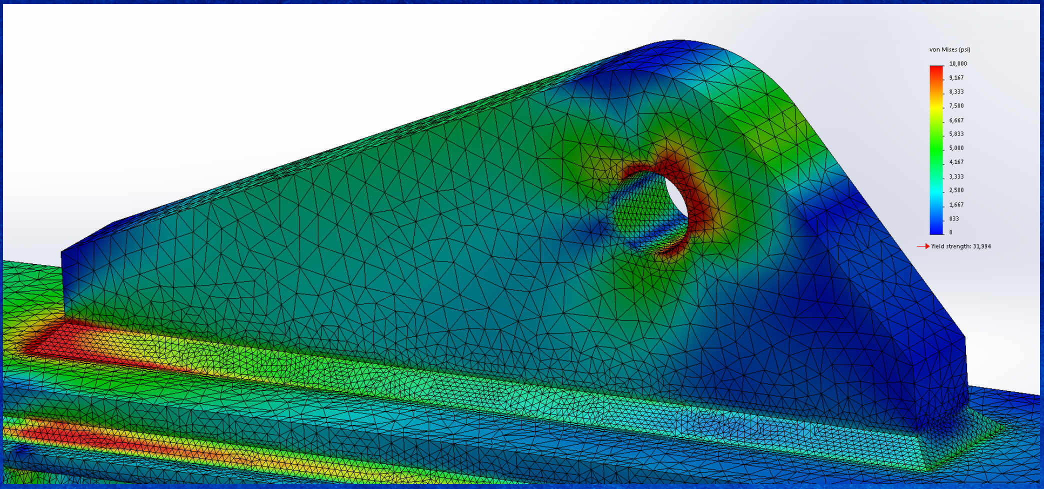
# A Simple Explanation of the Finite Element Method for Stress Analysis

Dr. Lonny L. Thompson 🧑🏻‍🏫

March 23 2025




Finite Element Stress Analysis of a Spreader Bar showing Interpolated Von-Mises Stress Contours.

Image: Innovex Engineering

https://www.innovexeng.com/engineering-services/fea-stress-analysis-services

*Swipe for the next page*

# Contents

# 1 A Simple Explanation of the Finite Element Method for Stress Analysis

- The Principle of Virtual Work (PVW) states that, for a system in equilibrium, the virtual work done by **external forces** must equal the **internal virtual work**.
- We derive the form of the stiffness matrix and nodal force equilibrium equations for stress analysis by defining both external and internal virtual work expressions.
- This is not a theory that needs to be disproven; this is the basis for the finite element method (FEM) and its application in finite element analysis (FEA) in stress analysis.

## 1.1 External Virtual Work, $W_{\text{ext}}$

- The external virtual work is the work done by external nodal forces $\{F_{\text{ext}}\}$ acting on the nodes of the element, resulting from virtual nodal displacements $\{\delta u_{\text{nod}}\}$:

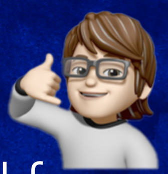
$$W_{\text{ext}} = \{\delta u_{\text{nod}}\}^T \{F_{\text{ext}}\}$$

$$W_{\text{ext}} = \begin{bmatrix} \qquad \delta u_{\text{nod}}^T \qquad \end{bmatrix} \begin{Bmatrix} F_{\text{ext}} \\ \\ \\ \\ \end{Bmatrix}$$

- Recall work is the product of displacements with forces.

- The superscript $T$ represents the transpose operation of a column vector, resulting in a row vector.

- Multiplying the row vector of virtual nodal displacements with the column vector of external nodal forces gives the dot-product (inner-product) of the two vectors, resulting in a scalar, as it should, since work is a scalar.



- If the units of displacement are meters and force in Newtons, then the product is J = N-m, the unit of work and energy.

## 1.2 Internal Virtual Work, $W_{\text{int}}$

- The internal virtual work within the element is due to virtual strains $\{\delta\varepsilon\}$ from virtual displacements.
- Using the stress-strain relationship $\{\sigma\} = [E]\{\varepsilon\}$, where $[E]$ is the elasticity matrix, we write:

$$W_{\text{int}} = \int_V \{\delta\varepsilon\}^T \{\sigma\}\, dV = \int_V \{\delta\varepsilon\}^T [E] \{\varepsilon\}\, dV$$

$$\begin{aligned} W_{\text{int}} &= \int_V \left[ \quad \delta\varepsilon^T \quad \right] \left\{ \sigma \right\} dV \\ &= \int_V \left[ \quad \delta\varepsilon^T \quad \right] \left[ \quad E \quad \right] \left\{ \varepsilon \right\} dV \end{aligned}$$

- Note the elasticity matrix $[E]$ has units of Young's modulus, same as stress; for example, Pa = N/m$^2$.
- After integrating over the volume, the result is units of work (internal energy) of J = N-m.

## 1.3 Structure and Form of the Element Stiffness Matrix

- Since virtual strain $\{\delta\varepsilon\}$ relates to virtual displacements $\{\delta u_{\text{nod}}\}$ through the strain-displacement matrix $[B]$ with units of one over length unit, we have:

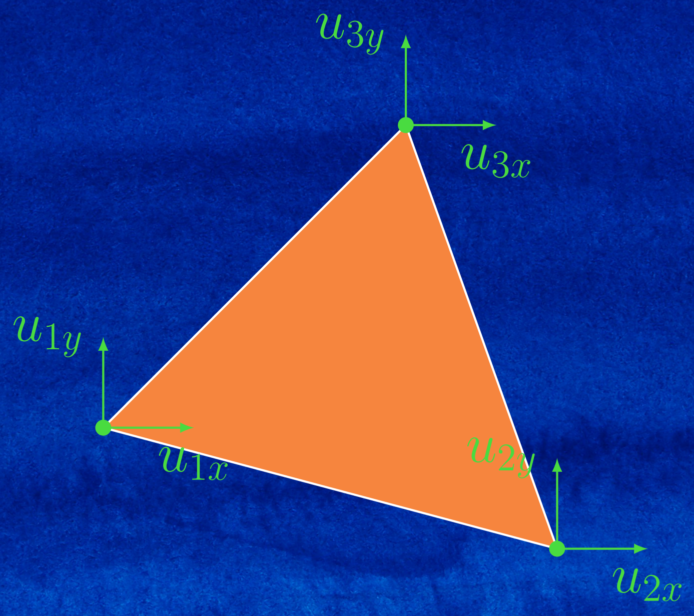
$$\{\delta\varepsilon\} = [B]\{\delta u_{\text{nod}}\}$$

$$\left\{ \varepsilon \right\} = \left[ \qquad B \qquad \right] \left\{ \begin{matrix} \delta u_{\text{nod}} \\ \\ \\ \end{matrix} \right\}$$

- The matrix $[B]$ is defined with derivatives of the dimensionless shape functions $[N]$ used to approximate the displacement field at points within an element in terms of nodal displacements.

- For a 3-node triangle element, the displacement approximation is expressed as a linear interpolation of unknown nodal displacements,

$$\boldsymbol{u}(x,y) = N_1^e(x,y)\,\boldsymbol{u}_1 + N_2^e(x,y)\,\boldsymbol{u}_2 + N_3^e(x,y)\,\boldsymbol{u}_3$$




Triangle element with two displacement DOF $(u_{ix}, u_{iy})$ at each of the three nodes.

