Linear Strain Triangle and other types of 2D elements

By
S. Ziaei Rad
Linear Strain Triangle (LST or T6)

This element is also called \textit{quadratic triangular element}.

\textbf{Quadratic Triangular Element}
There are six nodes on this element: three corner nodes and three midside nodes. Each node has two degrees of freedom (DOF) as before. The displacements \((u, v)\) are assumed to be quadratic functions of \((x, y)\),

\[
\begin{align*}
  u &= b_1 + b_2 x + b_3 y + b_4 x^2 + b_5 xy + b_6 y^2 \\
  v &= b_7 + b_8 x + b_9 y + b_{10} x^2 + b_{11} xy + b_{12} y^2
\end{align*}
\]

where \(b_i\) \((i = 1, 2, \ldots, 12)\) are constants. From these, the strains are found to be,

\[
\begin{align*}
  \varepsilon_x &= b_2 + 2b_4 x + b_5 y \\
  \varepsilon_y &= b_9 + b_{11} x + 2b_{12} y \\
  \gamma_{xy} &= (b_3 + b_8) + (b_5 + 2b_{10}) x + (2b_6 + b_{11}) y
\end{align*}
\]
which are linear functions. Thus, we have the “linear strain triangle” (LST), which provides better results than the CST.

In the natural coordinate system we defined earlier, the six shape functions for the LST element are,

\[
\begin{align*}
N_1 &= \xi(2\xi - 1) \\
N_2 &= \eta(2\eta - 1) \\
N_3 &= \zeta(2\zeta - 1) \\
N_4 &= 4\xi \eta \\
N_5 &= 4\eta \zeta \\
N_6 &= 4\zeta \xi
\end{align*}
\]

in which \( \zeta = 1 - \xi - \eta \). Each of these six shape functions represents a quadratic form on the element as shown in the figure.
Displacements can be written as,

\[ u = \sum_{i=1}^{6} N_i u_i, \quad v = \sum_{i=1}^{6} N_i v_i \]

The element stiffness matrix is still given by \( \mathbf{B}^T \mathbf{E} \mathbf{B} \) but here \( \mathbf{B}^T \mathbf{E} \mathbf{B} \) is quadratic in \( x \) and \( y \). In general, the integral has to be computed numerically.
Linear Quadrilateral Element (Q4)
Linear Quadrilateral Element (Q4)

There are four nodes at the corners of the quadrilateral shape. In the natural coordinate system \((\xi, \eta)\), the four shape functions are,

\[
N_1 = \frac{1}{4} (1 - \xi)(1 - \eta), \quad N_2 = \frac{1}{4} (1 + \xi)(1 - \eta) \\
N_3 = \frac{1}{4} (1 + \xi)(1 + \eta), \quad N_4 = \frac{1}{4} (1 - \xi)(1 + \eta)
\]

Note that \(\sum_{i=1}^{4} N_i = 1\) at any point inside the element, as expected. The displacement field is given by

\[
u = \sum_{i=1}^{4} N_i \nu_i
\]

which are bilinear functions over the element.
**Isoparametric Element**

If we use the same parameters (shape functions) to express Geometry, we are using an *isoparametric* formulation.

\[
\begin{bmatrix}
  u \\
  v
\end{bmatrix} =
\begin{bmatrix}
  N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\
  0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4
\end{bmatrix}
\begin{bmatrix}
  u_1 \\
  v_1 \\
  u_2 \\
  v_2 \\
  u_3 \\
  v_3 \\
  u_4 \\
  v_4
\end{bmatrix}
\]

\[
x = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4 \\
y = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4
\]
Isoparametric Element

\[
\begin{align*}
\frac{\partial f}{\partial \xi} = & \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}
\end{bmatrix} \frac{\partial f}{\partial \eta} \\
\frac{\partial f}{\partial \eta} = & \begin{bmatrix}
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \\
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi}
\end{bmatrix} \frac{\partial f}{\partial \xi}
\end{align*}
\]

or

\[
\begin{bmatrix}
\frac{\partial f}{\partial \xi} \\
\frac{\partial f}{\partial \eta}
\end{bmatrix} = [J] \begin{bmatrix}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \xi}
\end{bmatrix} \Rightarrow \begin{bmatrix}
\frac{\partial f}{\partial \xi} \\
\frac{\partial f}{\partial \eta}
\end{bmatrix} = [J]^{-1} \begin{bmatrix}
\frac{\partial f}{\partial \xi} \\
\frac{\partial f}{\partial \eta}
\end{bmatrix}
\]

\[
[J] = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}
\end{bmatrix} = \begin{bmatrix}
J_{11} & J_{12} \\
J_{21} & J_{22}
\end{bmatrix}
\]

\[
\{\varepsilon\} = \begin{bmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial v}{\partial y} \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}
\end{bmatrix} = \frac{1}{\text{det} J} \begin{bmatrix}
J_{22} & -J_{12} & 0 & 0 \\
0 & 0 & -J_{21} & J_{11} \\
-J_{21} & J_{11} & J_{22} & -J_{12}
\end{bmatrix}
\]

