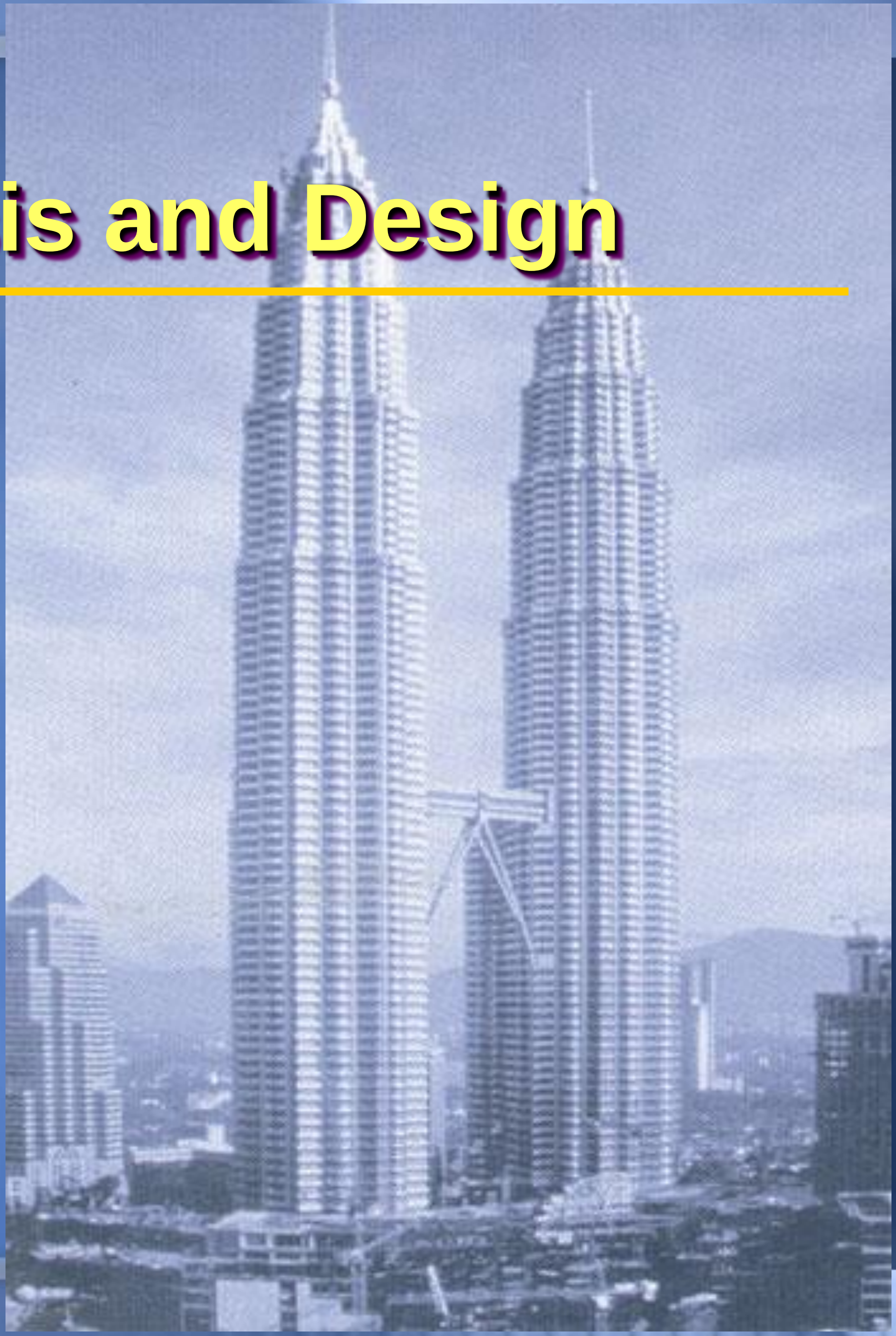
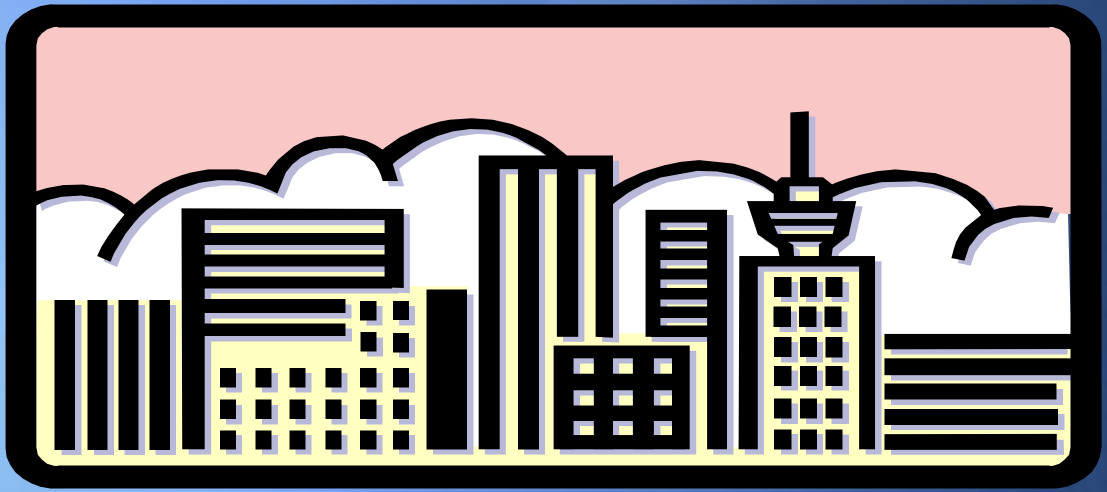
Computer Aided Analysis and Design Of Building Structures