- In component-matrix form with nodal DOF $(u_{ix}, u_{iy})$:

$$\{u\} = \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} = \begin{Bmatrix} N_1 u_{1x} + N_2 u_{2x} + N_3 u_{3x} \\ N_1 u_{1y} + N_2 u_{2y} + N_3 u_{3y} \end{Bmatrix}$$

$$= \left[ \qquad\qquad\qquad\qquad \right] \left\{ \begin{array}{c} u_{1x} \\ u_{1y} \\ \hline u_{2x} \\ u_{2y} \\ \hline u_{3x} \\ u_{3y} \end{array} \right\} = [N]\{u_{\text{nod}}\}$$

- With nodal position coordinates at each of the three nodes, $(x_i, y_i)$, the local element shape functions are linear in $x$ and $y$:

$$N_i(\boldsymbol{x}) = a_i + b_i x + c_i y, \quad i = 1, 2, 3.$$

- The coefficients $a_i, b_i, c_i$ are computed using the standard area coordinate interpolation formulas:

$$a_i = \frac{x_k y_\ell - x_\ell y_k}{2A}, \quad b_i = \frac{y_k - y_\ell}{2A}, \quad c_i = \frac{x_\ell - x_k}{2A},$$

where:

- $(x_k, y_k)$ and $(x_\ell, y_\ell)$ are the coordinates of the other two nodes in element, excluding node $i$,

- $A$ is the area of triange element:

$$A = \frac{1}{2} \left| x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2) \right| .$$

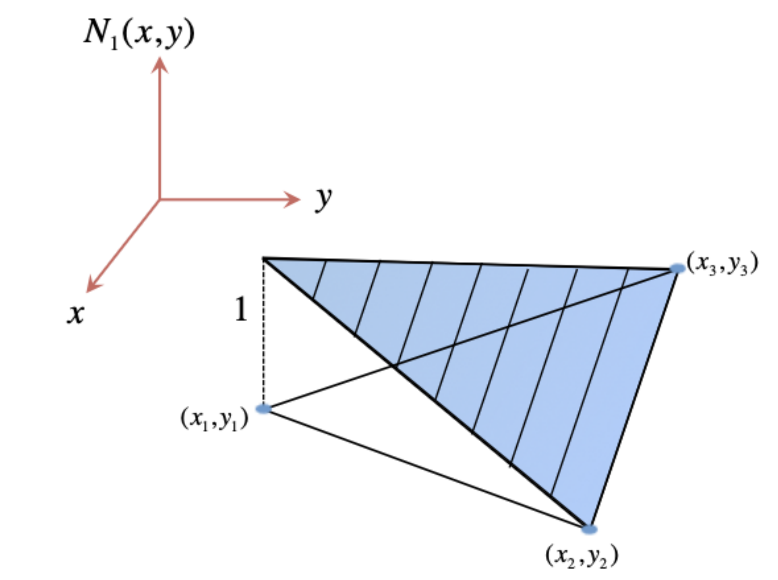
- The local element nodal shape functions $N_i(\boldsymbol{x})$ satisfy the interpolation property that they evaluate to one at their node and are zero at the other two nodes (and zero on the back edge between these two other nodes). This property is summarized by the Kronecker delta function:

$$N_i(\boldsymbol{x}_j) = \delta_{ij}.$$

where

$$\delta_{ij} = \begin{cases} 1, & \text{if } i = j, \\ 0, & \text{if } i \neq j. \end{cases}$$

This ensures that at node $i$, the shape function associated with node $j$ evaluates to 1 if $i = j$ and 0 otherwise, providing the essential partition of unity property in finite element interpolation.




Example of Linear shape function $N_1(x, y)$ with unit value at node 1, used to interpolate nodal displacements.

- Substituting this together with the stress

$$\{\sigma\} = [E]\{\varepsilon\} = [E][B]\{u_{\text{nod}}\}$$

into the internal virtual work due to the product of stress and virtual strain gives:

$$\begin{aligned} W_{\text{int}} &= \int_V \{\delta u_{\text{nod}}\}^T [B]^T [E][B]\,\{u_{\text{nod}}\}\,dV \\ &= \{\delta u_{\text{nod}}\}^T \int_V [B]^T [E][B]\,\{u_{\text{nod}}\}\,dV \\ &= \{\delta u_{\text{nod}}\}^T \underbrace{\left(\int_V [B]^T [E][B]\,dV\right)}_{[K_e]} \{u_{\text{nod}}\} \\ &= \{\delta u_{\text{nod}}\}^T [K_e]\,\{u_{\text{nod}}\} \\ &= \{\delta u_{\text{nod}}\}^T\,\{F_{\text{int}}\} \end{aligned}$$

where we have identified the element stiffness matrix

$$\boxed{[K_e] = \int_V [B]^T [E][B] \, dV}$$

and

$$\begin{aligned}
\{F_{\text{int}}\} &\triangleq \int_V [B]^T \{\sigma\} \, dV \\
&= \int_V [B]^T [E] \{\varepsilon\} \, dV \\
&= \int_V [B]^T [E][B] \, \{u_{\text{nod}}\} \, dV \\
&= \left(\int_V [B]^T [E][B] \, dV\right) \{u_{\text{nod}}\} \\
&= [K_e] \, \{u_{\text{nod}}\}
\end{aligned}$$

represents the nodal internal forces as the product of the element stiffness $[K_e]$ with nodal displacements $\{u_{\text{nod}}\}$.

## Element Internal Nodal Forces and Stiffness Matrix

$$\{F_{\text{int}}\} = [K_e]\{u_{\text{nod}}\}, \qquad [K_e] = \int_V [B]^T [E][B]\, dV$$

$$\left\{ \begin{matrix} F_{\text{int}} \\ \\ \\ \end{matrix} \right\} = \left[ \begin{matrix} \\ \qquad\qquad K_e \qquad\qquad \\ \\ \end{matrix} \right] \left\{ \begin{matrix} u_{\text{nod}} \\ \\ \\ \end{matrix} \right\}$$

- Thus, we have shown that the internal virtual work for the element can be expressed in alternative forms:

$$W_{\text{int}} = \int_V \{\delta\varepsilon\}^T \{\sigma\}\, dV = \{\delta u_{\text{nod}}\}^T \{F_{\text{int}}\}$$

or

$$W_{\text{int}} = \int_V \{\delta\varepsilon\}^T [E]\{\varepsilon\}\, dV = \{\delta u_{\text{nod}}\}^T [K_e]\, \{u_{\text{nod}}\}$$

## 1.4 Properties of Element Stiffness Matrix

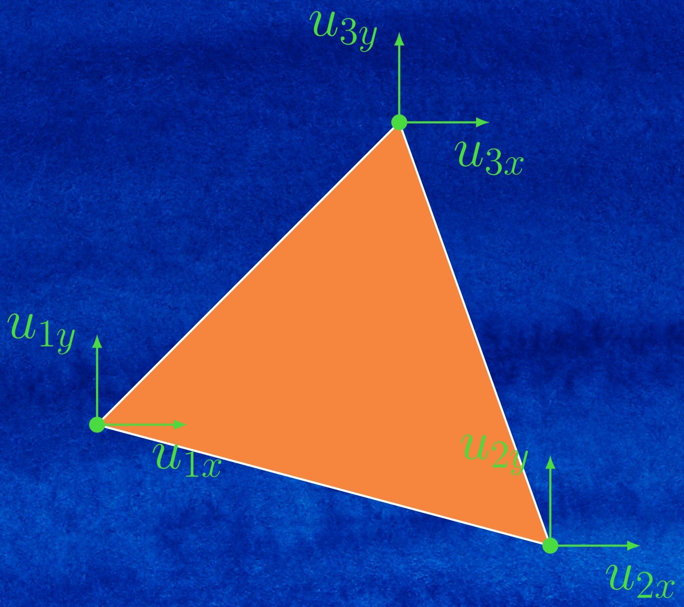
### Dimensions of the element stiffness matrix

- The dimensions of the element stiffness matrix $[K_e]$ depend on the number of nodes of the element and the number of nodal displacement components, also called degrees of freedom (DOF) per node.

