Plane Stress & Plane Strain Elements

The Plane Stress and Plane Strain Elements incorporated in the Finite Element Library of the MIDAS Family Programs are the 3-node triangular and 4-node quadrilateral elements shown in Fig. 1. Finite Element Formulation of both element types is based on the isoparametric procedure (the element geometry and displacements are interpolated in the same way).

![2-D Finite Elements](image)

**Figure 1 2-D Finite Elements**

The nodal degrees of freedom (DOF’s) are illustrated in Fig. 1. The element geometry and displacement field are defined in terms of nodal coordinates and DOF’s by the following functions:

\[
\begin{align*}
    x &= \sum_{i=1}^{n} f_i(\xi, \eta) x_i \\
    y &= \sum_{i=1}^{n} f_i(\xi, \eta) y_i \\
    u &= \sum_{i=1}^{n} f_i(\xi, \eta) u_i \\
    v &= \sum_{i=1}^{n} f_i(\xi, \eta) v_i
\end{align*}
\]

where

- \(x_i, y_i\) = global x-y coordinates at \(i\)-th node
- \(u_i, v_i\) = displacements at \(i\)-th node along global \(X, Y\)-axes, respectively
- \(f_i(\xi, \eta)\) = interpolation function related to \(i\)-th node and defined in the natural coordinates variables \(\xi, \eta\) \((-1 \leq \xi \leq +1\) and \(-1 \leq \eta \leq +1\))
- \(\xi_i, \eta_i\) = natural coordinates of the \(i\)-th element node
- \(n\) = number of nodes in element
The nodal interpolation functions \( f_i(\xi, \eta) \) are of the following form:

**For triangular element**

\[
f_i(\xi, \eta) = \begin{cases} 
\frac{1}{2} (1 + \eta) & \text{for } i = 3 \\
\frac{1}{4} (1 + \xi \xi) (1 - \eta) & \text{for } i = 1, 2 
\end{cases}
\]

**For quadrilateral element**

\[
f_i(\xi, \eta) = \frac{1}{4} (1 + \xi \xi) (1 + \eta \eta) \text{ for } i = 1, 2, 3, 4
\]

Now, the strains at any point within element are expressed in terms of nodal displacements as,

\[
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} = 
\begin{bmatrix}
\frac{\partial u}{\partial x} & 0 \\
0 & \frac{\partial v}{\partial y} \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \frac{\partial u}{\partial y} \frac{\partial v}{\partial x}
\end{bmatrix} 
\begin{bmatrix}
u \\
v
\end{bmatrix} = 
\begin{bmatrix}
\frac{\partial}{\partial x} & 0 \\
0 & \frac{\partial}{\partial y} \\
\frac{\partial}{\partial y} & \frac{\partial}{\partial x}
\end{bmatrix} 
\begin{bmatrix}
f_1 & 0 & f_2 & 0 & \cdots & 0 & f_n \\
0 & f_1 & 0 & f_2 & \cdots & 0 & f_n
\end{bmatrix} 
\begin{bmatrix}
u_1 \\
u_2 \\
\vdots \\
u_n
\end{bmatrix}
\]

\[
\therefore \varepsilon = du = dfq = Bq
\]

where

\[
B = \text{strain-displacement matrix} \\
q = \text{vector of nodal displacements}
\]

Then, the stress-strain relation become,

\[
\sigma = D\varepsilon = DBq
\]

where, \( D \) is the elasticity matrix defining mechanical properties of the material. Also, the matrix \( D \) is the only difference that distinguishes plane stress elements from plain strain elements in finite element analysis. For an isotropic material, the \( D \) matrix takes following form:
for a plane stress element

\[ \mathbf{D} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \]

for a plain strain element

\[ \mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix} \]

in which

\[ E = \text{Young modulus} \]
\[ \nu = \text{Poisson's ratio} \]

