Training on NBC Compliant Computer Aided Design of buildings for Engineers/Designers of Kathmandu Valley

Introduction to Finite Element Modeling

17 - 23 Septe,ber , 2014

Kathmandu

Binay Shrestha

Center of Resilience Development

Technical support from CoRD

Objective

- To understand the fundamentals of the Finite Element Method and the Finite Element Analysis
- To apply the Finite Element Analysis Tools for Modeling and Analysis of Structures
- Use SAP2000/ STAAD.Pro as Tool for Finite Element Modeling and Analysis of Structures

Need for Analysis

We need to determine the Response of the Structure to Excitation

Analysis

So that

Design

We can ensure that the structure can sustain the excitation with an acceptable level of response

Need for Modeling

- Real structure can not be analyzed: it can only be load tested.

- We can only analyze a "Model" of the structure

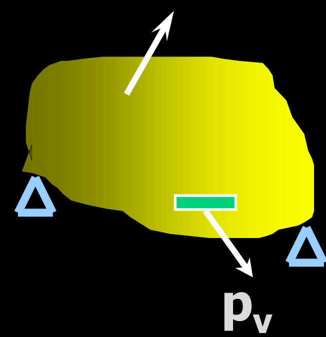
- We therefore need tools to Model the structure and to analyze the model

Need for Structural Model

STRUCTURE

EXCITATION

- Loads
- Vibrations
- Settlements
- Thermal Changes

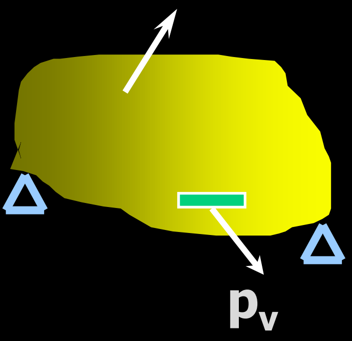


RESPONSE

- Displacements
- Strains
- Stress
- Stress Resultants

Analysis of Structure

Real structure is governed by "Partial Differential Equations" of various order.



Direct solution is only possible for:

- Simple geometry
- Simple Boundary
- Simple Loading

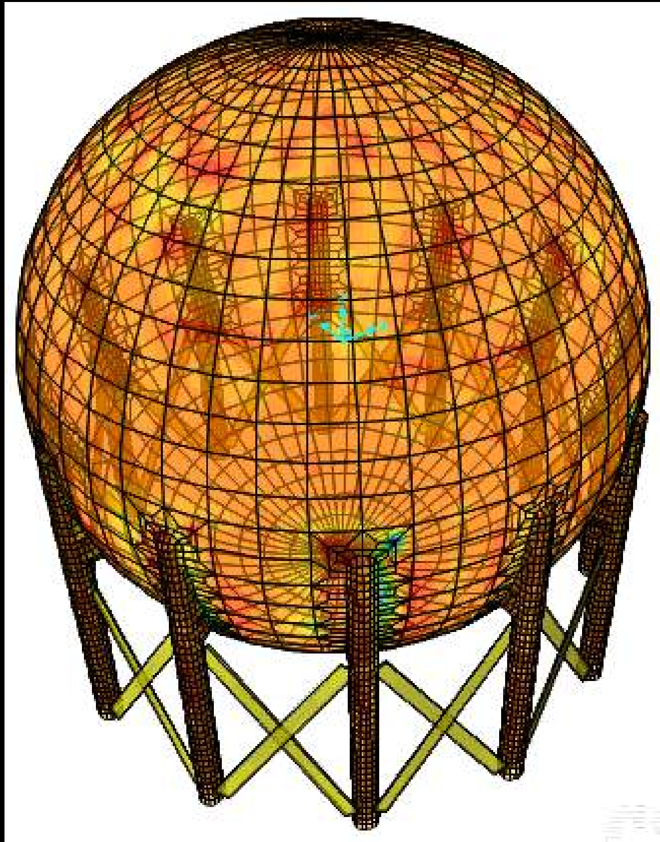
$$\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} + p_{vx} = 0$$

For problem involving complicated geometries, loadings and material properties, it is generally not possible to obtain closed-form solution

Finite Element Method (FEM) & FEA

The finite element formulation results in system of simultaneous algebraic equation.

These numerical methods yield approximate values of the unknown at discrete number of points.



FROM CLASSICAL TO FEM

Classical

Actual Structure

$$\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} + p_{vx} = 0$$

"Partial Differential Equations"

Assumptions

Equilibrium

Stress-Strain Law

Compatibility

$$\int \sigma^t \, \bar{\varepsilon} \, dV = \int_v p_v^t \, \bar{u} \, dV + \int p_s^t \, \bar{u} \, ds$$

(Principle of Virtual Work)

FEM

Structural Model

$$Kr = R$$

"Algebraic Equations"

