University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-03-01

CONTINUUM ELEMENTS

Lecture Outline

1. **Introduction**
2. **2D Continuum Elements**
3. **Axisymmetric Continuum Elements**
4. **3D Continuum Elements**
5. **Summary of Key Points**

3.1 Introduction

- In continuum elements, the degrees of freedom are the **displacement components**.
- An approximation for the **displacement function** is assumed within each element, e.g. a linear or a quadratic variation.
- The assumptions used to model **2D, axisymmetric and 3D** continuum problems are discussed here.
- The FE formulation for **2D linear triangular elements** is presented.

3.2 2D Continuum Elements

2D plane stress or plane strain?

There is no such problem as a truly 2D problem; all 2D solutions are approximations of 3D solutions, and the z-direction is not ignored in the approximations.

(a) 2D Plane stress

- 'Thin' geometries in the z-direction
- Stress across the thickness is neglected (i.e. $\sigma_{zz} = 0$),
- BUT strain ε_{zz} is not zero.

(b) 2D Plane strain

- Very 'thick' geometries in the z-direction
- Strain across the thickness is neglected (i.e. $\varepsilon_{zz} = 0$),
- BUT the stress σ_{zz} is not zero.
- The x-y section is remote from the ends where $z = \pm\infty$.

3D Stress-Strain Relationships (Hooke's Law)

Stress-strain relationships are often called *"Constitutive Equations"*. For isotropic linear elastic materials with thermal strain, the following 3D stress-strain equations (Hooke's law) can be used:

$$\varepsilon_{xx} = \frac{1}{E}\left[\sigma_{xx} - \nu\left(\sigma_{yy} + \sigma_{zz}\right)\right] + \alpha\left(\Delta T\right)$$

$$\varepsilon_{yy} = \frac{1}{E}\left[\sigma_{yy} - \nu\left(\sigma_{xx} + \sigma_{zz}\right)\right] + \alpha\left(\Delta T\right)$$

$$\varepsilon_{zz} = \frac{1}{E}\left[\sigma_{zz} - \nu\left(\sigma_{xx} + \sigma_{yy}\right)\right] + \alpha\left(\Delta T\right)$$

$$\varepsilon_{xy} = \frac{1}{2\mu}\sigma_{xy}$$

$$\varepsilon_{xz} = \frac{1}{2\mu}\sigma_{xz}$$

$$\varepsilon_{yz} = \frac{1}{2\mu}\sigma_{yz}$$

where
E = Young's modulus (units: N m^{-2})
ν = Poisson's ratio (no units)
μ = Shear modulus (units: N m^{-2})
α = Coefficient of thermal expansion (units: per ^{o}C)
ΔT = Temperature change from a reference value (units: ^{o}C)

The shear modulus μ is defined as follows:

$$\mu = \frac{E}{2\left(1 + \nu\right)}$$

3D Stress-Strain Relationships (Hooke's Law)

In matrix form, this becomes:

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \\ 2\varepsilon_{12} \end{bmatrix} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{23} \\ \gamma_{13} \\ \gamma_{12} \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2+2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 2+2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 2+2\nu \end{bmatrix} \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{bmatrix}$$

Where $\gamma_{ij} = 2\varepsilon_{ij}$ is the engineering shear strain, the inverse is written as:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \\ 2\varepsilon_{12} \end{bmatrix}$$

2D Stress-Strain Relationships

Plane stress $\sigma_{33} = \sigma_{13} = \sigma_{31} = \sigma_{23} = \sigma_{32} = 0$

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{bmatrix}$$

Plane strain $\varepsilon_{33} = \varepsilon_{13} = \varepsilon_{31} = \varepsilon_{23} = \varepsilon_{32} = 0$

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{bmatrix}$$

Step 1: Define the element and the shape functions

•Since displacement is assumed to vary linearly over the element, the strain (which is a differential of the displacement) is therefore constant over the element.

