Element 3

Plane Stress Quadrilateral

Element 3 is a four-node, isoparametric, arbitrary quadrilateral written for plane stress 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, one needs more of these lower-order elements than the higher-order elements such as 26 or 53. 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.

All constitutive models can be used with this element.

Note: To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 3

Plane stress quadrilateral.

Connectivity

Node numbering must be counterclockwise (see Figure 3-3).

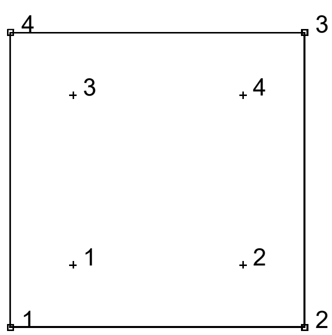

Figure 3-3 Plane Stress Quadrilateral

Geometry

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

The second field is not used.

In the third field, a one activates the assumed strain formulation.

Coordinates

Two global coordinates x and y.

Degrees of Freedom

1 = u (displacement in the global x direction)
2 = v (displacement in the global y direction)

Distributed Loads

Load types for distributed loads are defined 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 per 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.
* 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 user 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.

Load Type	Description
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 two magnitudes: the first value is gravity acceleration in x-direction; the second is gravity acceleration in the y-direction.
103	Coriolis and centrifugal loading; 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. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. 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 is as follows:

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

Note: Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See *Marc Volume A: Theory and User Information*, Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Output of Stresses is the same as for the Output of Strains.

Transformation

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

Tying

Use UFORMSN user subroutine.

Output Points

Output is available at the centroid or at the four numerical integration points shown in Figure 3-3.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of true stress and logarithmic strain in global coordinate directions. Thickness is updated if LARGE STRAIN is specified.

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 in-plane bending behavior. This increases the stiffness assembly costs per element and improves the accuracy.

Design Variables

The thickness can be considered a design variable for this element.