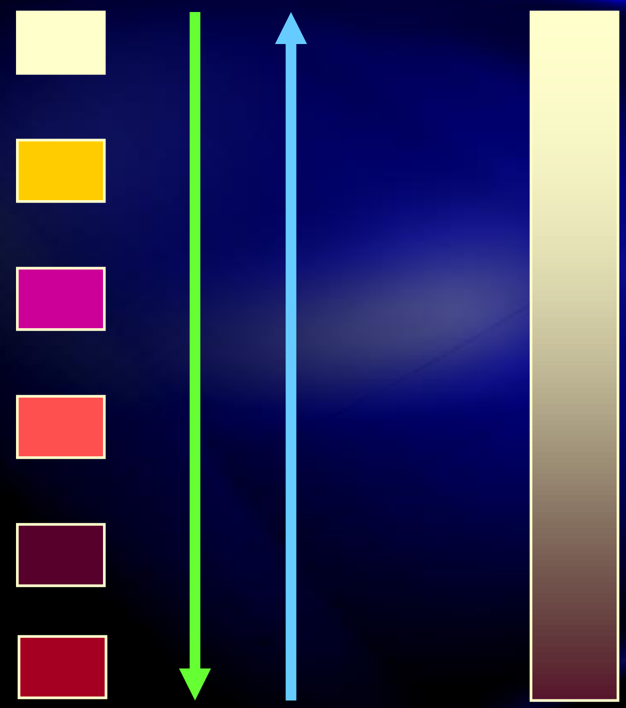
Building Structures
Modeling and Analysis Concepts



Asian Center for Engineering Computations and Software, ACECOMS, AIT

Overall Design Process

- Conception
- Modeling
- Analysis
- Design
- Detailing
- Drafting
- Costing

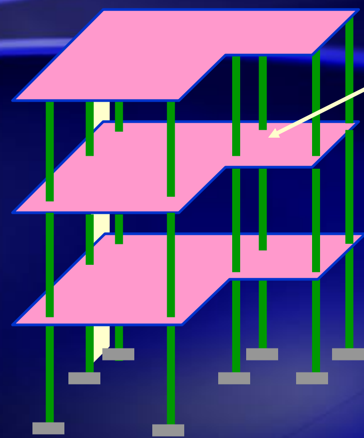


Integrated Design Process

Building Systems

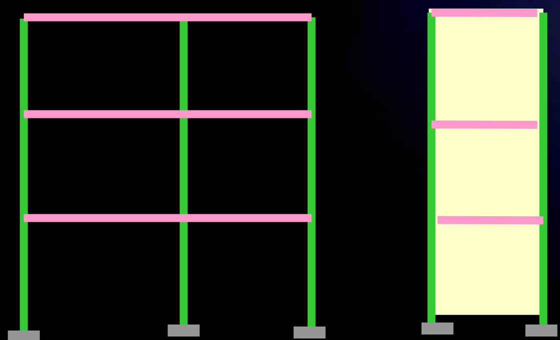
- **Building is an assemblage of various Systems**
  - Basic Functional System
  - Structural System
  - HVAC System
  - Plumbing and Drainage System
  - Electrical, Electronic and Communication System
  - Security System
  - Other specialized systems

Building Structure

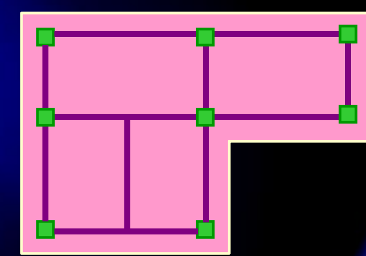


Floor Diaphragm

Frame and Shear Walls
Lateral Load Resisting System



Floor Slab System
Gravity Load Resisting System



Sub-structure and Member Design

Beams, Columns, Two-way Slabs, Flat Slabs, Pile caps
Shear Walls, Deep Beams, Isolated Footings, Combined Footings

The Building Structural System - Conceptual

- **The Gravity Load Resisting System (GLRS)**
  - The structural system (beams, slab, girders, columns, etc) that act primarily to support the gravity or vertical loads
- **The Lateral Load Resisting System (LLRS)**
  - The structural system (columns, shear walls, bracing, etc) that primarily acts to resist the lateral loads
- **The Floor Diaphragm (FD)**
  - The structural system that transfers lateral loads to the lateral load resisting system and provides in-plane floor stiffness

Building Response

- **Objective: To determine the load path gravity and lateral loads**

- **For Gravity Loads - How Gravity Loads are Distributed**
  - Analysis of Gravity Load Resisting System for:
    - *Dead Load, Live Live Load, Pattern Loads, temperature, shrinkage*
  - Important Elements: Floor slabs, beams, openings, Joists, etc.