•This element is often called a constant strain triangle (CST) since the strain (and therefore the stress) is constant per element.

•Alternatively, a quadrilateral 4-node straight-sided linear element can be used.

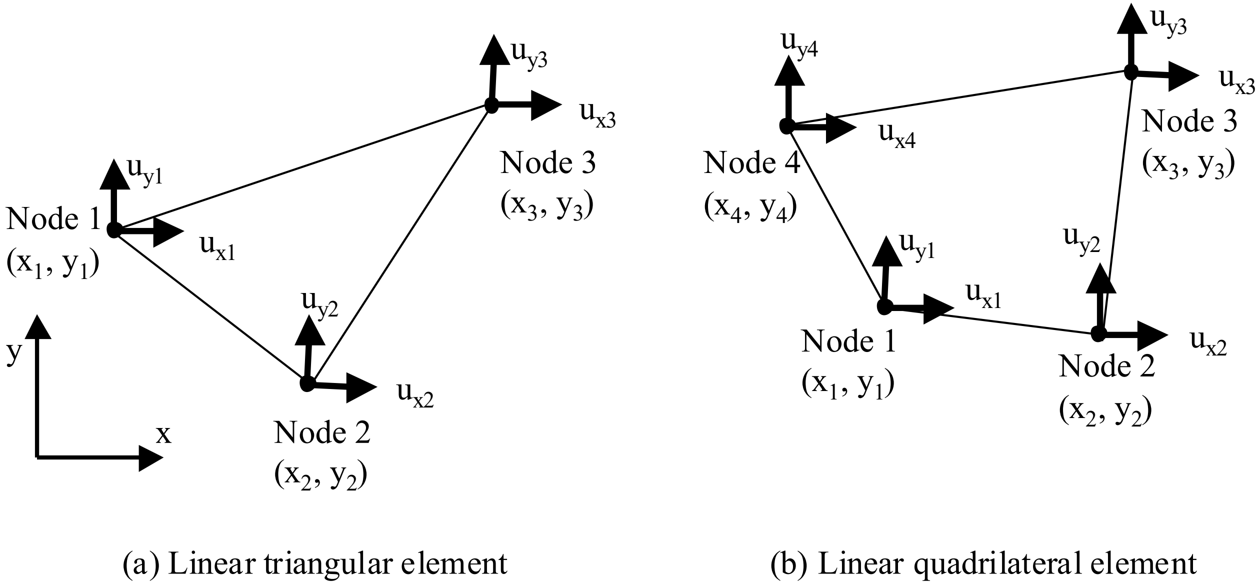


Figure 2: 2D triangular and quadrilateral elements

Assuming a linear variation of displacement over each element, an expression for the displacement in terms of x and y can be written as follows:

$$u_x(x,y) = C_1 + C_2\,x + C_3\,y$$
$$u_y(x,y) = C_4 + C_5\,x + C_6\,y$$

where C_1 to C_6 are six constants which can be expressed in terms of the coordinates of the nodes as follows:

$$\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_1 & y_1 \\ 1 & x_2 & y_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_2 & y_2 \\ 1 & x_3 & y_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_3 & y_3 \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \\ C_6 \end{bmatrix}$$

These equations can be abbreviated to:

$$[u_e] = [A][C]$$

where $[u_e]$ is the displacement vector of the element, $[A]$ is a "*coordinate matrix*", and $[C]$ is the vector of the constant C_1 to C_6. Equation (4) can be solved to determine the $[C]$ constants as follows:

$$[C] = \left\lfloor A^{-1} \right\rfloor [u_e]$$

Step 2: Satisfy the material law (constitutive equations)

Using the strain-displacement definitions, the element strain can be determined by differentiating the displacements, as follows:

$$\varepsilon_{xx} = \frac{\partial u_x}{\partial x} = C_2$$

$$\varepsilon_{yy} = \frac{\partial u_y}{\partial y} = C_6$$

$$\varepsilon_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} = C_3 + C_5$$

The element strain vector, $[\varepsilon_e]$, can then be written in matrix form as follows:

$$[\varepsilon] = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \\ C_6 \end{bmatrix}$$

or, in more concise form:

$$[\varepsilon] = [X][C]$$

Substituting $[C]$ results in the following expression of element strain in terms of displacement:

$$[\varepsilon] = [X]\left[A^{-1}\right][u_e] = [B][u_e]$$

where $[B]$ is a "*dimension matrix*".