K = Stiffness
r = Response
R = Loads

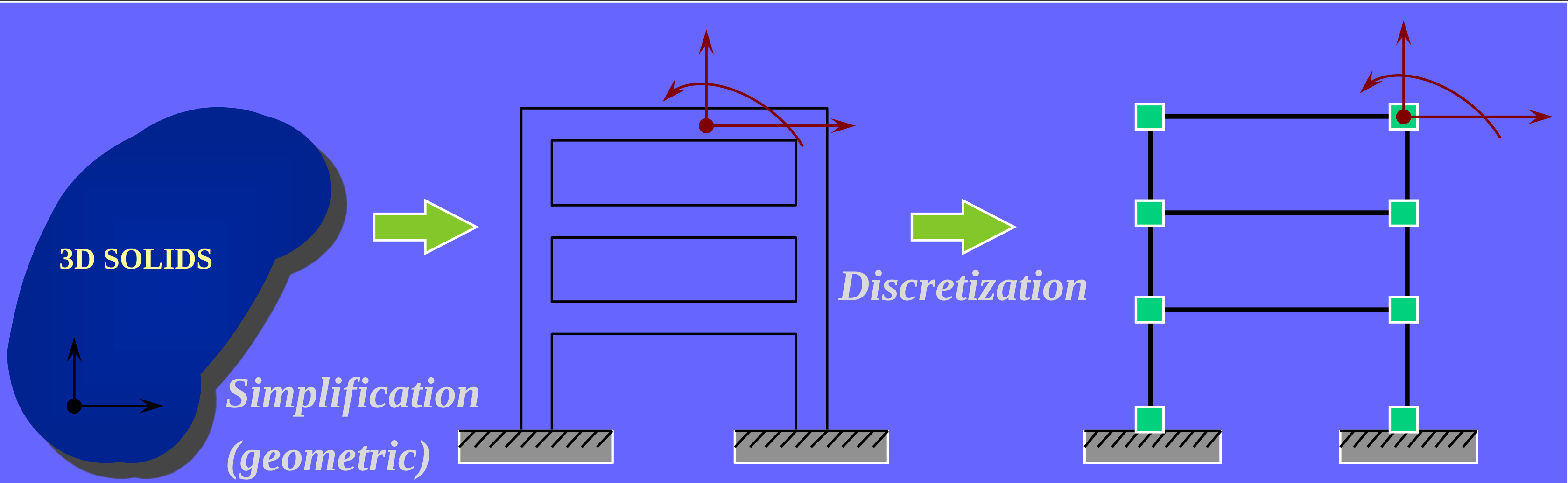
Finite Element Method (FEM) & FEA

FEM : A numerical procedure for solving (partial) differential equations associated with field problems, with an accuracy acceptable to engineers

FEA : A discretized solution to a continuum problem using FEM

day 1\day 1 session 3 & 4 Reference\demo SS beam.sdb

SOLID – STRUCTURE - MODEL



3D-CONTINUM MODEL

(Governed by partial differential equations)

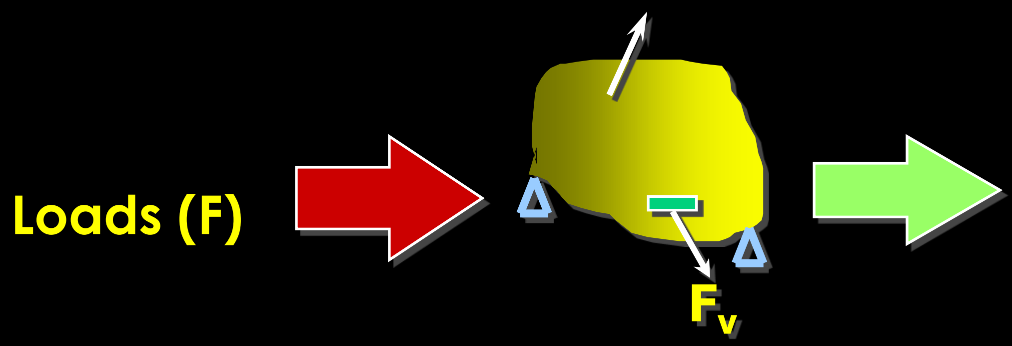
CONTINUOUS MODEL OF STRUCTURE

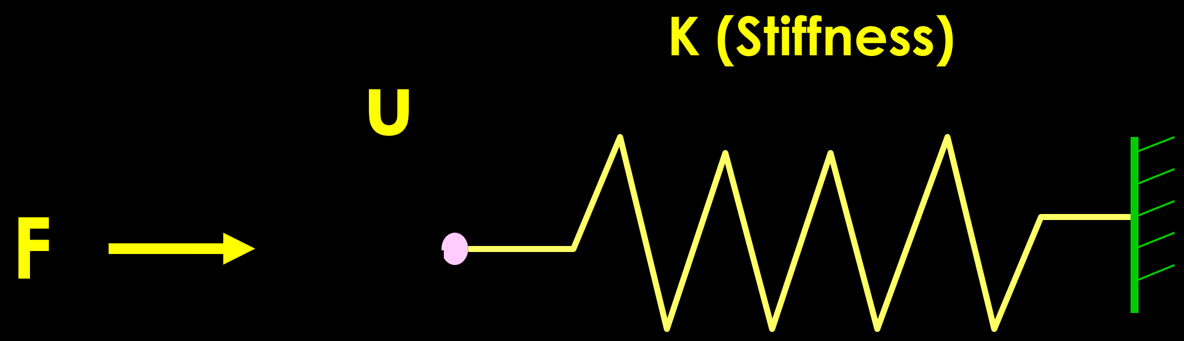
(Governed by either partial or total differential equations)

DISCRETE MODEL OF STRUCTURE

(Governed by algebraic equations)

Simplified structural system





Equilibrium Equation

$$F = K u$$

Finite Element Method (FEM)

FEA : Modeling a body by dividing it into smaller units called “finite elements” connected at points called “nodes” common to 2 or more elements.

In FEA, instead of solving the problem for entire body, equation for each finite elements are formulated and combined to obtain the overall solution

Flexibility method (force) $\{D\}=\{F\}\{P\}$

Stiffness method (displacement) $\{F\}=\{K\}\{D\}$

Element level matrix is formed for each element

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ \vdots \\ F_{nz} \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ \vdots & & & \\ K_{n1} & K_{n2} & \cdots & K_{nn} \end{bmatrix} \begin{Bmatrix} d_{1x} \\ d_{1y} \\ \vdots \\ d_{nz} \end{Bmatrix}$$

These matrices are assemble to form a global matrix and solved.

Nodes and elements

- Nodes are imaginary points used to describe arbitrary quantities and serve to provide connectivity across element boundaries

- The Finite Elements are discretized representation of the continuous structure

- Generally they correspond to the physical structural components but sometimes dummy or idealized elements my also be used

- Elements behavior is completely defined within its boundaries and is not directly related to other elements

Converting Structure to FE Model

Convert the real structure "geometry" and real excitation "load" into finite element representation