- **For Lateral Loads – How Lateral Loads are Distributed**
  - Analysis of Lateral Load Resisting System for:
    - *Wind Loads, Seismic Loads, Structural Un-symmetry*
  - Important elements: Columns, shear walls, bracing , beams

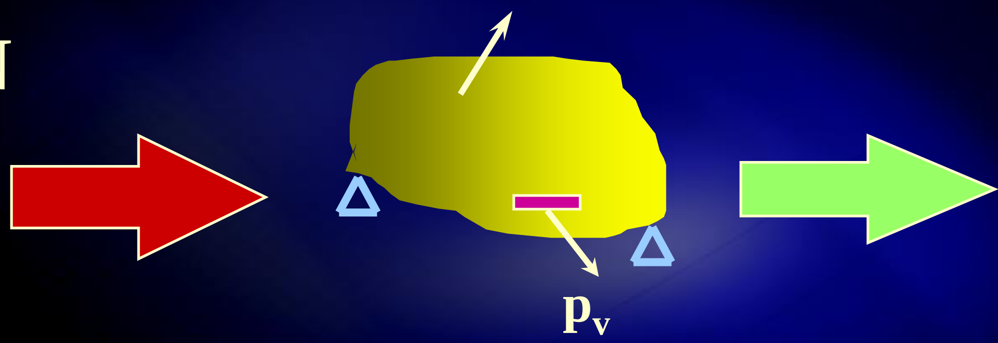
Structural Response To Loads

The Simplified Structural System

STRUCTURE

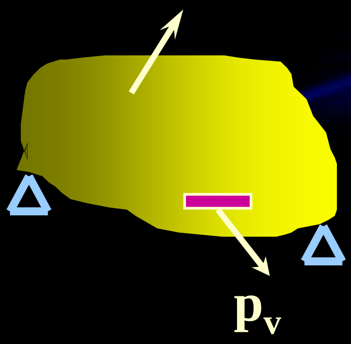
EXCITATION

Loads
Vibrations
Settlements
Thermal Changes



RESPONSES

Displacements
Strains
Stress
Stress Resultants



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

Real Structure is governed by "Partial Differential Equations" of various order

Direct solution is only possible for:

- Simple geometry
- Simple Boundary
- Simple Loading.

The Need for Modeling

A - Real Structure cannot be Analyzed:
It can only be "Load Tested" to determine response

B - We can only analyze a "Model" of the Structure

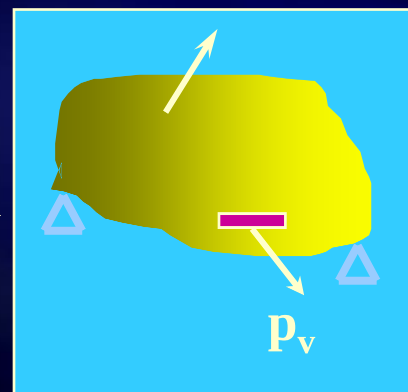
C - We therefore need tools to Model the Structure and to Analyze the Model

The Need for Structural Model

STRUCTURE

EXCITATION

Loads
Vibrations
Settlements
Thermal Changes



RESPONSES

Displacements
Strains
Stress
Stress Resultants

Structural Model

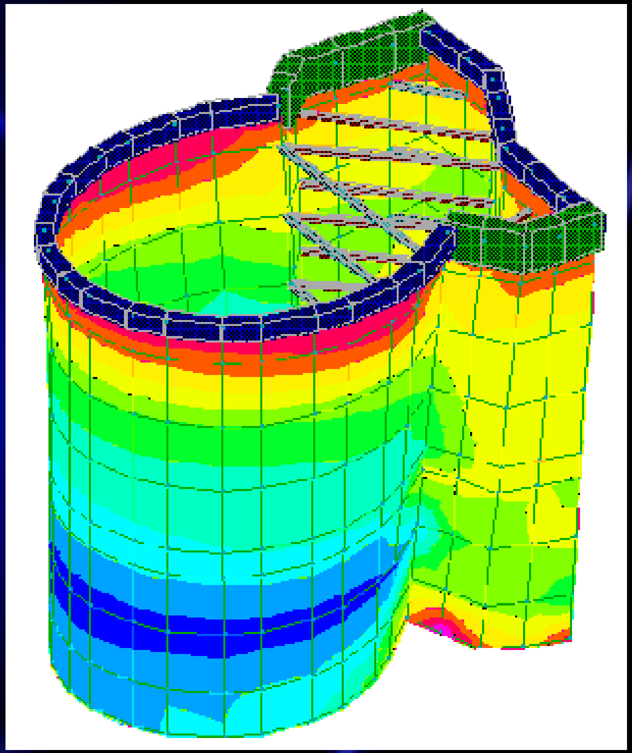
Finite Element Method: The Analysis Tool