To derive stress-strain relationships, Hooke's law can be used and re-arranged such that the stresses can be written in terms of strains, as follows:

$$[\sigma] = [D][\varepsilon]$$

where $[D]$ is a "*material property matrix*". Note that matrix $[D]$ is symmetric, i.e. $[D]^T = [D]$.

Therefore, the stress vector $[\sigma]$ can be written in terms of the displacement vector $[u_e]$ as follows:

$$[\sigma] = [D][B][u_e]$$

Step 3 : Derive the element stiffness matrix

The total potential energy (*T.P.E.*) of the element can be expressed as follows:

$$T.P.E. = \int_v \frac{1}{2} [\sigma]^T [\varepsilon] dv - [u_e]^T [F_e]$$

where v is the volume of the element.

Substituting for the stress and strain results in:

$$T.P.E. = \int_v \frac{1}{2} \left([D][B][u_e] \right)^T \left([B][u_e] \right) dv - [u_e]^T [F_e]$$

$$= \int_v \frac{1}{2} [u_e]^T [B]^T [D] [B] [u_e] dv - [u_e]^T [F_e]$$

Using the **principle of minimum total potential energy**, the differential of T.P.E. with respect to the displacement $[u]$ must be zero, i.e.

$$\frac{\partial(T.P.E.)}{\partial[u]} = 0 = \int_v [B]^T [D][B][u_e]\, dv - [F_e]$$

which can be expressed in terms of the element stiffness matrix $[k_e]$, as follows:

$$[k_e][u_e] = [F_e]$$

where

$$[k_e] = \int_v [B]^T [D][B]\, dv$$

Note that to compute the element stiffness, the **integration over the volume** has to be performed numerically over each element.

Step 4: Assemble the overall stiffness matrix

As in the pin-jointed elements, the final global (assembly) system of equations can be arrived at as follows:

$$[K]_{Assembly}\ [u]_{Assembly} = [F]_{Assembly}$$

For a given problem with N nodes, with each node having two degrees of freedom (here u_x and u_y), the $[K]_{Assembly}$ matrix is of size ($2N$ x $2N$), while the global $[u]_{Assembly}$ and $[F]_{Assembly}$ vectors are of size ($2N$ x 1).

The summation process is performed only over elements which share a particular node. Therefore, the overall $[K]$ is **sparsely populated**, i.e. with very few non-zero coefficients.

Example

For a $[K]$ matrix of size 1000 x 1000 (i.e. 500 nodes), assuming no more than, say, 9 elements can be connected to a given node, then the maximum number of non-zero coefficients in any given row of 1000 coefficients is 18.

Step 5: Apply the boundary conditions and external loads

To obtain a unique solution for the displacement at every node, some constraints (boundary conditions) and loads must be specified at the surface nodes.

Step 6: Solve the equations

Since the $[K]$ matrix is sparsely populated, storing the full $[K]_{Assembly}$ matrix (with all the zero coefficients) is very wasteful of computer storage space.

Special techniques can be used to "condense" the $[K]_{Assembly}$ matrix such that only the non-zero coefficients are stored, resulting in a substantial reduction in the storage requirements.

These techniques usually employ routines to minimise the size of the "*bandwidth*" (the maximum number of non-zero elements in a given row).

Step 7: Compute other variables

The strains can be obtained by differentiating the displacements over each element.

The stresses can then be derived from the strains using the material law (the constitutive equations).