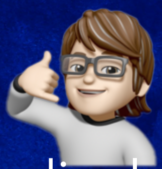
| Dimension | Element | Nodes | dof/node | degrees-of-freedom |
|---|---|---|---|---|
| 1D | Bar | 2 | 1 | $2 \times 1 = 2$ |
| 2D | Triangle $\triangle$ | 3 | 2 | $3 \times 2 = 6$ |
| 2D | Quadrilateral $\square$ | 4 | 2 | $4 \times 2 = 8$ |
| 3D | Tetrahedral | 4 | 3 | $4 \times 3 = 12$ |
| 3D | Hexahedral | 8 | 3 | $8 \times 3 = 24$ |

## 1.5 Linear Displacement Triangle Element

- For the Linear Displacement, Constant Strain Triangle (CST) Element, the number of nodes is three, and there are two perpendicular displacement components per node, resulting in $3 \times 2 = 6$ nodal displacement components for the element.




Constant Strain Triangle (CST) element showing two plane displacement components $(u_{ix}, u_{iy})$ at each of the three nodes



- For the plane stress CST element, the strain-displacement matrix $[B]$ has dimensions of three rows for the three independent strain components (two perpendicular normal strains and in-plane shear strain) and six columns corresponding to the six nodal displacement components (DOF) for the element.

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u_x}{\partial x} \\ \dfrac{\partial u_y}{\partial y} \\ \dfrac{\partial u_x}{\partial y} + \dfrac{\partial u_y}{\partial x} \end{Bmatrix} = \begin{bmatrix} & & \\ & B & \\ & & \end{bmatrix} \begin{Bmatrix} u_{1x} \\ u_{1y} \\ \hline u_{2x} \\ u_{2y} \\ \hline u_{3x} \\ u_{3y} \end{Bmatrix}$$

- For plane stress, the elasticity matrix $[E]$ in $\{\sigma\} = [E]\{\varepsilon\}$ is a symmetric square matrix, $[E]^T = [E]$, with three rows and three columns.

- For isotropic materials:

$$\{\sigma\} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1}{2}(1-\nu) \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix}$$

- Multiplying the transpose of the matrix $[B]^T$ with dimensions of 6 rows and three columns after the transpose operation, with the three-by-three elasticity matrix $[E]$, and the strain-displacement matrix $[B]$, the product of the three matrices $[B]^T[E][B]$ results in a square matrix with six rows and six columns.
- After integrating over the volume (area times thickness for plain stress), the stiffness matrix $[K_e]$ has dimensions of $6 \times 6$ corresponding to the 6 DOF for the triangle stress element.

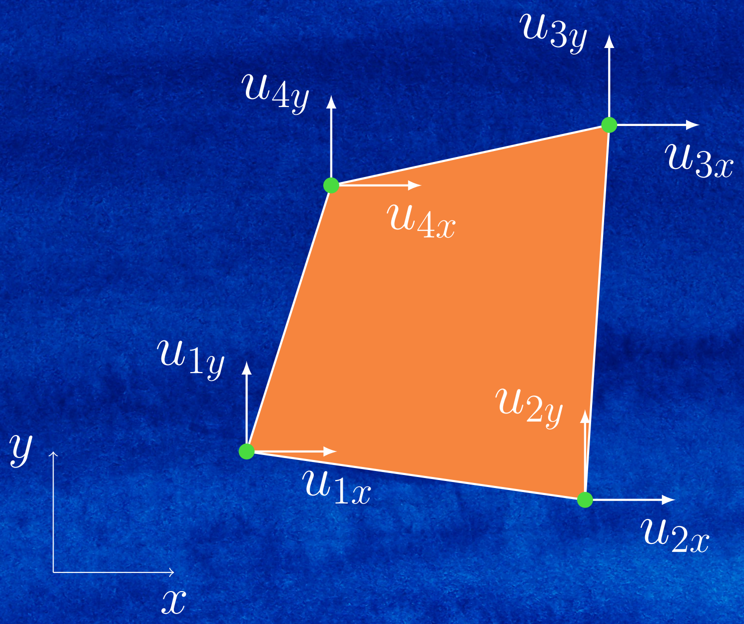
$$\underset{6\times6}{[K_e]} = \int_V \underset{(6\times3)}{[B]^T} \underset{(3\times3)}{[E]} \underset{(3\times6)}{[B]} \, dV$$

$$= \int_V \left[ \; B^T \; \right] \left[ \; E \; \right] \left[ \; B \; \right] dV$$

$$= \left[ \; K_e \; \right]$$

## 1.6 4-Node Quadrilateral (QUAD4) Element

- For the planar 4-node quadrilateral element, the number of nodes is four. Since there are two perpendicular displacement components, also called Degrees of Freedom (DOF) per node, this element has $4 \times 2 = 8$ nodal DOF.




Quadrilateral 4-node element (QUAD4) showing two plane displacement components at each of the nodes.

- For plane stress Finite Element Analysis (FEA) using this element, the matrix $[B]$ relating the strain approximation to nodal DOF has dimensions of three

rows and eight columns:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u_x}{\partial x} \\ \dfrac{\partial u_y}{\partial y} \\ \dfrac{\partial u_x}{\partial y} + \dfrac{\partial u_y}{\partial x} \end{Bmatrix} = \begin{bmatrix} & & & & B & & & \end{bmatrix} \left\{ \begin{array}{c} u_{1x} \\ u_{1y} \\ \hline u_{2x} \\ u_{2y} \\ \hline u_{3x} \\ u_{3y} \\ \hline u_{4x} \\ u_{4y} \end{array} \right\}$$

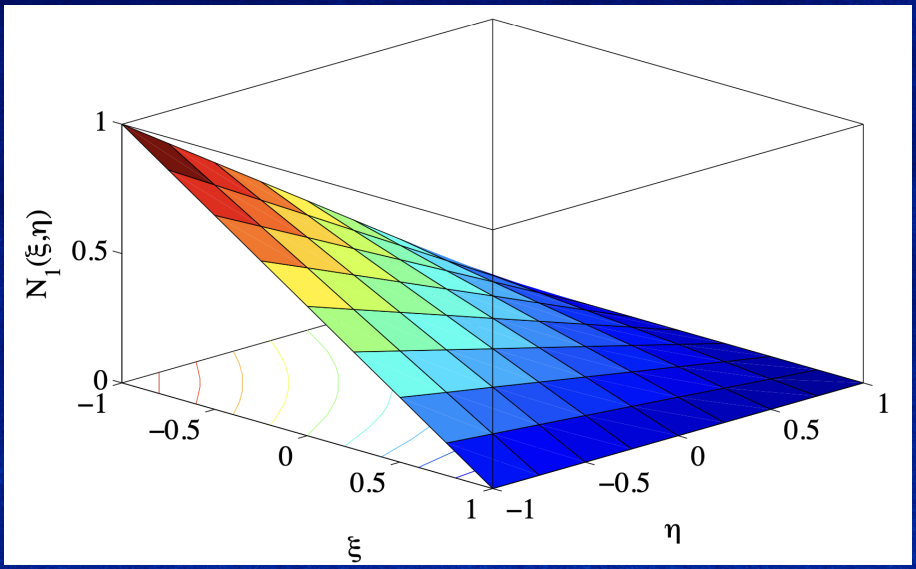
- For a 4-node quadrilateral element, the bilinear shape functions are defined in terms of local dimensionless coordinates $\xi$ and $\eta$ to facilitate numerical integration of the stiffness matrix:

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta), \qquad N_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

$$N_4 = \frac{1}{4}(1-\xi)(1+\eta), \qquad N_3 = \frac{1}{4}(1+\xi)(1+\eta)$$

- The stiffness matrix $[K_e]$ for this element has dimensions of $8 \times 8$ corresponding to the 8 DOF for the quadrilateral stress element.

$$\boxed{\underset{8\times 8}{[K_e]} = \int_V \underset{(8\times 3)}{[B]^T} \underset{(3\times 3)}{[E]} \underset{(3\times 8)}{[B]} \, dV}$$



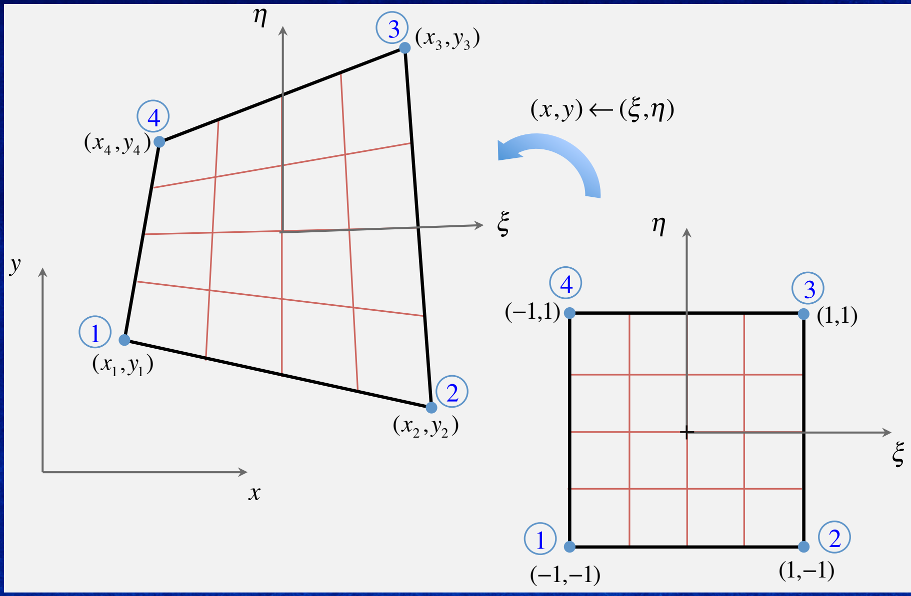

Bilinear shape (interpolation) function $N_1(\xi,\eta)$ with unit value at node 1, for 4-node Quadrilateral (QUAD4) element

- The double integration of the complicated quadrilateral shape is performed using mapping and changing variables.

$$\boxed{\boldsymbol{K_e} = \int_{-1}^{1}\int_{-1}^{1} \boldsymbol{B}^T \boldsymbol{E} \boldsymbol{B} \, \text{det}(\boldsymbol{J}) \, d\xi d\eta.}$$

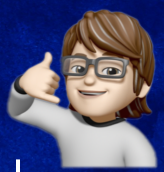
  where det$(\boldsymbol{J})$ is the determinant of a Jacobian matrix transformation of derivatives.

- The determinant of the Jacobian mapping matrix has units of square of length $[L]^2$ because it scales the physical differential area $dA = dxdy$ from a dimensionless $d\xi d\eta$, with $dA = \text{det}(\boldsymbol{J}) d\xi d\eta$.




Dimensionless points $(\xi, \eta)$ in the reference bilinear square, are mapped to physical points $(x, y)$ in the quadrilateral ⏢.

- For the special quadrilateral geometry of a rectangle, the determinant is a constant equal to the ratio of the rectangle area $A$ divided by the area of a reference biunit square equal to $2 \times 2 = 4$.
- For quadrilateral-shaped finite elements, numerical integration with Gaussian (Gauss-Legendre) quadrature is used to evaluate the stiffness matrix.



- For stress analysis, 4-node quadrilateral elements are less stiff than 3-node triangle elements and are preferred.
- The sweet spot between accuracy and computational efficiency for finite element stress analysis is often the use of quadrilateral and triangle elements with quadratic (not linear) displacement approximation.

## Symmetry of Element Stiffness Matrix

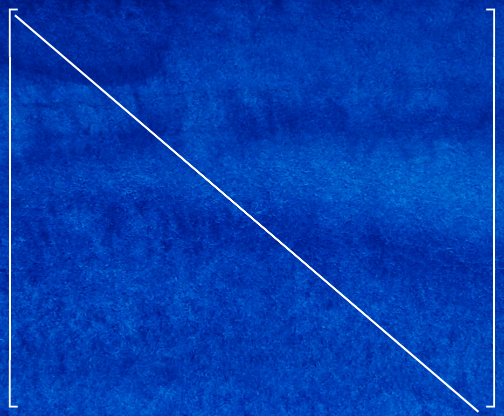
- The stiffness matrix is symmetric because the elasticity matrix $[E]$ is symmetric.
- Since $[E]^T = [E]$, then

$$\begin{aligned}
[K_e]^T &= \int_V ([B]^T[E][B])^T \, dV \\
&= \int_V [B]^T[E]^T[B] \, dV \\
&= \int_V [B]^T[E][B] \, dV = [K_e]
\end{aligned}$$

Since the stiffness matrix is symmetric, interchanging the rows and columns does not change the matrix.

$$K_{ij} = K_{ji}$$

$$[K_e]^T = [K_e] =$$



## Units of Stiffness Matrix

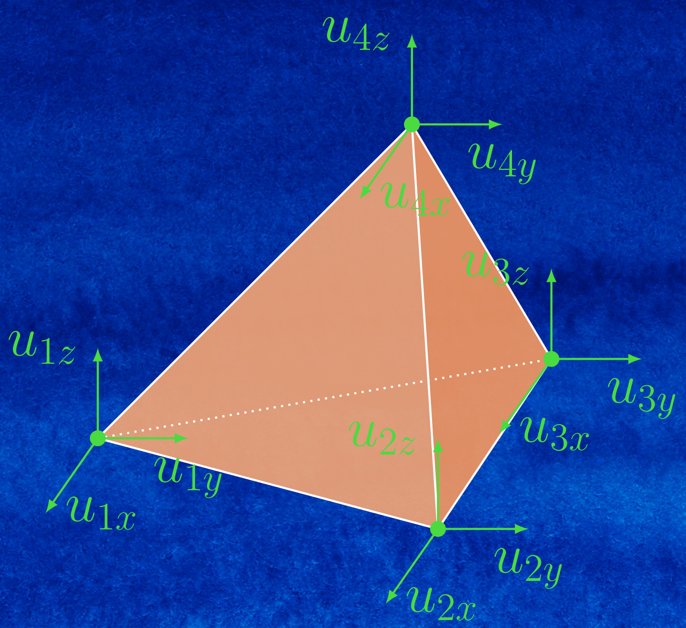
- The units of the stiffness matrix are the units of the matrix products $[B]^T[E][B]$ integrated over the volume.
    - The units of $[B]$ and its transpose $[B]^T$ are one over length, for example $\mathrm{m}^{-1}$.
    - The units of $[E]$ are $\mathrm{N/m}^2$, multiplying, the units of the products $[B]^T[E][B]$ we have $\mathrm{N/m}^4$.
    - After integrating over the volume with units $\mathrm{m}^3$, the values in the stiffness matrix $[K_e]$ have units:

      $$(\mathrm{N/m}^4)(\mathrm{m}^3) = \mathrm{N/m},$$

      - the units of a linear spring as expected.