- **Finite Element Analysis (FEA)**

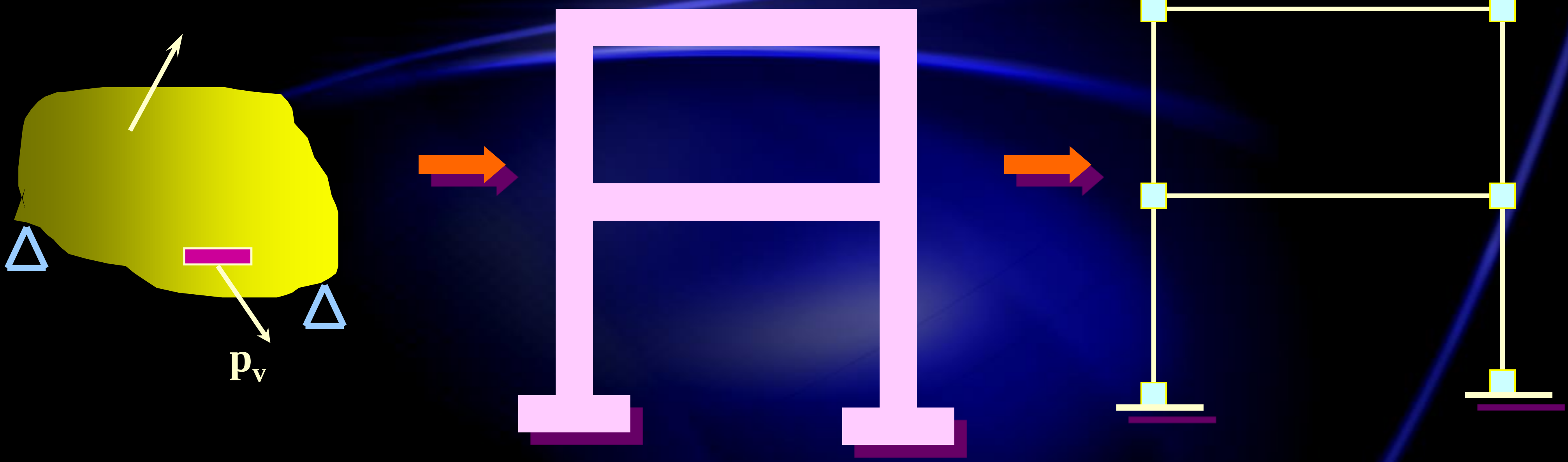
  "A discretized solution to a continuum problem using FEM"

- **Finite Element Method (FEM)**

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



Continuum to Discrete 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)

From Classical to FEM Solution

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

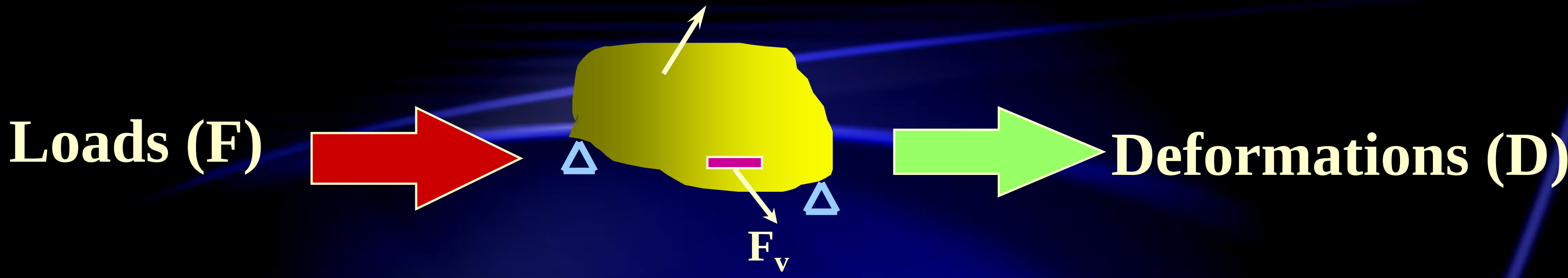
Structural Model

$$Kr = R$$

"Algebraic Equations"

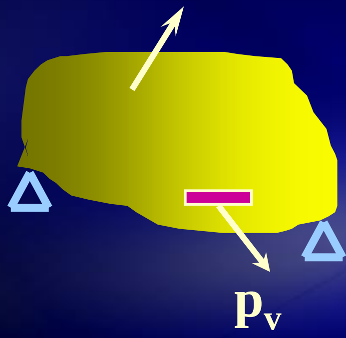
K = Stiffness
r = Response
R = Loads

Simplified Structural System



$$F = K\,D$$

STRUCTURE

EXCITATION → 

→ RESPONSES

- Static
- Dynamic

- Elastic
- Inelastic

- Linear
- Nonlinear

The Equilibrium Equations

1. **Linear-Static** **Elastic OR Inelastic**

$$Ku = F$$

2. **Linear-Dynamic Elastic**

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) = F(t)$$

3. **Nonlinear - Static** **Elastic OR Inelastic**

$$Ku + F_{NL} = F$$

4. **Nonlinear-Dynamic** **Elastic OR Inelastic**

$$M\ddot{u}(t) + C\dot{u}(t) + Ku(t) + F(t)_{NL} = F(t)$$

Basic Steps in FEA

- Evaluate Real Structure
- Create Structural Model
- Discretize Model in FE
- Solve FE Model
- Interpret FEA Results
- Physical significance of Results