3.3 Axisymmetric Continuum Elements

- Axisymmetric problems may be viewed as 2D (flat) geometries, but the Cartesian *x* and *y* directions are replaced by the radial (*r*) and axial (*z*) direction, as shown in Figure 3.

- Axisymmetric geometries, or bodies of revolution, are formed by rotating a 2D flat plane through 360° about the z-axis.

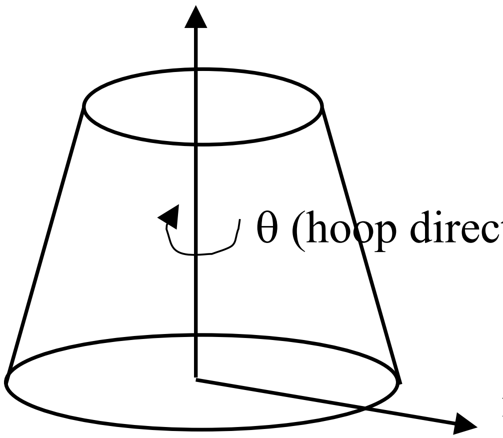


Figure 3: Axisymmetric Geometry

- For an axisymmetric assumption to be valid, both the geometry and all the loads must be axisymmetric (not just the geometry). Therefore, all loads must be ring loads.

- The variables in axisymmetric problems are:

$$[u]=\begin{bmatrix} u_r \\ u_z \end{bmatrix} \; ; \; [\varepsilon]=\begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{zz} \\ \varepsilon_{rz} \end{bmatrix} \; ; \; [\sigma]=\begin{bmatrix} \sigma_{rr} \\ \sigma_{zz} \\ \sigma_{rz} \end{bmatrix}$$

- The shear stresses $\sigma_{r\theta}$ and $\sigma_{z\theta}$ are zero.

- The strain-displacement definitions are the same as those used in 2D problems, except that x and y components are replaced by r and z, as follows:

$$[\varepsilon]=\begin{bmatrix} \varepsilon_{rr} \\ \varepsilon_{zz} \\ \varepsilon_{rz} \end{bmatrix}=\begin{bmatrix} \dfrac{\partial u_r}{\partial r} \\ \dfrac{\partial u_z}{\partial z} \\ \dfrac{\partial u_r}{\partial z}+\dfrac{\partial u_z}{\partial r} \end{bmatrix}$$

- The hoop strain is defined as follows:

$$\varepsilon_{\theta\theta}=\frac{u_r}{r}$$

- The FE formulation for axisymmetric problems follows the same steps as those for 2D problems. Axisymmetric elements, however, should be regarded as **"ring elements"**.

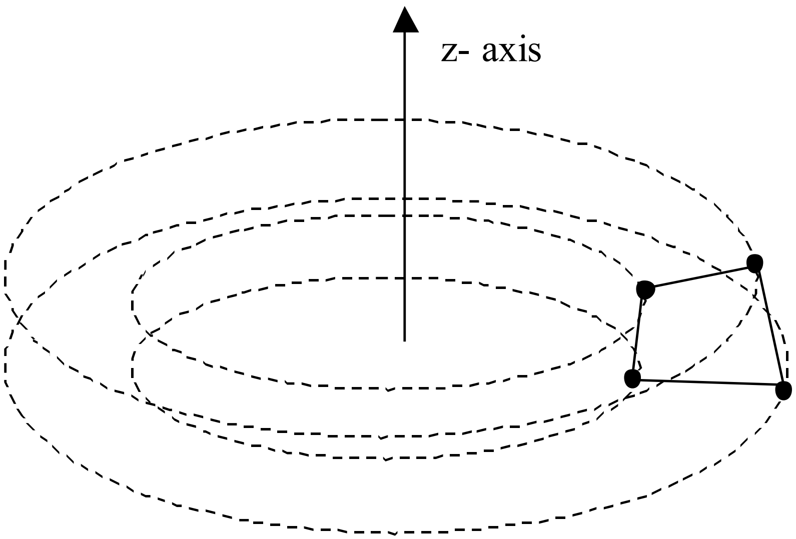


Figure 4: Axisymmetric (ring) continuum element

Non-axisymmetric loads

•If an axisymmetric geometry has non-axisymmetric (arbitrary) boundary conditions, then it must be treated as a 3D continuum problem.