Accordingly, the stiffness matrix and force vectors for a typical isoparametric 2D plane element are defined by the following integrals:

\[ \mathbf{K} = \int_{\Omega} \mathbf{B}^T \mathbf{D} \mathbf{B} \, dV = t \int_{A} \mathbf{B}^T \mathbf{D} \mathbf{B} \, dA \]

\[ \mathbf{p}_b = \int_{\Omega} \mathbf{f}^T \mathbf{b} \, d\Omega \]

\[ \mathbf{p}_0 = \int_{\Omega} \mathbf{B}^T \mathbf{D} \mathbf{\varepsilon}_0 \, d\Omega \]

where

\[ \mathbf{K} = \text{stiffness matrix} \]
\[ V = \text{volume of the element} \]
\[ A = \text{surface area of the element} \]
\[ t = \text{thickness of the element} \]
\[ \mathbf{p}_b = \text{nodal force vector due to distributed body forces} \mathbf{b} \]
\[ \mathbf{p}_0 = \text{nodal force vector due to initial strain} \mathbf{\varepsilon}_0 \]
\[ \Omega = \text{range of the integration (e.g. edge length for edge load, no integration for point load, volume for temperature load, etc.).} \]

Up to this moment we have reviewed the standard isoparametric formulation procedure, which is identical for both types of the 2D plane elements (3-node triangular and 4-node quadrilateral). It should be noted that the 3-node element presented here is in fact a degenerated form of the 4-node quadrilateral element. Formulation of the triangular element by collapsing the quadrilateral element allows evaluating the above integrals for both element types by use of the same standard 2D numerical integration
procedure, based on the Gauss-Legendre quadrature. Thus the numerical integration formulas used for 2D plane elements are:

\[
K = t \sum_{k=1}^{m} \sum_{j=1}^{n} W_{f} W_{k} B^{T} \left( \xi_{j}, \eta_{k} \right) \mathbf{DB} \left( \xi_{j}, \eta_{k} \right) \left| J \left( \xi_{j}, \eta_{k} \right) \right|
\]

\[
p_{b} = t \sum_{k=1}^{m} \sum_{j=1}^{n} W_{f} W_{k} f^{T} \left( \xi_{j}, \eta_{k} \right) \mathbf{b} \left( \xi_{j}, \eta_{k} \right) \left| J \left( \xi_{j}, \eta_{k} \right) \right|
\]

\[
p_{0} = t \sum_{k=1}^{m} \sum_{j=1}^{n} W_{f} W_{k} B^{T} \left( \xi_{j}, \eta_{k} \right) \mathbf{D} e_{0} \left( \xi_{j}, \eta_{k} \right) \left| J \left( \xi_{j}, \eta_{k} \right) \right|
\]

where:

- \( t \) = thickness of the element
- \( W_{i} \) = weighting factor of \( i \)-th integration point
- \( \xi_{j}, \eta_{k} \) = natural coordinates of \( i \)-th integration point
- \( \left| J \left( \xi_{j}, \eta_{k} \right) \right| \) = determinant of the Jacobian matrix
- \( m, n \) = number of integration points in direction of \( \xi \) and \( \eta \), respectively.

The appropriate order of numerical integration and corresponding locations of integration points used in the triangular and quadrilateral elements are shown in Table below.

**Table Gauss-Legendre Integration for 2D Plane Elements**

<table>
<thead>
<tr>
<th>Integration order</th>
<th>Location of integration points</th>
</tr>
</thead>
<tbody>
<tr>
<td>2×2</td>
<td><img src="image" alt="2x2 integration points" /></td>
</tr>
<tr>
<td>3×3</td>
<td><img src="image" alt="3x3 integration points" /></td>
</tr>
</tbody>
</table>
Here it should be noted that the compatible 4-node quadrilateral element based on standard *isoparametric* formulation does not produce accurate results in many cases for both displacements and stresses. In order to improve the performance of this element are added so called the *incompatible displacements modes*. Now we consider the procedure, which involves the incompatible modes. Taking into account the incompatible modes, the displacement field is defined as,

$$ u = \sum_{i=1}^{n} f_i(\xi, \eta) u_i + \alpha_1 P_1 + \alpha_2 P_2 $$

$$ v = \sum_{i=1}^{n} f_i(\xi, \eta) v_i + \alpha_3 P_1 + \alpha_4 P_2 $$

where $P_1, P_2$ are the extra shape functions related to the additional incompatible modes expressed as,

$$ P_1(\xi, \eta) = 1 - \xi^2 \quad \text{and} \quad P_2(\xi, \eta) = 1 - \eta^2 $$

and $\alpha_i (i = 1, 2, 3, 4)$ are so called “nodeless extra DOF’s”. Note that the extra shape functions permit a parabolic deformation along the element edge and improve the element stiffness performance.