Legend:
- Engineer
- Engineer + Software
- Software

Discretization of Continuums

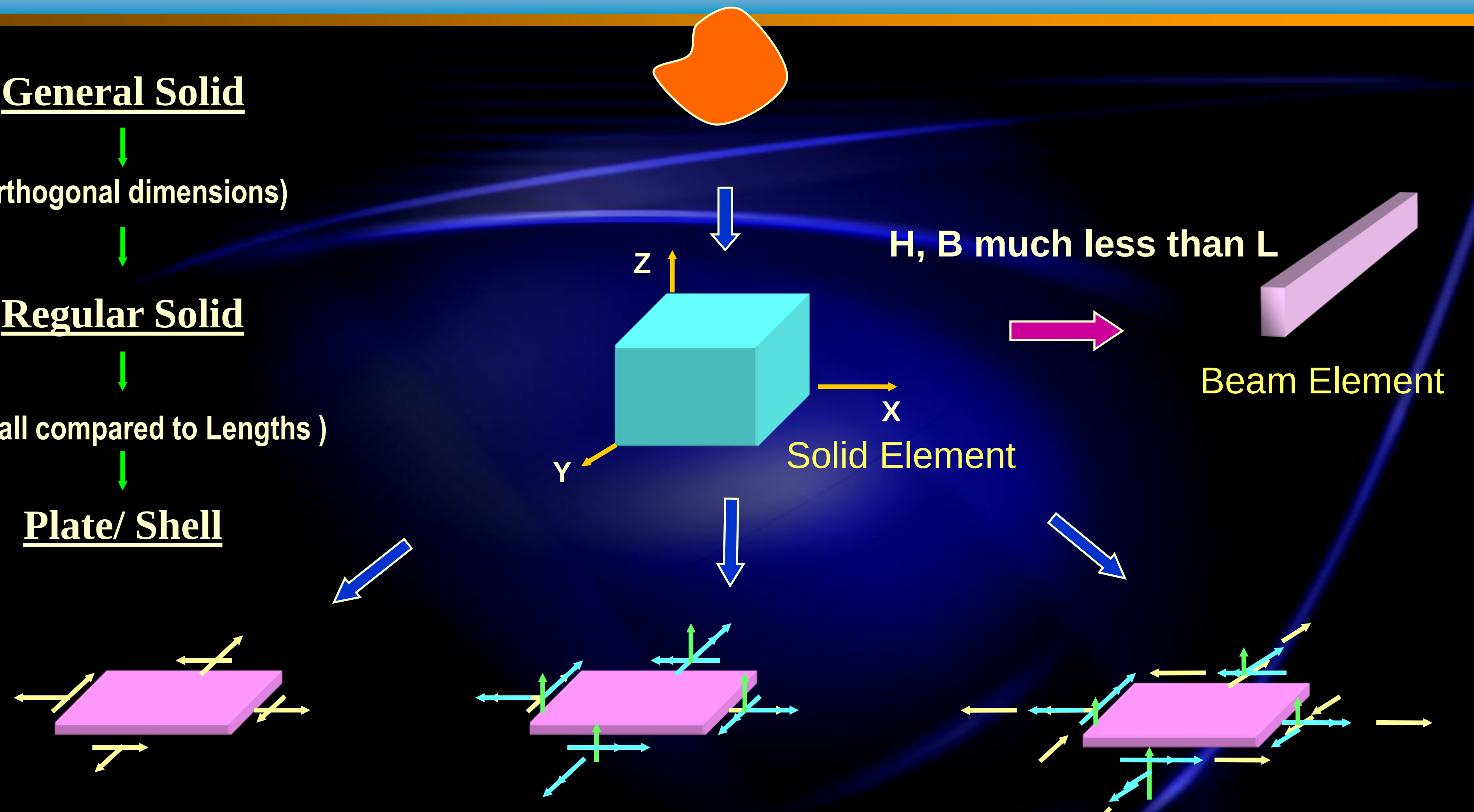
General Solid

(Orthogonal dimensions)

Regular Solid

(T small compared to Lengths)