## 1.7 Linear Displacement Tetrahedral Element

- For the Linear Displacement Tetrahedral Stress Element (pyramid-shaped volume with four triangular faces) for 3D applications, there are four nodes at the vertices, with three perpendicular displacement components per node, resulting in $4 \times 3 = 12$ nodal displacement components (DOF) for the element.




Tetrahedon 3D solid element showing three perpendicular displacement components at each of the four nodes

- The strain-displacement matrix $[B]$ has dimensions of six rows for the six independent strain components (three perpendicular normal strains and three shear strains) and twelve columns corresponding to the twelve nodal displacement components (DOF) for the element.

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \\ 2\varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u_x}{\partial x} \\ \frac{\partial u_y}{\partial y} \\ \frac{\partial u_z}{\partial z} \\ \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial y} \\ \frac{\partial u_x}{\partial z} + \frac{\partial u_z}{\partial x} \\ \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \end{Bmatrix} = \underset{6\times 12}{[B]} \left\{ \begin{array}{c} u_{1x} \\ u_{1y} \\ u_{1z} \\ \hline u_{2x} \\ u_{2y} \\ u_{2z} \\ \hline u_{3x} \\ u_{3y} \\ u_{3z} \\ \hline u_{4x} \\ u_{4y} \\ u_{4z} \end{array} \right\}$$

- The elasticity matrix $[E]$ in $\{\sigma\} = [E]\{\varepsilon\}$ is a square symmetric matrix with six rows and six columns.
- For isotropic materials:

$$\{\sigma\} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{yz} \\ \sigma_{xz} \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} \lambda + 2G & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda + 2G & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda + 2G & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{yz} \\ \gamma_{xz} \\ \gamma_{xy} \end{Bmatrix}$$

where

$$\lambda = \frac{E\nu}{(1-2\nu)(1+\nu)}, \qquad G = \frac{E}{2(1+\nu)}$$

- Multiplying the transpose of the matrix $[B]^T$ with dimensions of 12 rows and 6 columns after the transpose operation, with the six-by-six elasticity matrix $[E]$, and the strain-displacement matrix $[B]$, the product of the three matrices $[B]^T[E][B]$ results in a square matrix with 12 rows and 12 columns.

$$\boxed{\underset{12\times 12}{[K_e]} = \int_V \underset{(12\times 6)}{[B]^T} \; \underset{(6\times 6)}{[E]} \; \underset{(6\times 12)}{[B]} \; dV}$$

- After integrating over the volume, the stiffness matrix $[K_e]$ has dimensions of $12 \times 12$ corresponding to the 12 DOF for the tetrahedron stress element.

## 1.8 Application of the Principle of Virtual Work in the Finite Element Method

- According to the PVW, we equate internal and external virtual works:

$$W_{\text{ext}} = W_{\text{int}}$$

- Equating, we obtain:

$$\{\delta u_{\text{nod}}\}^T \{F_{\text{ext}}\} = \{\delta u_{\text{nod}}\}^T \{F_{\text{int}}\}$$

that is,

$$\{\delta u_{\text{nod}}\}^T \{F_{\text{ext}}\} = \{\delta u_{\text{nod}}\}^T [K_e] \{u_{\text{nod}}\}$$

- Since this holds for any arbitrary virtual displacement $\{\delta u_{\text{nod}}\}$, subject to any displacement constraints, we conclude:

$$\{F_{\text{ext}}\} = \{F_{\text{int}}\}$$

that is,

$$\{F_{\text{ext}}\} = [K_e] \{u_{\text{nod}}\}$$

- These equations represent force equilibrium at each node, balancing external with internal forces.

## 1.9 Solving the Nodal Force Balance Equations

- After applying nodal displacement constraints to reduce the number of equations, we solve for the remaining unknown nodal displacements $\{\bar{u}_{\text{nod}}\}$, in terms of applied external nodal forces combined with forces due to stiffness multiplied by any specified nodal displacements, denoted $\{\bar{F}_{\text{ext}}\}$.
- After doing this, the reduced system of equations can be expressed as,

$$[\bar{K}_e]\,\{\bar{u}_{\text{nod}}\} = \{\bar{F}_{\text{ext}}\}$$

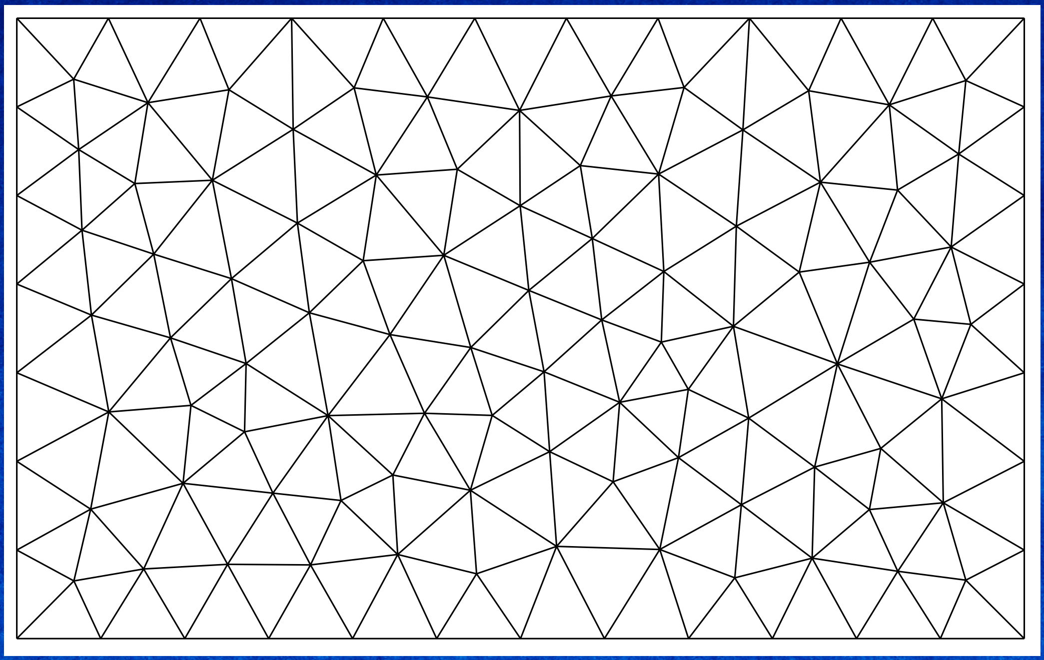
- In abstract form, we can solve these equations for nodal displacements using the matrix inverse,