A1
\[ \{ \varepsilon \} = [A1][A2]\{d\} = [B]\{d\} \]

\[ [K]^\varepsilon = t \int \int_{-1}^{1} [B]^T [E][B] \det J \, d\xi d\eta \]
Quadratic Quadrilateral Element (Q8)

This is the most widely used element for 2-D problems due to its high accuracy in analysis and flexibility in modeling.
**Quadratic Quadrilateral Element (Q8)**

\[ N_i = F_i(\xi, \eta)G_i(\xi, \eta) \]

**\( F_i(\xi, \eta) \)** Give a value of zero along the sides of the element That the given node does not contact

**\( G_i(\xi, \eta) \)** Select such that when multiply by \( F_i \), it will produce A value of unity at node \( i \) and a value of zero at other neighboring nodes.

**Example:** Consider N3

\[ F_3(\xi, \eta) = (1 + \xi)(1 + \eta) \]

\[ G_3(\xi, \eta) = c_1 + c_2\xi + c_3\eta \]

\[ G_3(1,0) = 0; \quad G_3(0,1) = 0; \quad N_3(1,1) = F_3(1,1)G_3(1,1) = 1 \]
Quadratic Quadrilateral Element (Q8)

\[ G_3(1,0) = 0 \Rightarrow c_1 + c_2 = 0 \]
\[ G_3(0,1) = 0 \Rightarrow c_1 + c_3 = 0 \]
\[ N_3(1,1) = 1 \Rightarrow 4(c_1 + c_2 + c_3) = 1 \]
\[ \therefore c_1 = -1/4 \]
\[ c_2 = 1/4 \]
\[ c_3 = 1/4 \]
\[ N_3 = \frac{1}{4}(1 + \xi)(1 + \eta)(\xi + \eta - 1) \]
**Quadratic Quadrilateral Element (Q8)**

There are eight nodes for this element, four corners nodes and four midside nodes. In the natural coordinate system \((\xi, \eta)\), the eight shape functions are,

<table>
<thead>
<tr>
<th>( N_1 )</th>
<th>( \frac{1}{4}(1 - \xi)(\eta - 1)(\xi + \eta + 1) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( N_2 )</td>
<td>( \frac{1}{4}(1 + \xi)(\eta - 1)(\eta - \xi + 1) )</td>
</tr>
<tr>
<td>( N_3 )</td>
<td>( \frac{1}{4}(1 + \xi)(1 + \eta)(\xi + \eta - 1) )</td>
</tr>
<tr>
<td>( N_4 )</td>
<td>( \frac{1}{4}(\xi - 1)(\eta + 1)(\xi - \eta + 1) )</td>
</tr>
<tr>
<td>( N_5 )</td>
<td>( \frac{1}{2}(1 - \eta)(1 - \xi^2) )</td>
</tr>
<tr>
<td>( N_6 )</td>
<td>( \frac{1}{2}(1 + \xi)(1 - \eta^2) )</td>
</tr>
<tr>
<td>( N_7 )</td>
<td>( \frac{1}{2}(1 + \eta)(1 - \xi^2) )</td>
</tr>
<tr>
<td>( N_8 )</td>
<td>( \frac{1}{2}(1 - \xi)(1 - \eta^2) )</td>
</tr>
</tbody>
</table>
Quadratic Quadrilateral Element (Q8)

Again, we have \( \sum_{i=1}^{8} N_i = 1 \) at any point inside the element. The displacement field is given by

\[
\begin{align*}
    u &= \sum_{i=1}^{8} N_i u_i, \\
    v &= \sum_{i=1}^{8} N_i v_i
\end{align*}
\]

which are quadratic functions over the element. Strains and stresses over a quadratic quadrilateral element are linear functions, which are better representations.

**Notes:**
- Q4 and T3 are usually used together in a mesh with linear elements.
- Q8 and T6 are usually applied in a mesh composed of quadratic elements.
- Quadratic elements are preferred for stress analysis, because of their high accuracy and the flexibility in modeling complex geometry, such as curved boundaries.
Example 3.2

A square plate with a hole at the center and under pressure in one direction.

