Element 26

Plane Stress, Eight-node Distorted Quadrilateral

Element type 26 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

Lower-order elements, such as type 3, are preferred in contact analyses.

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

All constitutive models can be used with this element.

Quick Reference

Type 26

Second-order, isoparametric, distorted quadrilateral. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 3-33.

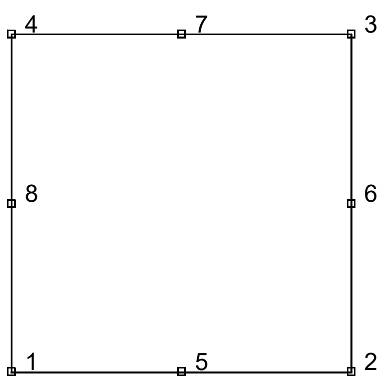

Figure 3-33 Nodes of Eight-node, 2-D Element

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, magnitude represents square of angular velocity [rad/time]. Rotation axis specified in the ROTATION A option.

Load Type (IBODY)	Description
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.

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. For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

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 or element integration points (see Figure 3-34 and Output Points) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = γ_{xy}, shear

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 Output of Strains.

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

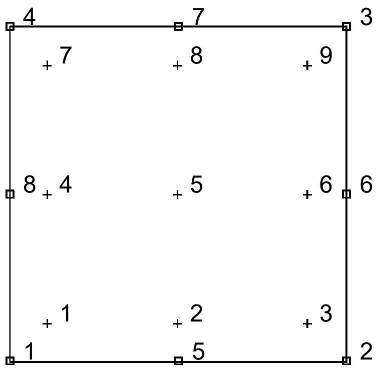

Figure 3-34 Integration Points of Eight-node, 2-D Element

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 3-34.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-34. This is the usual option for a second order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness is updated.

Note: Distortion of element during analysis can cause bad solution. Element type 3 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

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