•Example: A chimney or a cooling tower is subjected to wind load at its side.

•Some FE code use "special" axisymmetric elements that allow arbitrary boundary conditions, in which the non-axisymmetric loads and displacements are prescribed as a Fourier series. These "*Fourier elements*", however, are not popular.

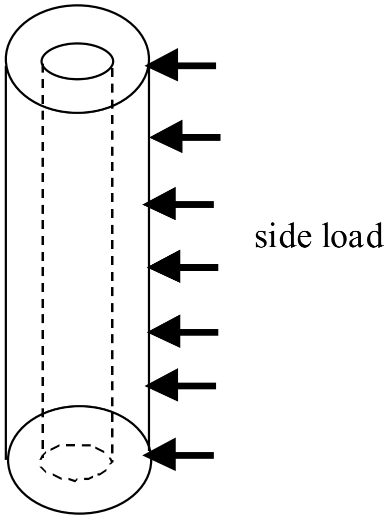
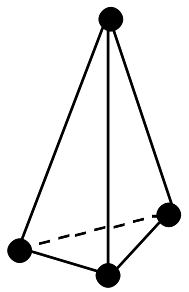


Figure 5: An axisymmetric geometry with non-axisymmetric boundary conditions.

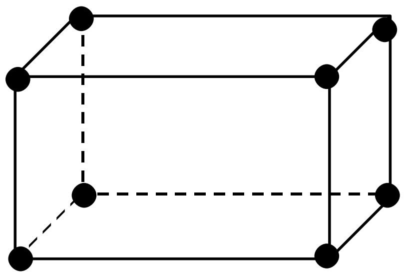
3.4 3D Continuum Elements

- The computational cost associated with 3D problems is considerably higher than that used in 2D problems.

- For example, for a 2D problem with 100 nodes and 2 degrees of freedom per node, 200 equations will be generated. For a similar spread of nodes in a 3D problem (i.e. 10 x 10 x 10 nodes), 3000 equations will be generated.



(a) 4-node tetrahedron element



(b) 8-node hexahedron element

Figure 6: 3D (linear) continuum elements

•The simplest 3D element is a 4-node constant-strain tetrahedron with a linear variation of displacement as follows:

$$
\begin{aligned}
u_x(x,y,z) &= C_1 + C_2\,x + C_3\,y + C_4\,z \\
u_y(x,y,z) &= C_5 + C_6\,x + C_7\,y + C_8\,z \\
u_z(x,y,z) &= C_9 + C_{10}\,x + C_{11}\,y + C_{12}\,z
\end{aligned}
$$

•Derivation of the 3D FE formulation follows the same steps as those for other elements, with the same matrix notation, except that the 3D matrices are larger than the corresponding 2D matrices.

3.5 Summary of Key Points

- In continuum elements, the displacement components are the only independent variables (degrees of freedom).

- In 2D continuum elements, each node has two degrees of freedom; the displacement vectors u_x and u_y in the x and y directions, respectively, whereas in 3D continuum elements, a third degree of freedom u_z is used.

- Axisymmetric continuum elements are similar to 2D elements, but use the radial and axial displacements u_r and u_z respectively.

- The simplest type of 2D continuum element is a triangular 3-node element with a linear variation of displacements (linear shape function). Since displacement is linear, the stress and strain per element is constant.

- The derivation of the FE formulation for 2D, axisymmetric and 3D continuum elements follows the same steps as the pin-jointed elements.

- In practical engineering problems, the overall assembly stiffness matrix (the solution matrix) is usually very large in size, but is sparsely populated.