$$\{\bar{u}_{\text{nod}}\} = [\bar{K}_e]^{-1}\,\{\bar{F}_{\text{ext}}\}$$

## Practical Considerations

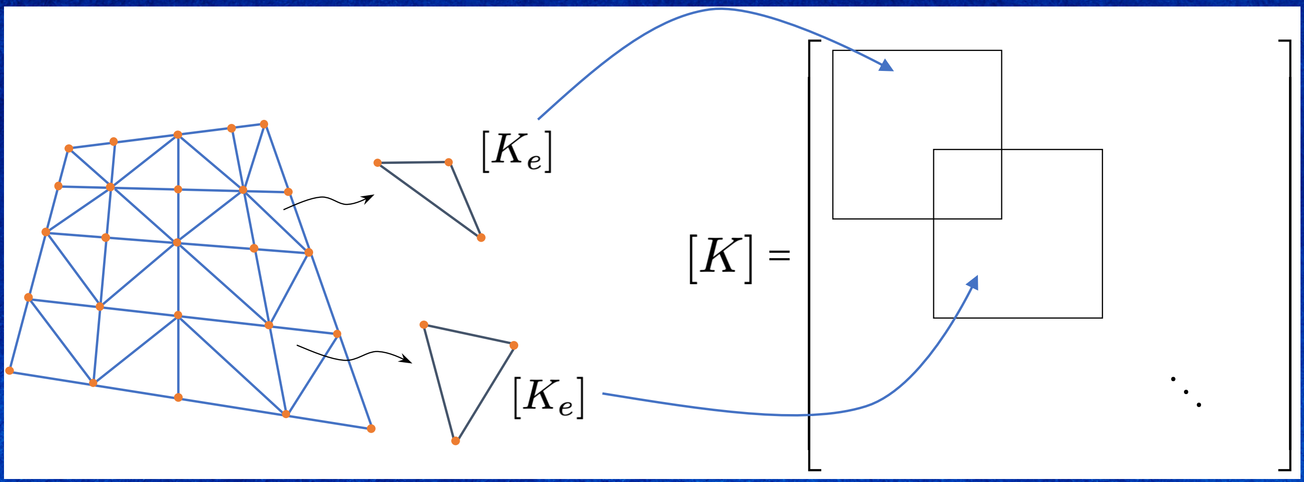
### Mesh of Connected Finite Elements

- In practice, to obtain more accurate solutions, more than one element is used in a finite element mesh of connected elements.
- In general, the promise is that the accuracy of the solution will improve by subdividing the part volume into many smaller elements.

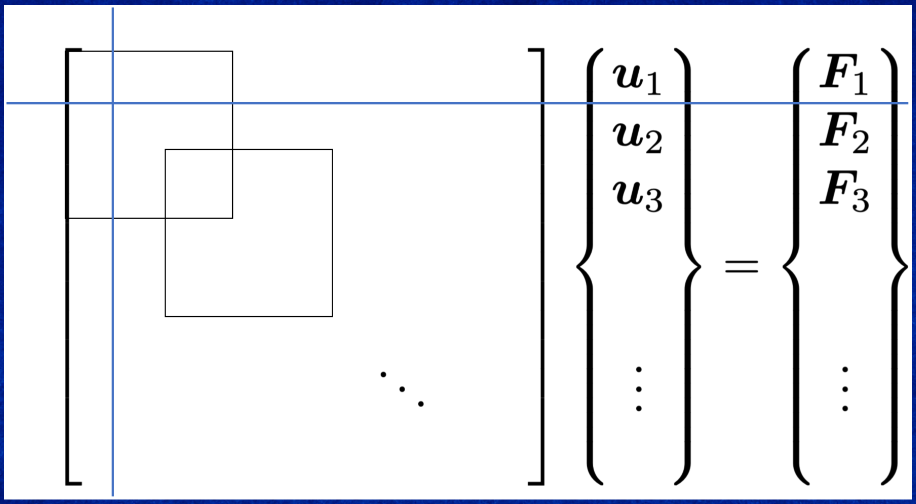


## 1.10 Assembly of Global Stiffness Matrix from Element Contributions

- In practice, the global stiffness matrix $[K]$ for all nodes in the finite element mesh is determined by assembling local element matrices $[K_e]$ in appropriate rows and columns, accounting for nodal displacement compatibility between connected elements, using information from node and displacement component numbers connected with each element.



## 1.11 Application of Displacement Constraints with Matrix Partitioning




- After applying essential nodal displacement constraints, for example, $\boldsymbol{u}_1 = \mathbf{0}$ for fixed or nonzero values like $1$ mm for prescribed, we solve the remaining unknown nodal displacements $\{\bar{u}_{\text{nod}}\}$, in terms of applied external nodal forces combined with forces due to stiffness multiplied by the specified nodal displacements.

$$\left[\begin{array}{c|c} & \\ \hline & K \\ & \end{array}\right]\left\{\begin{array}{c} \boldsymbol{u}_1 \\ \hline \boldsymbol{u}_2 \\ \boldsymbol{u}_3 \\ \vdots \end{array}\right\} = \left\{\begin{array}{c} \boldsymbol{R}_1 \\ \hline \boldsymbol{F}_2 \\ \boldsymbol{F}_3 \\ \vdots \end{array}\right\}$$

$$[\bar{K}]\,\{\bar{u}_{\text{nod}}\} = \{\bar{F}_{\text{ext}}\}$$

$$\begin{bmatrix} & & \\ & \bar{K} & \\ & & \end{bmatrix} \begin{Bmatrix} \boldsymbol{u}_2 \\ \boldsymbol{u}_3 \\ \\ \vdots \\ \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{F}_2 \\ \boldsymbol{F}_3 \\ \\ \vdots \\ \end{Bmatrix} - \begin{Bmatrix} \\ \\ \\ \\ \end{Bmatrix} \boldsymbol{u}_1$$

- Since $\boldsymbol{u}_1$ is prescribed, the corresponding force $\boldsymbol{R}_1$ is a reaction.

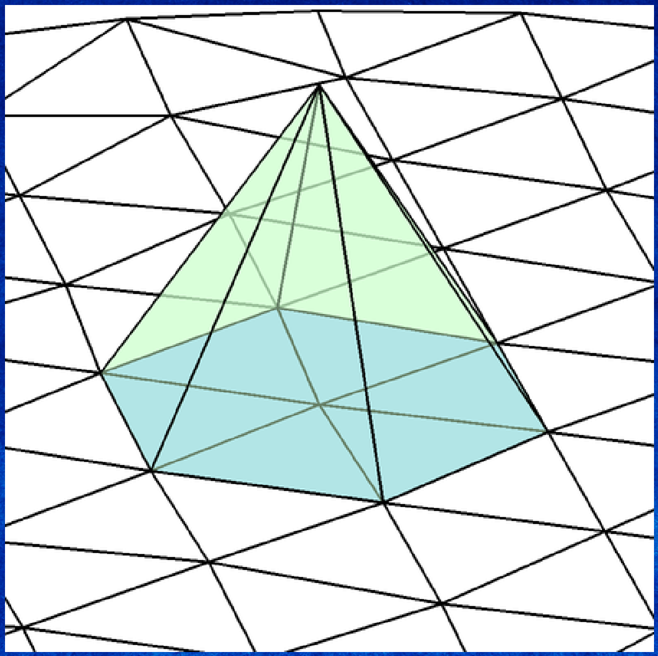
- For solutions to exist and be unique, the degrees-of-freedom in the nodal constraints $\boldsymbol{u}_1$ must be specified such that there are no rigid-body (zero non-deformation) motions, both rigid-body translation and rotation.

## 1.12 Global Nodal Basis for Mesh Nodes

- When describing the finite element method (FEM), we often emphasize the role of elements and their connections in a mesh; however, the big-picture is driven by the nodes and degrees of freedom connecting these elements.

