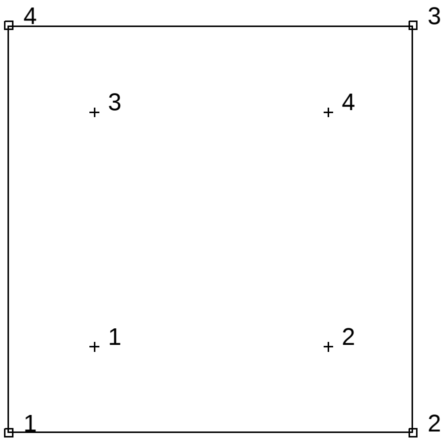

Figure 3-13 Gaussian Integration Points for Element Type 11

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.

Load Type	Description
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM user subroutine
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load; magnitude represents square angular velocity [rad/time]. Rotation axis is specified in the ROTATION A option.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Coriolis and centrifugal load; magnitude represents square of angular velocity [rad/time]. Rotation axis is specified in the ROTATION A option.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

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

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Two global degrees of freedom can be transformed into local coordinates.

Tying

Use the UFORMSN user subroutine.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 3-13.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration is recommended. (See **Geometry**.)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 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.

Assumed Strain

The assumed strain formulation is available to improve the bending characteristics of this element. Although this increases the stiffness assembly costs per element, it improves the accuracy.