Finite element model is made up of Elements and Nodes

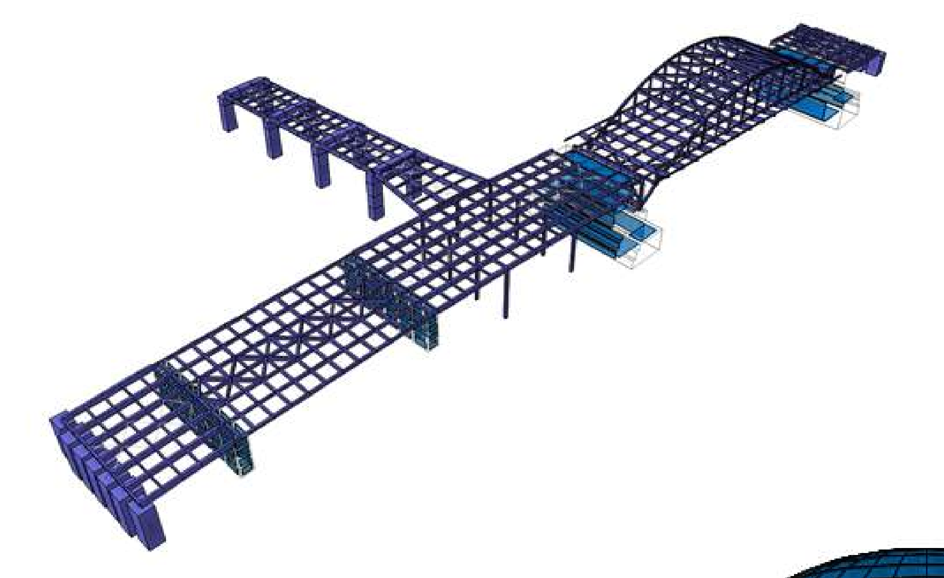
1. Nodes
   - Geometry - Coordinates
   - Boundary conditions
   - Displacements
   - Mass
2. Elements
   - Geometry - Connectivity
   - Material properties
   - Mass, etc

General Step of FEM

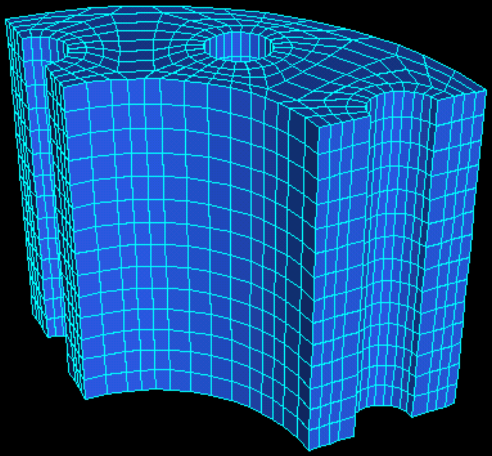
1. Discretize + select element type
2. Select a displacement function
3. Define strain-displacement relation $\varepsilon = \dfrac{du}{dx}$ and stress-strain relation $\sigma_x = E\varepsilon_x$
4. Define element stiffness matrix : a)direct stiffness, b) work or energy method, c) method of weighted residual
5. Assemble the element equation to obtain global equation and apply boundary condition
6. Solve for unknown DOFs (displacements)
7. Solve for element strain and stresses etc
8. Interpret the result

Global Modeling "Macro Model"

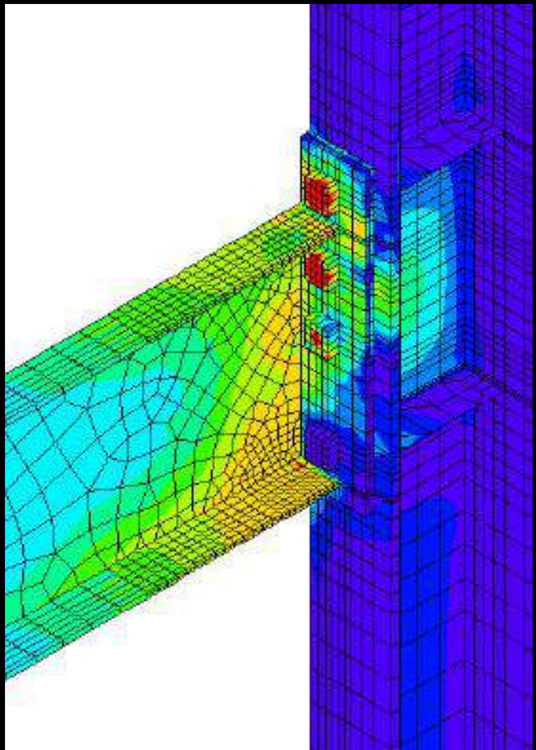
- A model of the whole structure
- Objective is to get overall structural response
- Results in the form of member forces and stress pattern
- May be a simple 2D beam/ frame model or a sophisticated full 3D FE model
- Generally adequate for design of usual structures



Local Modeling "Micro Model"



- Model of a single member or part of a member
- Model of the cross-section, opening, joints, connection
- Objective: to determine local stress concentration, cross-section behavior, modeling of cracking, bond, anchorage etc.
- Needs finite element modeling, often using very fine mesh
- Mostly suitable for research, simulation, experimental verification and theoretical studies