The dimension of the plate is 10 in. x 10 in., thickness is 0.1 in. and radius of the hole is 1 in. Assume $E = 10 \times 10^6$ psi, $\nu = 0.3$ and $p = 100$ psi. Find the maximum stress in the plate.
Example 3.2

FE Analysis:
From the knowledge of stress concentrations, we should expect the maximum stresses occur at points A and B on the edge of the hole. Value of this stress should be around $3\rho$ (= 300 psi) which is the exact solution for an infinitely large plate with a hole.

We use the ANSYS FEA software to do the modeling (meshing) and analysis, using quadratic triangular (T6 or LST), linear quadrilateral (Q4) and quadratic quadrilateral (Q8) elements. Linear triangles (CST or T3) is NOT available in ANSYS.

The stress calculations are listed in the following table, along with the number of elements and DOF used, for comparison.
Example 3.2

Table. FEA Stress Results

<table>
<thead>
<tr>
<th>Elem. Type</th>
<th>No. Elem.</th>
<th>DOF</th>
<th>Max. $\sigma$ (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T6</td>
<td>966</td>
<td>4056</td>
<td>310.1</td>
</tr>
<tr>
<td>Q4</td>
<td>493</td>
<td>1082</td>
<td>286.0</td>
</tr>
<tr>
<td>Q8</td>
<td>493</td>
<td>3150</td>
<td>327.1</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>Q8</td>
<td>2727</td>
<td>16,826</td>
<td>322.3</td>
</tr>
</tbody>
</table>

**Discussions:**
- Check the deformed shape of the plate
- Check convergence (use a finer mesh, if possible)
- Less elements (~ 100) should be enough to achieve the same accuracy with a better or “smarter” mesh
- We’ll redo this example in next chapter employing the symmetry conditions.
Example 3.2

**FEA Mesh (Q8, 493 elements)**

**FEA Stress Plot (Q8, 493 elements)**
Concentrated load (point forces), surface traction (pressure loads) and body force (weight) are the main types of loads applied to a structure. Both traction and body forces need to be converted to nodal forces in the FEA, since they cannot be applied to the FE model directly. The conversions of these loads are based on the same idea (the equivalent-work concept) which we have used for the cases of bar and beam elements.

Traction on a Q4 element
Suppose, for example, we have a linearly varying traction $q$ on a Q4 element edge, as shown in the figure. The traction is normal to the boundary. Using the local (tangential) coordinate $s$, we can write the work done by the traction $q$ as,

$$W_q = t \int_0^L u_n(s)q(s)ds$$

where $t$ is the thickness, $L$ the side length and $u_n$ the component of displacement normal to the edge $AB$.

For the Q4 element (linear displacement field), we have

$$u_n(s) = (1 - s/L)u_{nA} + (s/L)u_{nB}$$

The traction $q(s)$, which is also linear, is given in a similar way,
Transformation of Loads

Thus, we have,

\[ q(s) = (1 - s / L)q_A + (s / L)q_B \]

and the equivalent nodal force vector is,

\[
W_q = t \int_0^L \left( \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} \begin{bmatrix} 1 - s / L \\ s / L \end{bmatrix} \right) \left( \begin{bmatrix} 1 - s / L & s / L \end{bmatrix} \begin{bmatrix} q_A \\ q_B \end{bmatrix} \right) ds
\]

\[
= \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} t \int_0^L \begin{bmatrix} (1 - s / L)^2 & (s / L)(1 - s / L) \\ (s / L)(1 - s / L) & (s / L)^2 \end{bmatrix} ds \begin{bmatrix} q_A \\ q_B \end{bmatrix}
\]

\[
= \begin{bmatrix} u_{nA} & u_{nB} \end{bmatrix} \frac{tL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{bmatrix} q_A \\ q_B \end{bmatrix}
\]
Transformation of Loads

For quadratic elements (either triangular or quadrilateral), the traction is converted to forces at three nodes along the edge, instead of two nodes.

Traction tangent to the boundary, as well as body forces, are converted to nodal forces in a similar way.

\[
\begin{bmatrix}
  f_A \\
  f_B \\
\end{bmatrix} = \frac{tL}{6} \begin{bmatrix}
  2 & 1 \\
  1 & 2 \\
\end{bmatrix} \begin{bmatrix}
  q_A \\
  q_B \\
\end{bmatrix}
\]