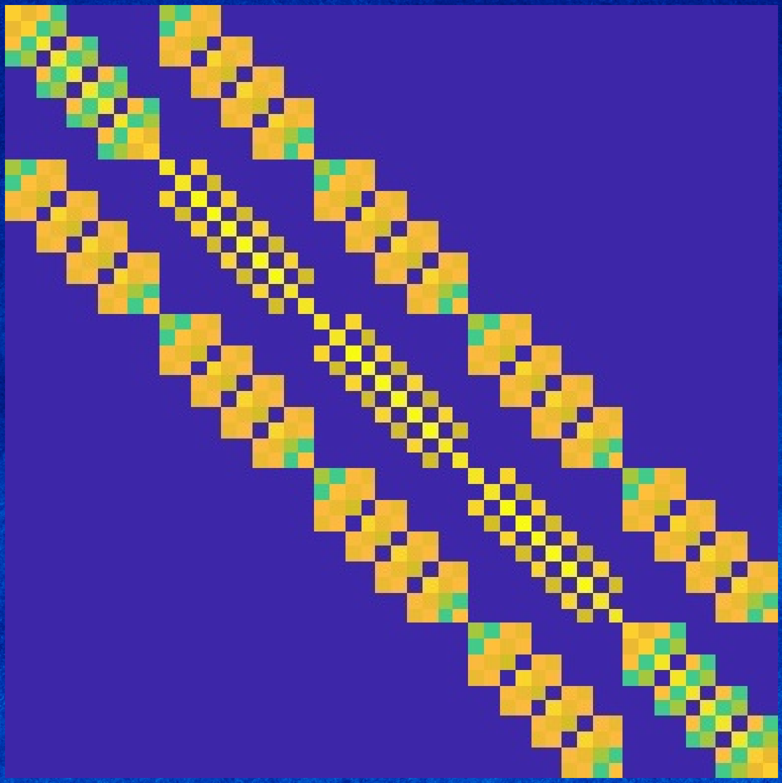
- Each row in the global stiffness matrix represents the equilibrium equation and stiffness relation associated with that degree of freedom (DOF), coupling all other nodes that are influenced by this node through the support of the basis (shape) functions associated with this DOF.

- The global basis (shape) function $\varphi_i$ of a node in the mesh is constructed from the local shape functions $N_i$ associated with the elements that connect at this node.
- The global basis (shape) functions used in the finite element method (FEM) have compact support, meaning they influence only a local patch of elements and their connected nodes and DOF.



## 1.13 Sparsity of the Assembled Finite Element Stiffness Matrix

- The assembled finite element global stiffness matrix (or mass matrix) is often filled with many zeros—hence, it is called a sparse matrix.
- The global stiffness matrix $[\boldsymbol{K}]$ is sparse, meaning most of its entries are zero.



Sparsity Pattern of Nonzero values for the Stiffness Matrix for 2D Plane Elasticity with 2-DOF/node, $(u_x, u_y)$ with a $4 \times 4$ square mesh of 4-node Quadrilateral (QUAD4) Elements.

- This sparsity arises because each finite element only interacts with a small subset of the total degrees of freedom in the system.
- The number of nonzero entries in any row corresponds to the nodes that interact with a given node in the mesh. This interaction is dictated by the local support of the nodal basis function and the element connectivity.
- Knowing the sparsity counts of nonzero values and patterns in advance helps optimize storage and solver performance.
- In practice, this can make the difference between waiting 24 hours or 1 hour for a solver job to complete and whether you can run this job on your laptop or need a more powerful computer.
- Sparse direct solvers, iterative equation-solving methods, and preconditioners all benefit from this information.

## Equation Solving Methods

- In practice, the inverse $[\bar{K}]^{-1}$ is never taken to solve for unknown nodal DOF since it would be a very computationally expensive operation for a large number of equations in a finite element mesh.
  - Instead, the equations are solved using direct algorithms such as factorizations or other generalizations of Gaussian elimination learned in linear algebra or for very large systems of equations, for example, one-million equations, to reduce the computer memory requirements, iterative solvers such as the Conjugate-Gradient method are more efficient.

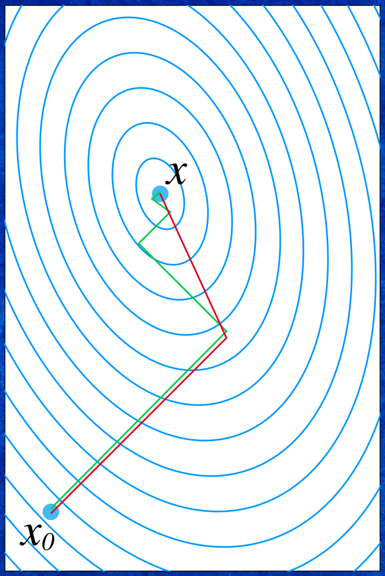


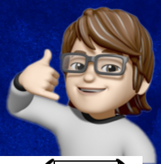

Illustration of convergence to a solution $x$ from starting estimate $x_0$ using iterative equation solving process.

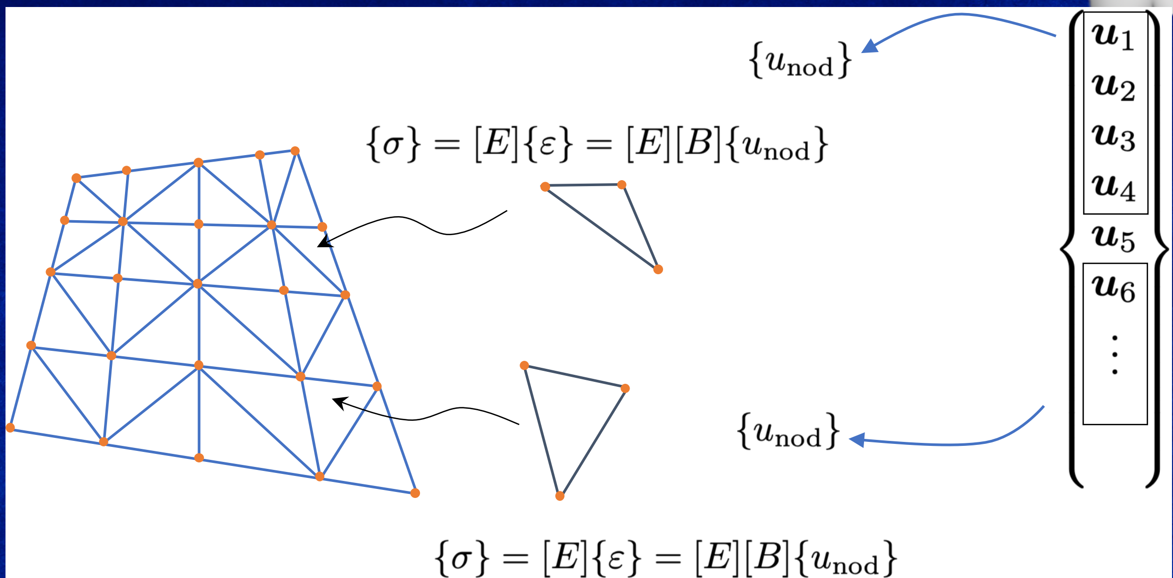
## 1.14 Calculation of Element Stresses

- After the nodal displacements are solved from the nodal force balance (stiffness equations), we can distribute nodal displacements to each element and calculate stress within each element using the elasticity matrix $[E]$ and the linear elastic relationship between stress and strain:

$$\{\sigma\} = [E]\{\varepsilon\} = [E][B]\{u_{\text{nod}}\}$$

$$\left\{ \sigma \right\} = \left[ \quad E \quad \right] \left[ \qquad B \qquad \right] \left\{ \delta u_{\text{nod}} \right\}$$