FEA Process

Engineer

- Evaluate Real Structure
- Create Structural Model
- Discretize Model in FE

Software

- Solve FE Model
- Interpret FEA Results

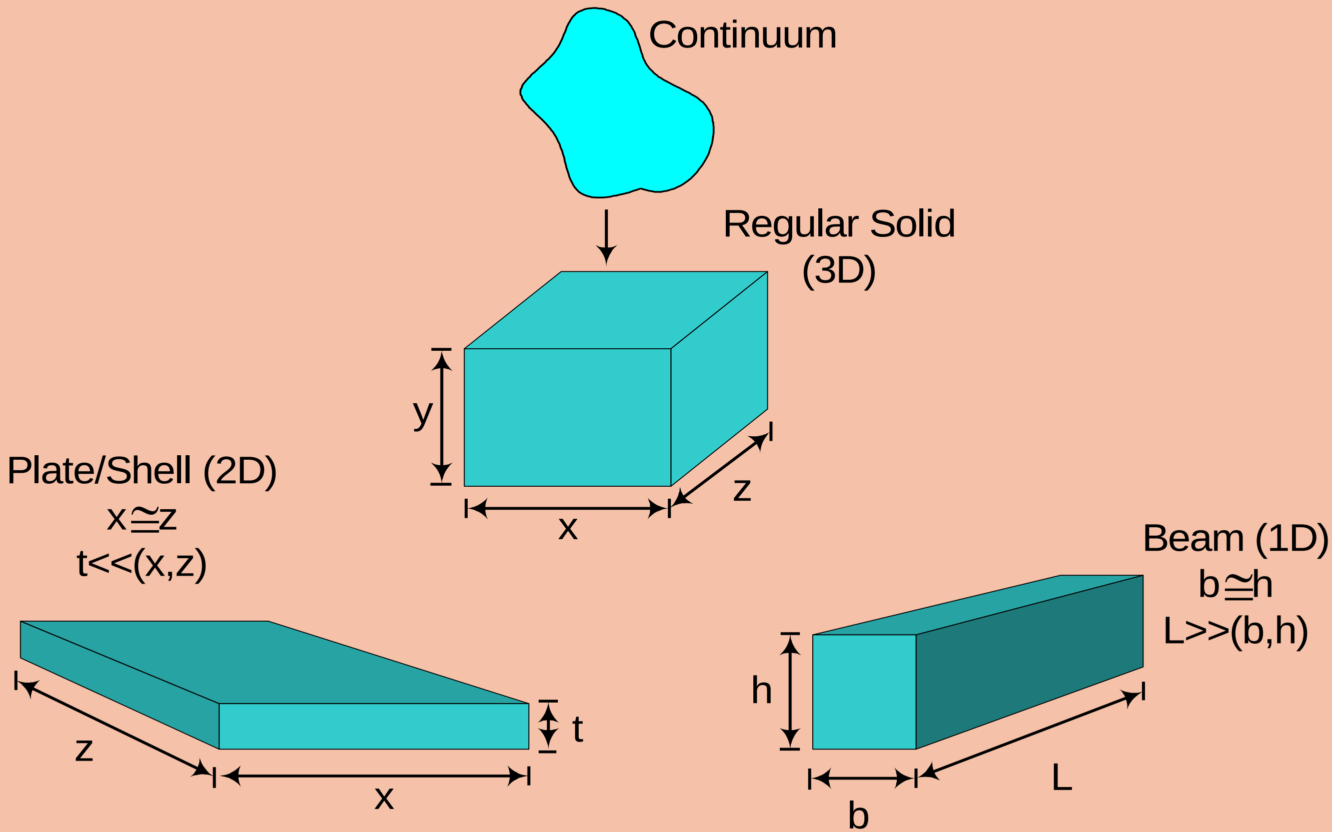
Engineer

- Physical significance of Results

Structure, Member, Element

- Structure can be considered as an assemblage of “Physical Components” called Members
  - Slabs, Beams, Columns, Footings, etc.
- Physical Members can be modeled by using one or more “Conceptual Components” called Elements
  - 1D elements **(Truss , beam**, **frame)** ,
  - 2D element **(plane stress**, **plane strain**, **plate**, **shell)**

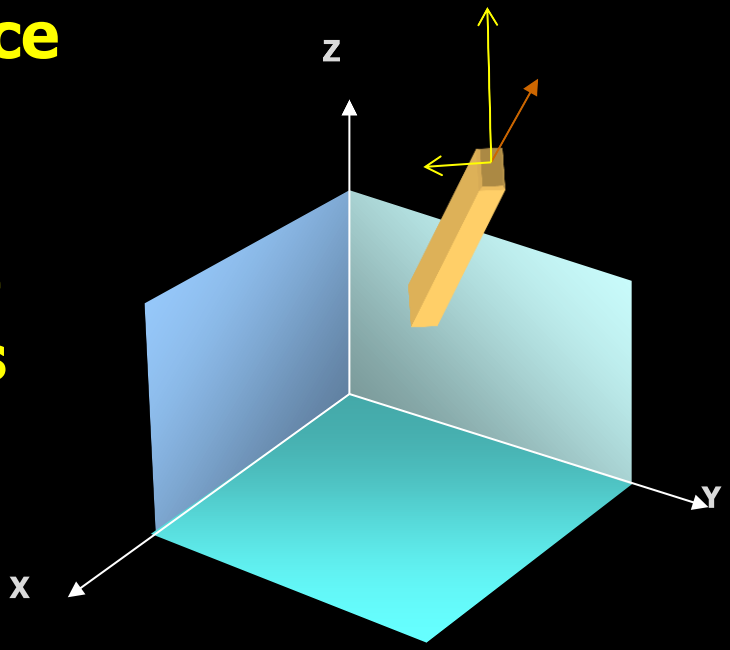
Structure, Member, Element



Dimensional Hierarchy of Structural Members

Structure, Member, Element

To convert continuum to structures, the first step is to define a finite number of reference dimensions



Global Axis used to reference the overall structure and to locate its components:
Also called the Structure Axis

Local Axis used to reference the quantities on part of a structure or a member or an element:
Also called the Member Axis or Element Axis

Degree of Freedom (DOF)

- In a continuum, each point can move in infinite ways
- In Structure, movement of each point is represented or resolved in limited number of ways, called Degrees Of Freedom (DOF)
- The DOF of range from 1 to 6 depending on type and level of structural model and the element being considered
- Global and Local DOF have different meaning and significance
- Relationship between Global, Local and Natural DOF is established through Transformation Matrices

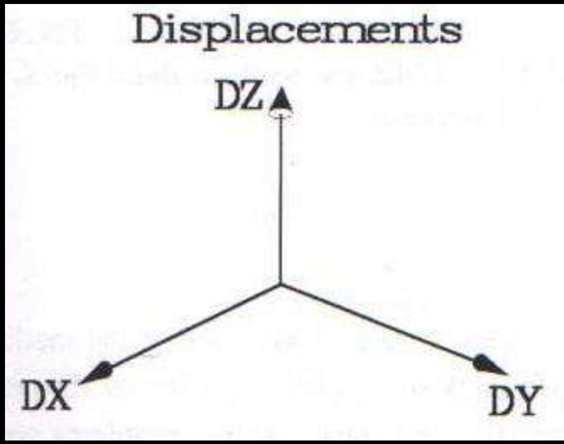
Basic 6-DOF

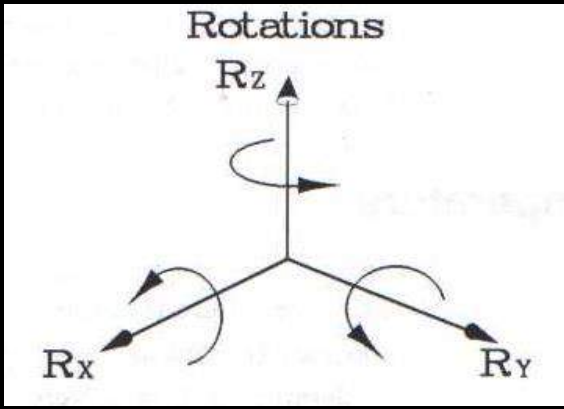
Three Translations along the reference axis