Plate/ Shell



Membrane/ Panel
In-Plane, Only Axial

Plate/ Slab
Out of Plane, Only Bending

Shell
In-Plane and Bending

Global Modeling of Structural Geometry

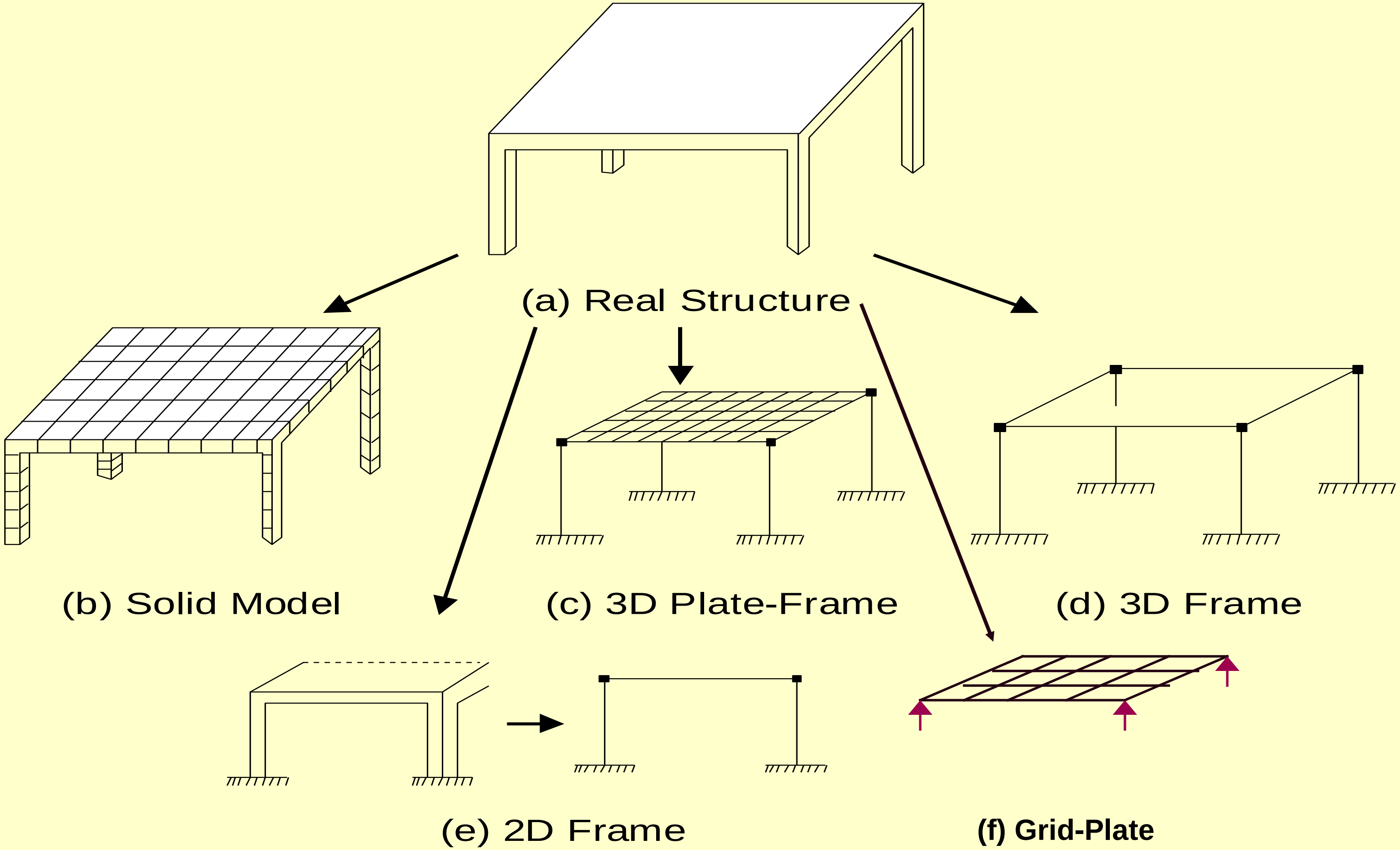
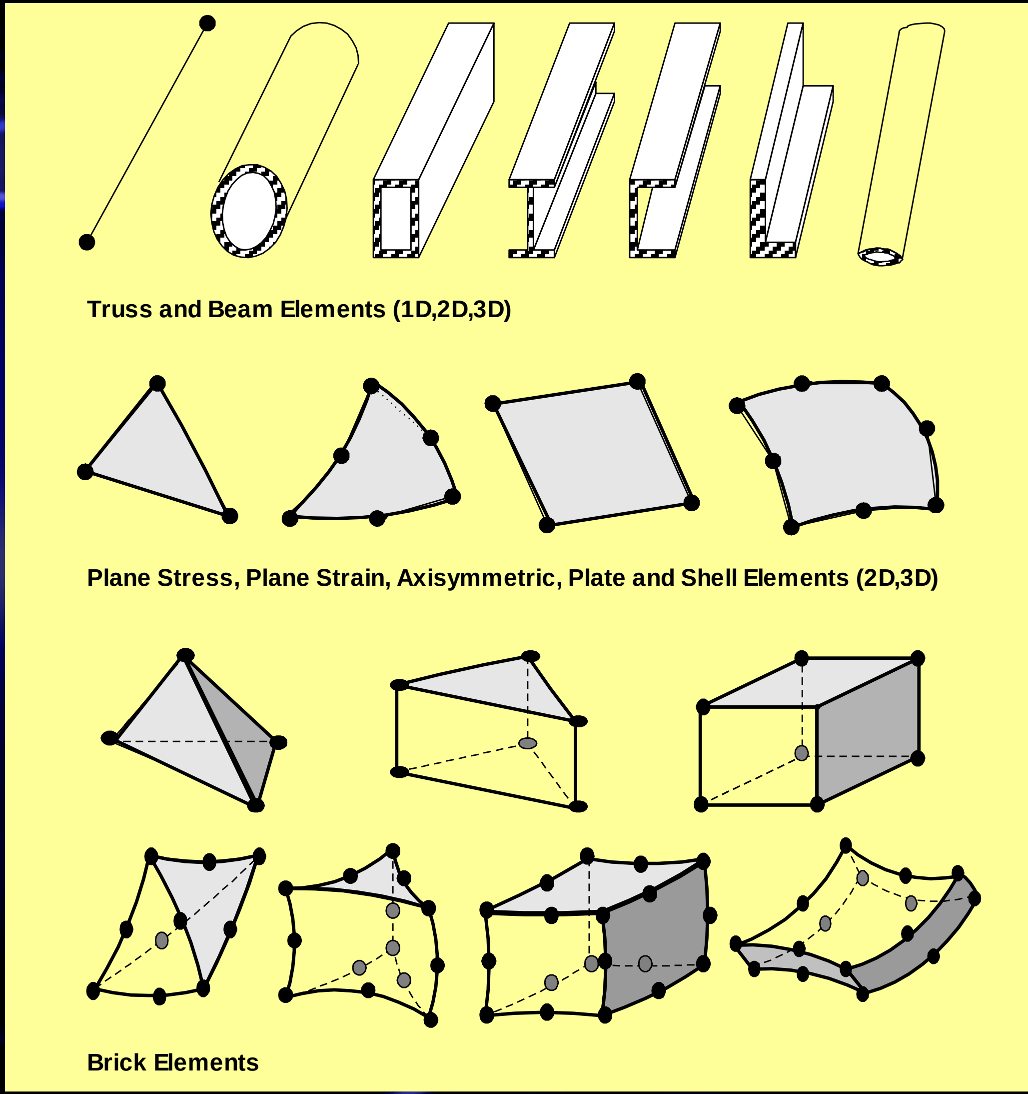