Note, for constant \( q \), we have,

\[
\begin{bmatrix}
  f_A \\
  f_B \\
\end{bmatrix} = \frac{qtL}{2} \begin{bmatrix}
  1 \\
  1 \\
\end{bmatrix}
\]
Transformation of Loads

\[ W_q = t \int_0^L u_n(s)q(s)ds \]

\[ s = 0 \quad @ \quad \xi = -1 \]
\[ s = L \quad @ \quad \xi = 1 \quad \Rightarrow \quad s = \frac{L}{2}(1 + \xi) \quad \Rightarrow \quad ds = \frac{L}{2} d\xi \]

\[ v = \frac{1}{2} (1 - \xi) v_1 + \frac{1}{2} (1 + \xi) v_2 \]
\[ q = \frac{1}{2} (1 - \xi) q_1 + \frac{1}{2} (1 + \xi) q_2 \]
Transformation of Loads

\[ W_q = t \int_{-1}^{1} \left[ v_1 \ v_2 \right] \begin{bmatrix} \frac{1}{2} (1 - \xi) \\ \frac{1}{2} (1 + \xi) \end{bmatrix} \begin{bmatrix} \frac{1}{2} (1 - \xi) \\ \frac{1}{2} (1 + \xi) \end{bmatrix} \frac{q_1}{q_2} \frac{L}{2} d\xi \rightarrow \begin{bmatrix} f_1 \\ f_2 \end{bmatrix} = \frac{tL}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \end{bmatrix} \]
**Transformation of Loads**

1- **Point Load** ➔ considered in a usual manner by having a Structural node at the point.

2- **Traction Force** ➔ As it was seen in the previous example First the force and the deflection along the side express by Use of shape functions and then numerical integration will Be used to calculate the equivalent nodal forces.

3- **Body Force** ➔ A Body force which is a distributed force Per unit volume, contribute to the global force vector $\mathbf{F}$. Assume $\{F\} = [f_x \ f_y]^T$ as constant within each element.

\[
\int_V u^T F dV = \sum_e d_e^T f_e^e \]

Where the 8*1 element body force is given by

\[
f_e^e = t_e \begin{bmatrix} \int_{-1}^{1} \int_{-1}^{1} N^T \det J d\xi d\eta \end{bmatrix} \begin{bmatrix} f_x \\ f_y \end{bmatrix}
\]
The stress in an element is determined by the following relation,

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} = E \begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} = EBd
\]

where \( \mathbf{B} \) is the strain-nodal displacement matrix and \( \mathbf{d} \) is the nodal displacement vector which is known for each element once the global FE equation has been solved.

Stresses can be evaluated at any point inside the element (such as the center) or at the nodes. Contour plots are usually used in FEA software packages (during post-process) for users to visually inspect the stress results.
The von Mises Stress

The von Mises stress is the effective or equivalent stress for 2-D and 3-D stress analysis. For a ductile material, the stress level is considered to be safe, if

$$\sigma_e \leq \sigma_Y$$

where \( \sigma_e \) is the von Mises stress and \( \sigma_Y \) the yield stress of the material. This is a generalization of the 1-D (experimental) result to 2-D and 3-D situations.

The von Mises stress is defined by

$$\sigma_e = \frac{1}{\sqrt{2}} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$$

in which \( \sigma_1, \sigma_2 \) and \( \sigma_3 \) are the three principle stresses at the considered point in a structure.
The von Mises Stress

For 2-D problems, the two principle stresses in the plane are determined by

\[
\sigma_1^p = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}
\]

\[
\sigma_2^p = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2}
\]

Thus, we can also express the von Mises stress in terms of the stress components in the xy coordinate system. For plane stress conditions, we have,

\[
\sigma_e = \sqrt{(\sigma_x + \sigma_y)^2 - 3(\sigma_x \sigma_y - \tau_{xy}^2)}
\]

Averaged Stresses:
Stresses are usually averaged at nodes in FEA software packages to provide more accurate stress values. This option should be turned off at nodes between two materials or other geometry discontinuity locations where stress discontinuity does exist.
**Discussions**

1) **Know the behaviors of each type of elements:**
   - T3 and Q4: linear displacement, constant strain and stress;
   - T6 and Q8: quadratic displacement, linear strain and stress.

2) **Choose the right type of elements for a given problem:**
   - When in doubt, use higher order elements or a finer mesh.

3) **Avoid elements with large aspect ratios and corner angles:**
   
   \[
   \text{Aspect ratio} = \frac{L_{\text{max}}}{L_{\text{min}}}
   \]

   where \(L_{\text{max}}\) and \(L_{\text{min}}\) are the largest and smallest characteristic lengths of an element, respectively.

---

**Elements with Bad Shapes**  **Elements with Nice Shapes**
**Discussions**

4) **Connect the elements properly:**
Don’t leave unintended gaps or free elements in FE models.

*Improper connections (gaps along AB and CD)*