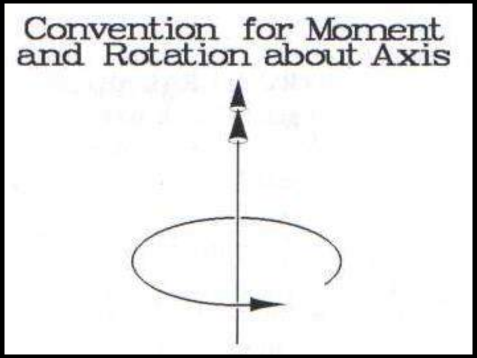
Dx, Dy, Dz

Three Rotations about the reference axis

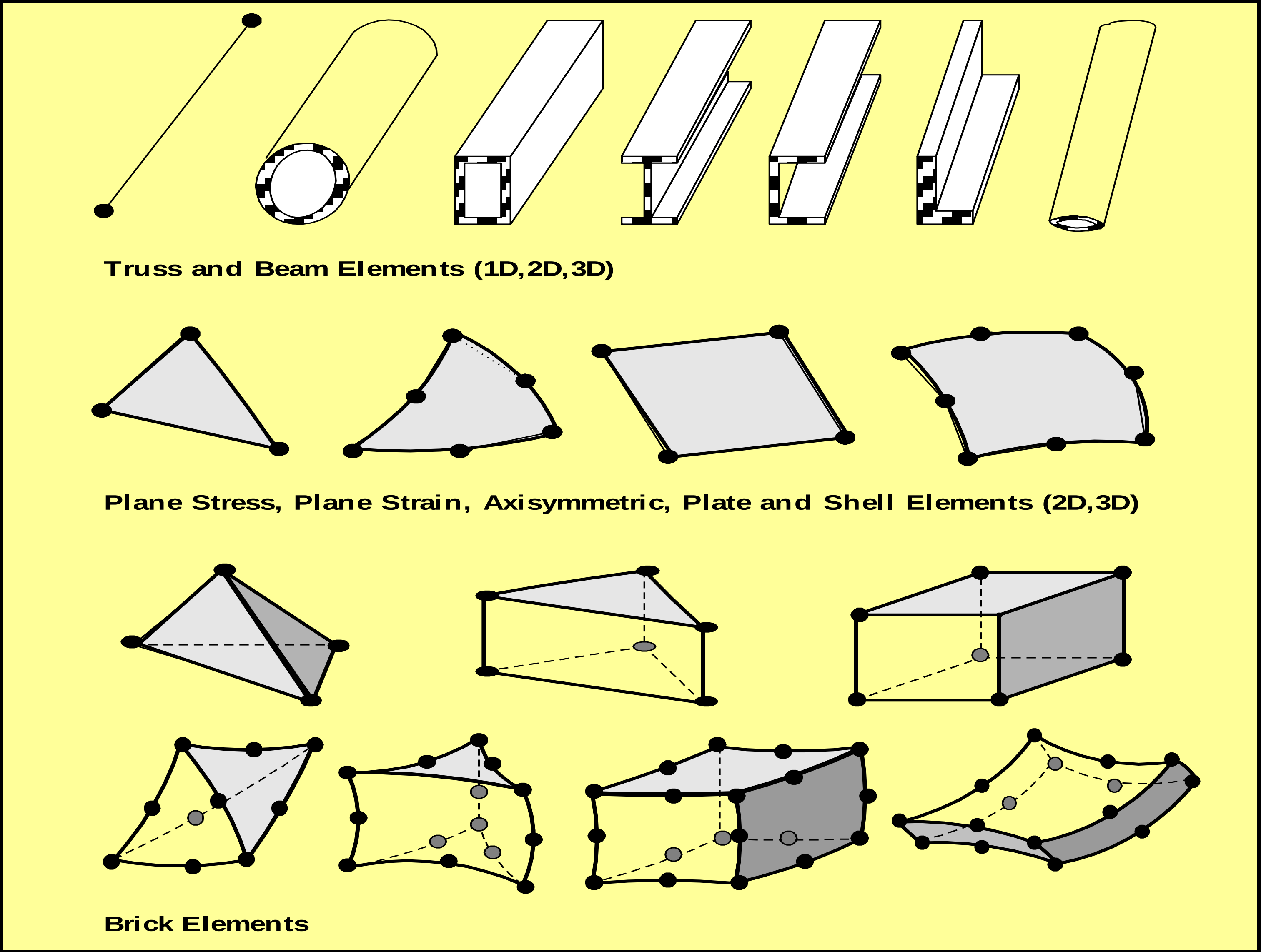
Rx, Ry, Rz







Some sample finite elements



Computing Stiffness

- 1 D Elements (Beam type)
  - Only one dimension is actually modeled as a line, other two dimensions are represented by stiffness properties
  - Can be used in 1D, 2D and 3D
- 2 D Elements (Plate type)
  - Only two dimensions are actually modeled as a surface, third dimension is represented by stiffness properties
  - Can be used in 2D and 3D Model
- 3 D Elements (Brick type)
  - All three dimensions are modeled as a solid
  - Can be used in 3D Model