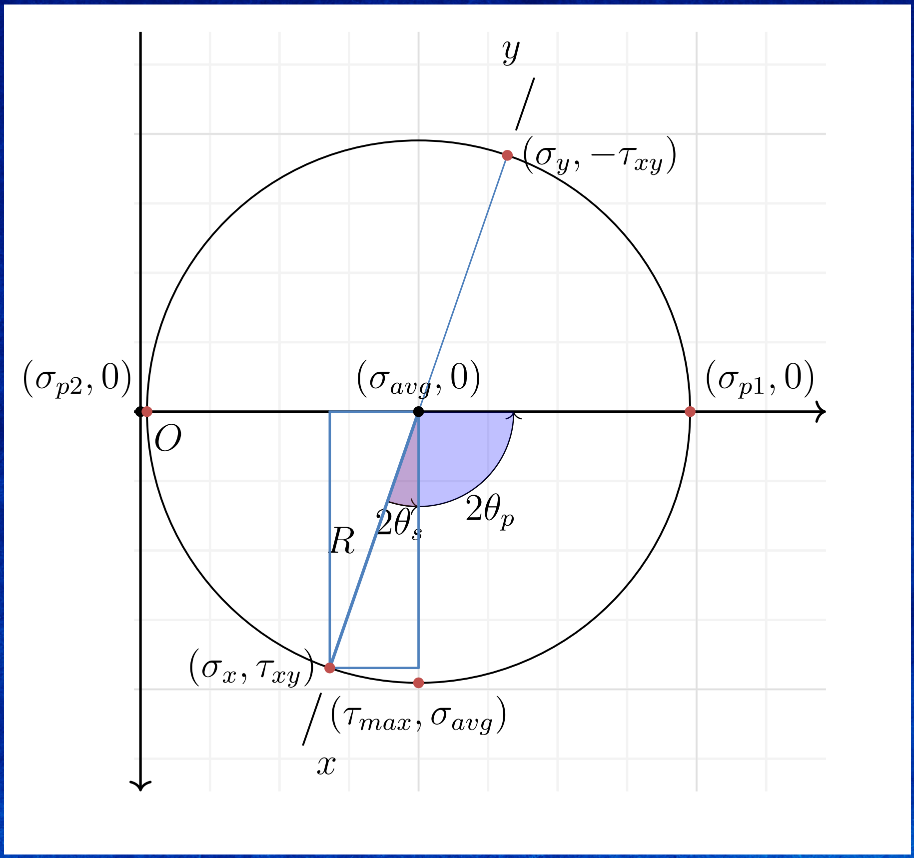





### 1.14.1 Stress Recovery

- The displacement approximation is piecewise linear and **continuous** across shared edges of adjacent elements in the finite element mesh.
- However, since the strains are defined in terms of derivatives of displacements, these are discontinuous between element boundaries.
- Since the stress is computed from strains through the material constitutive equations, the stresses are also discontinuous between elements and at nodes.

- Once the stress components are determined within each element, additional calculations to determine Principal Normal Stresses, Von-Mises stress, and other criteria are used for yield and failure analysis.




Mohr's circle graphically illustrates the state of plane stress, principle normal stresses, and maximum in-plane shear stress.

- The stresses within each element are then interpolated/extrapolated or smoothed to recover more accurate nodal stresses.
- A widely used technique is patch recovery, taking the stress values at Gauss integration points with each element, interpolating them across neighboring elements in a patch, and then smooths or averaging these values at the nodes.

# 1.15 Summary of the PVW for Deriving the Element Stiffness Matrix and Solution for Nodal Displacements

- In deriving the stiffness matrix for finite elements, we apply the **Principle of Virtual Work (PVW)**. The PVW states that at equilibrium, the virtual work done by **external forces** must balance the virtual work of **internal forces**.

- We proceed by defining the internal nodal forces from the matrix product of the element stiffness matrix with nodal displacements and show the internal virtual work for the stress element is

$$W_{\text{int}} = \int_V \{\delta\varepsilon\}^T \{\sigma\}\, dV = \{\delta u_{\text{nod}}\}^T \{F_{\text{int}}\}$$

from which we deduce the element stiffness matrix $[K_e]$:

$$W_{\text{int}} = \int_V \{\delta\varepsilon\}^T [E] \{\varepsilon\}\, dV = \{\delta u_{\text{nod}}\}^T [K_e] \{u_{\text{nod}}\}$$

- We then apply the PVW to determine the nodal equilibrium equations, which balance external nodal forces with internal nodal forces defined by the product of the stiffness matrix and nodal displacements.

### 1.15.1 Internal Nodal Forces, $\{F_{\text{int}}\}$

The internal nodal forces $\{F_{\text{int}}\}$ are defined by the product of the element stiffness matrix $[K_e]$ and the nodal displacements $\{u_{\text{nod}}\}$:

$$\{F_{\text{int}}\} = [K_e]\{u_{\text{nod}}\}, \qquad [K_e] = \int_V [B]^T[E][B]\, dV$$

### 1.15.2 External Virtual Work, $W_{\text{ext}}$

The virtual work done by the external forces $\{F_{\text{ext}}\}$ acting on the nodes of the element due to virtual displacements $\{\delta u_{\text{nod}}\}$ is:

$$W_{\text{ext}} = \{\delta u_{\text{nod}}\}^T\{F_{\text{ext}}\}$$

### 1.15.3 Internal Virtual Work, $W_{\text{int}}$

The virtual work done by the internal forces is similarly given by:

$$W_{\text{int}} = \{\delta u_{\text{nod}}\}^T\{F_{\text{int}}\} = \{\delta u_{\text{nod}}\}^T[K_e]\{u_{\text{nod}}\}$$

### 1.15.4 Applying the Principle of Virtual Work

- According to the Principle of Virtual Work, the external virtual work equals the internal virtual work:

$$W_{\text{ext}} = W_{\text{int}}$$

- Expanding both sides, we have:

$$\{\delta u_{\text{nod}}\}^T \{F_{\text{ext}}\} = \{\delta u_{\text{nod}}\}^T [K_e] \{u_{\text{nod}}\}$$

- Since this equality holds for any arbitrary virtual displacement $\{\delta u_{\text{nod}}\}$, it follows that:

$$\{F_{\text{ext}}\} = [K_e] \{u_{\text{nod}}\}$$

  subject to constraints on nodal virtual displacements.

- These equations represent nodal force equilibrium, balancing external with internal forces.

## Solution of Nodal Force Balance Equations

- This relationship confirms that after assembly of element stiffness matrix contributions in a finite element mesh of the part, and nodal displacement constraints are applied,

$$[\bar{K}]\{\bar{u}_{\text{nod}}\} = \{\bar{F}_{\text{ext}}\}$$

  the reduced **stiffness matrix** $[\bar{K}]$ represents the mapping between nodal displacements and external nodal forces in equilibrium adjusted by the product of stiffness and nodal constraints, represented by $\{\bar{F}_{\text{ext}}\}$.

- By expressing the internal virtual work for the element as the product of virtual displacements with the internal nodal forces equating to $\{F_{\text{int}}\} = [K_e]\{u_{\text{nod}}\}$, applying the Principle of Virtual Work with nodal displacement constraints, accounting for element-node connectivity in an assembly of finite elements, we obtain the system of nodal equilibrium equations with the solution for nodal displacement

$$\{\bar{u}_{\text{nod}}\} = [\bar{K}]^{-1}\{\bar{F}_{\text{ext}}\}$$

  After distributing nodal displacements to each connected element in the part mesh, stress is calculated from the linear elastic equation:

$$\{\sigma\} = [E]\{\varepsilon\} = [E][B]\{u_{\text{nod}}\}$$

# We Love the
# Finite Element Method (FEM)
# and its application in Finite Element Analysis (FEA)!