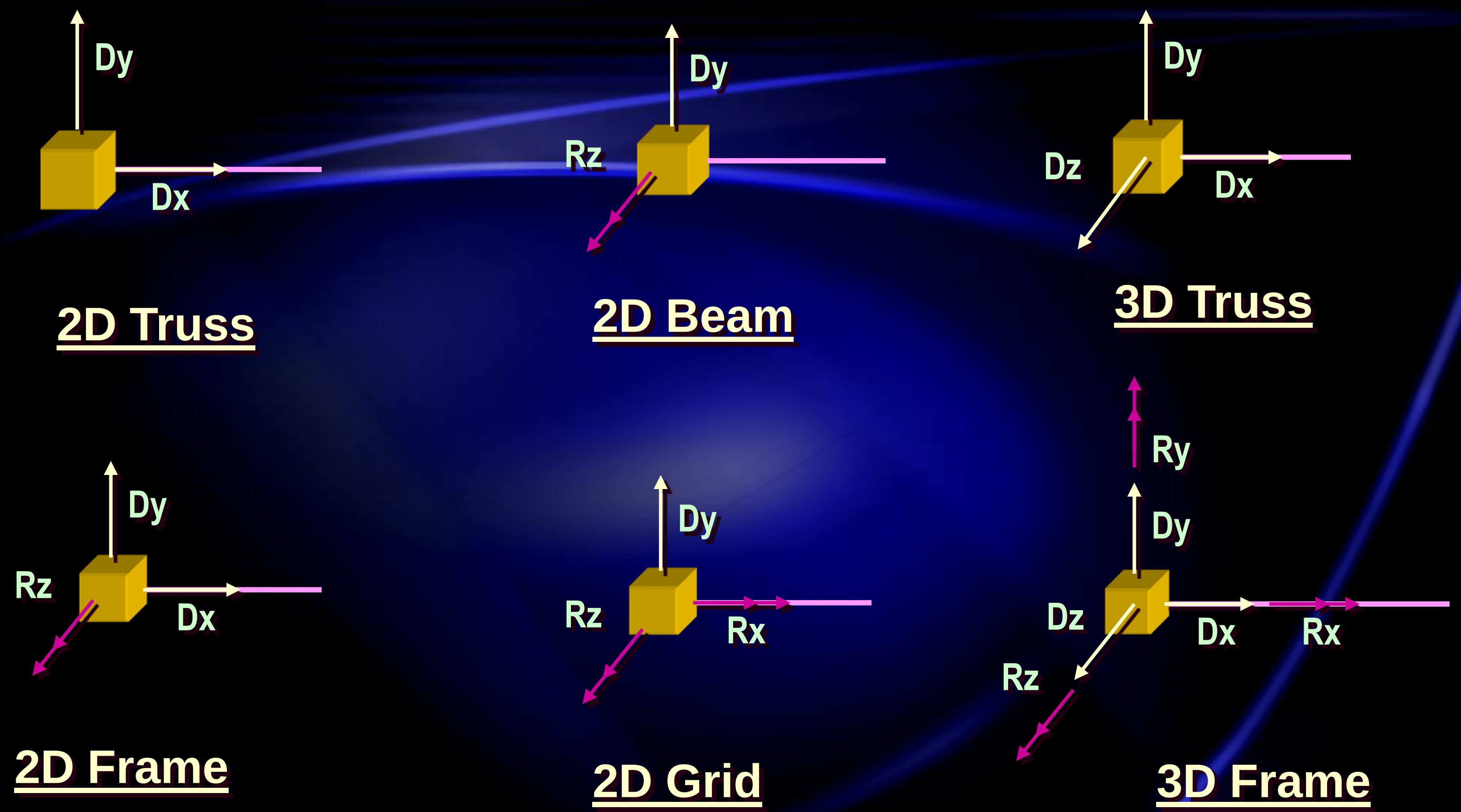
Fig. 1 Various Ways to Model a Real Struture