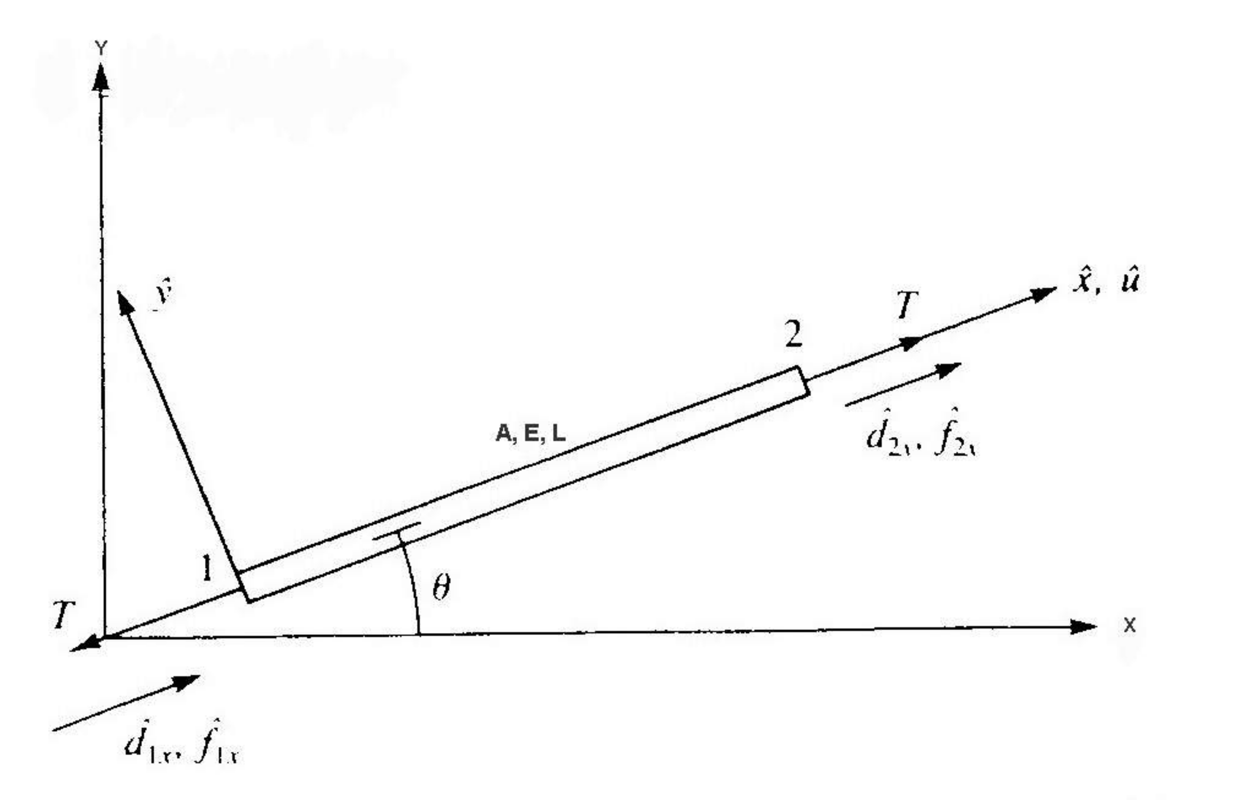
Basic properties of joints

- All elements are connected to the structure at the joints
- The structure is supported at the joints using Restraints and/or Springs
- Concentrated loads may be applied at the joints
- Lumped masses and rotational inertia may be placed at the joints
- Loads and masses applied to the elements are transferred to the joints
- Joints are the primary locations in the structure at which the displacements are known (the supports) or are to be determined

Bar (Truss) Element

- 1 DOF per node
- Normally 2 node per element

$$K = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

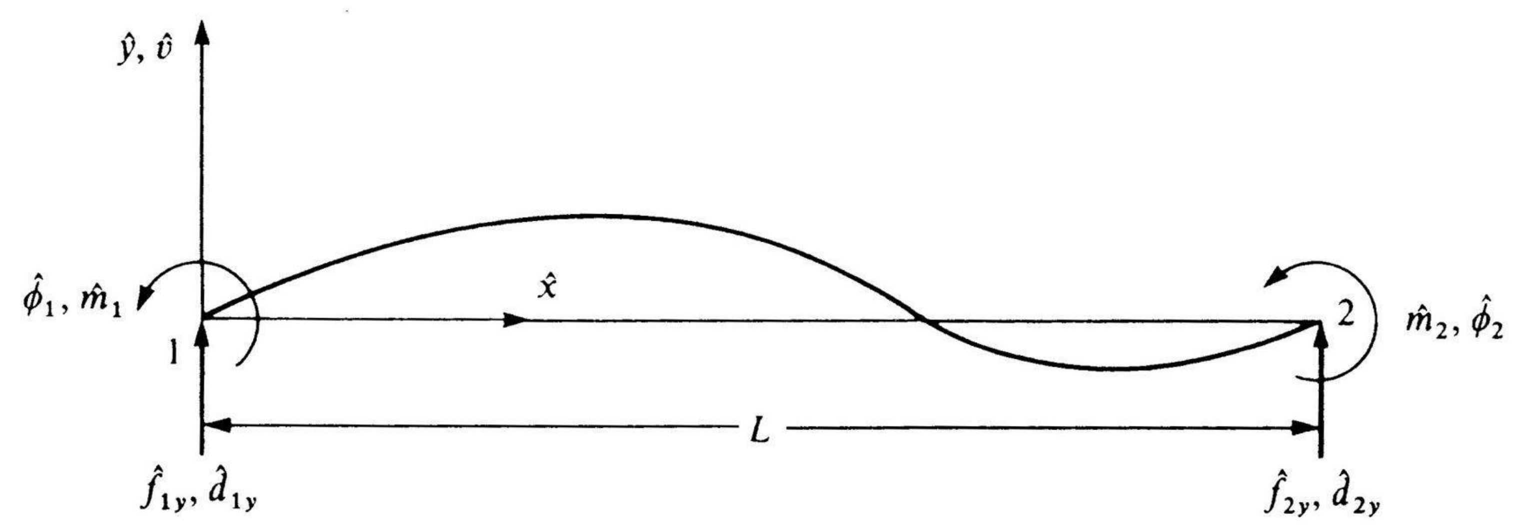


- The bar can not sustain shear force
- Any effect of transverse displacement does not induce axial force (small deflection)
- Axial force member without moment

Beam Element

- **2-DOF per node**
- **Normally 2 node per element**

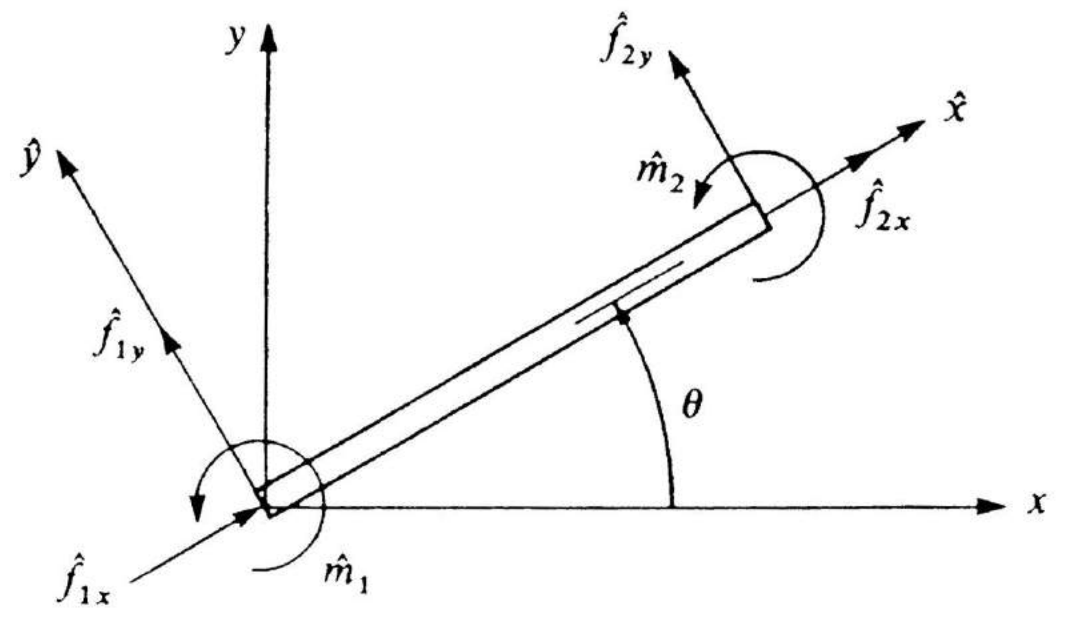
$$K = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$