Then, the strain-displacement matrix $\mathbf{B}$ can be presented as,

$$ \mathbf{B} = \begin{bmatrix} \mathbf{B}_C & \mathbf{B}_I \end{bmatrix} = \begin{bmatrix} \frac{\partial f_1}{\partial x} & 0 & \frac{\partial f_1}{\partial y} & 0 & \frac{\partial P_1}{\partial x} & 0 & \frac{\partial P_1}{\partial y} & 0 \\ 0 & \frac{\partial f_1}{\partial y} & 0 & \frac{\partial f_1}{\partial x} & 0 & \frac{\partial P_1}{\partial y} & 0 & \frac{\partial P_1}{\partial x} \\ \frac{\partial f_n}{\partial y} & \frac{\partial f_n}{\partial x} & \frac{\partial f_n}{\partial y} & \frac{\partial f_n}{\partial x} & \frac{\partial P_1}{\partial y} & \frac{\partial P_1}{\partial x} & \frac{\partial P_1}{\partial y} & \frac{\partial P_1}{\partial x} \end{bmatrix} $$

where, $\mathbf{B}_C$ is the strain-displacement matrix including terms of the original shape functions $f_i(\xi, \eta)$, and $\mathbf{B}_I$ is the matrix defining contribution of the incompatible modes.

Accordingly, the element stiffness matrix takes the following form:

$$ \mathbf{K} = \int_V \mathbf{B}^T \mathbf{D} \mathbf{B} dV = \int_V \begin{bmatrix} \mathbf{B}_C & \mathbf{B}_I \end{bmatrix}^T \mathbf{D} \begin{bmatrix} \mathbf{B}_C & \mathbf{B}_I \end{bmatrix} dV $$

$$ = \begin{bmatrix} \int_V \mathbf{B}_C^T \mathbf{D} \mathbf{B}_C dV & \int_V \mathbf{B}_C^T \mathbf{D} \mathbf{B}_I dV \\ \int_V \mathbf{B}_I^T \mathbf{D} \mathbf{B}_C dV & \int_V \mathbf{B}_I^T \mathbf{D} \mathbf{B}_I dV \end{bmatrix} = \begin{bmatrix} \mathbf{K}_{CC} & \mathbf{K}_{CI} \\ \mathbf{K}_{IC} & \mathbf{K}_{II} \end{bmatrix} $$
From the element equilibrium equations,

\[
\begin{bmatrix}
K_{cc} & K_{cl} \\
K_{lc} & K_{ll}
\end{bmatrix}
\begin{bmatrix}
u_c \\
\alpha
\end{bmatrix} = \begin{bmatrix}
p_c \\
0
\end{bmatrix}, \quad \alpha = \begin{bmatrix}
\alpha_1 \\
\alpha_2 \\
\alpha_3 \\
\alpha_4
\end{bmatrix}^T
\]

we define

\[
\alpha = -K_{ll}^\dagger K_{lc} u_c
\]

Thus the final stiffness matrix is obtained by static condensation of the incompatible modes as,

\[
K_F u_c = p_c
\]

where

\[
K_F = K_{cc} - K_{cl} K_{ll}^\dagger K_{lc}
\]

It should be noted that the incompatible modes are considered only in formulation of the stiffness matrix and are not included in the evaluation of body force vector, etc. Since the terms related to the incompatible modes are integrated at central point, \(\xi_0 = \eta_0 = 0\) must be substituted in the corresponding terms of the matrix \(B\).

The addition of the incompatible mode functions increase computational time required for generating the element stiffness matrix, however, the improvement in the element