- **No axial force**
- **Small deflection**
- **No shear deformation "Euler's beam"**

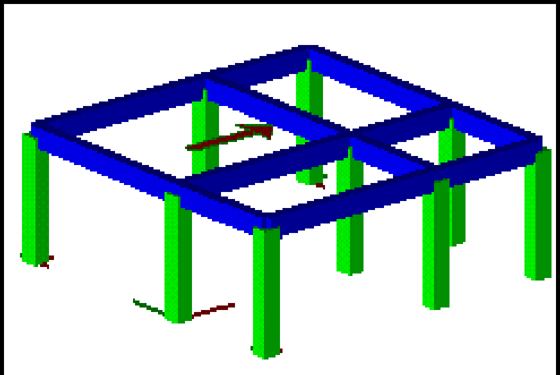
Frame Element

- 3-DOF per node
- Truss + Beam
- Normally 2 node per element

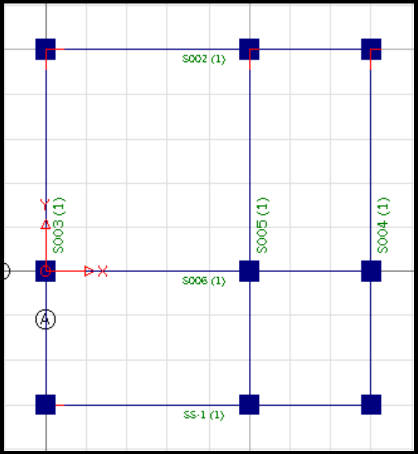


$$K = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ & \frac{12EI}{L^3} & \frac{6EI}{L^3} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^3} \\ & & \frac{4EI}{L} & 0 & -\frac{6EI}{L^3} & \frac{2EI}{L} \\ & & & \frac{AE}{L} & 0 & 0 \\ & & & & \frac{12EI}{L^3} & -\frac{6EI}{L^3} \\ & & & & & \frac{4EI}{L} \end{bmatrix}$$

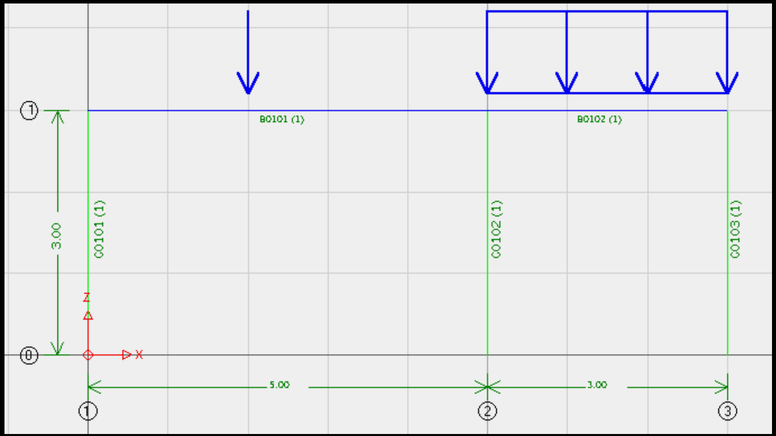
Useage 1D Elements



3D Frame



2D Grid



2D Frame

2D Plane element(membrane)

- 3-8 node per element
  - Total DOF per Node = 3 (or 2)
  - Total Displacements per Node = 2
  - Total Rotations per Node = 1 (or 0)
  - Membranes are modeled for flat surfaces
  - For Modeling surface elements carrying in-plane loads

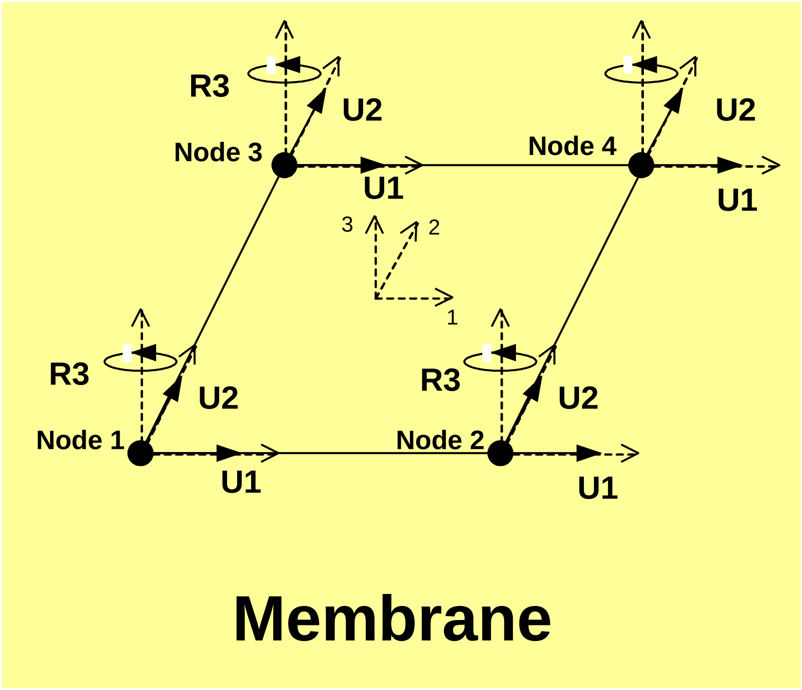
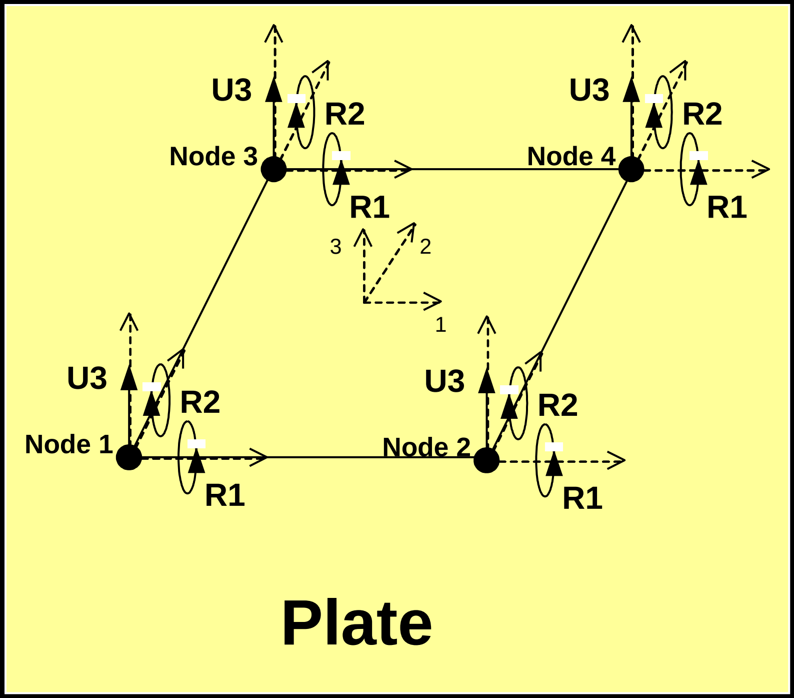


Plate element

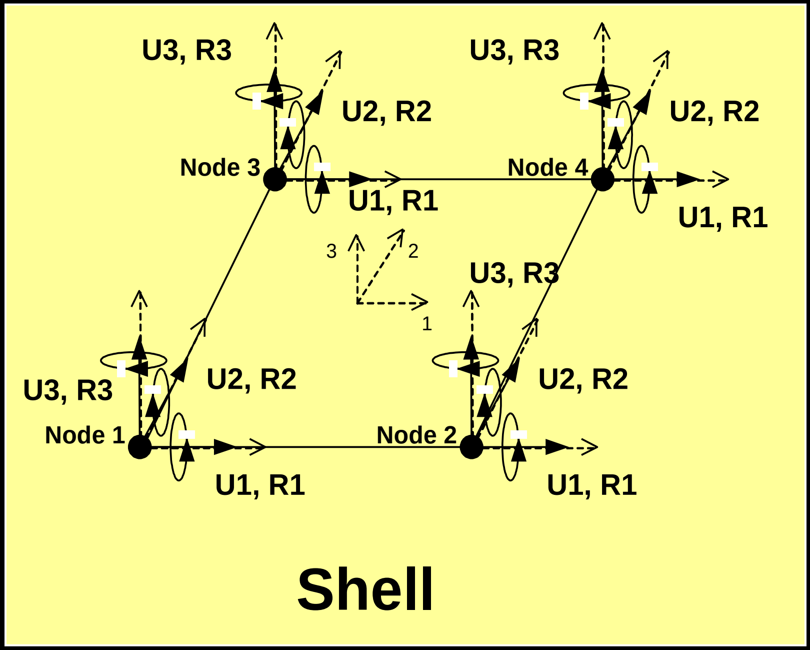
- Total DOF per Node = 3
- Total Displacements per Node = 1
- Total Rotations per Node = 2
- Plates are for flat surfaces

- For Modeling surface elements carrying out of plane loads



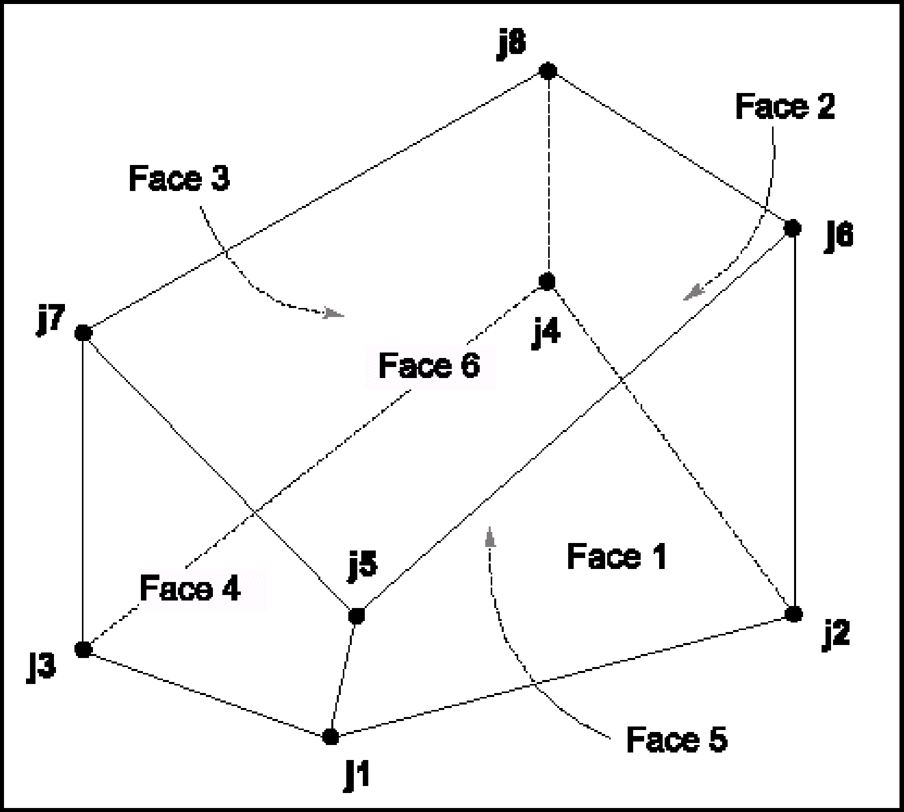
Shell element

- Total DOF per Node = 6 (or 5)
- Total Displacements per Node = 3
- Total Rotations per Node = 3
- Used for curved surfaces
- For Modeling surface elements carrying general loads



Solid element

- 6-20 node element
- Common element in 3D stress analysis
- Popular in mechanical engineering
- In civil engineering, used to investigate local effects



Thank you