Dimensions of Elements

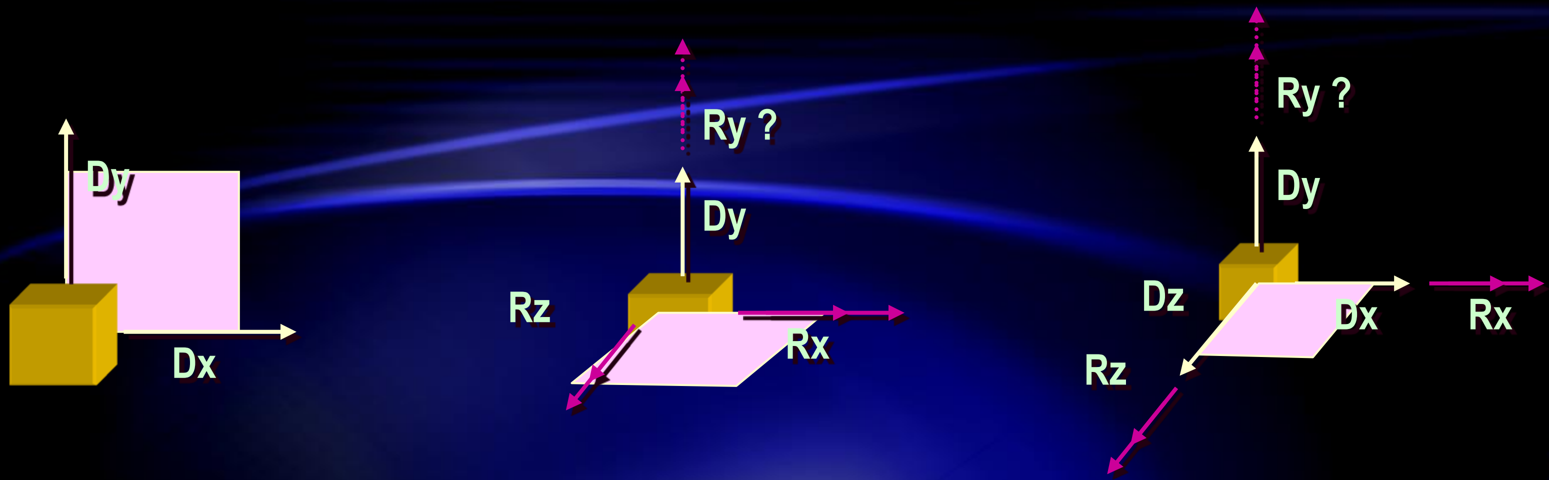
- **1 D Elements (Beam type)**
  - Can be used in 1D, 2D and 3D
  - 2-3 Nodes. A, I etc.
- **2 D Elements (Plate type)**
  - Can be used in 2D and 3D Model
  - 3-9 nodes. Thickness
- **3 D Elements (Brick type)**
  - Can be used in 3D Model
  - 6-20 Nodes.



DOF for 1D Elements



DOF for 2D Elements

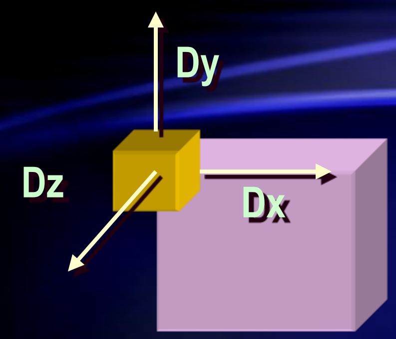


Membrane

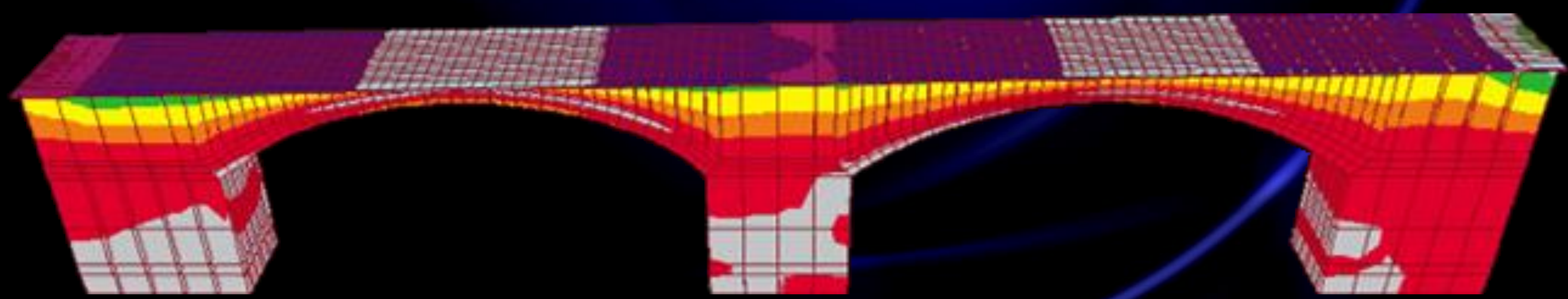
Plate

Shell

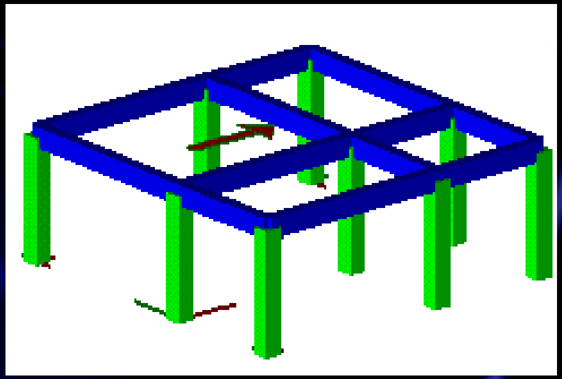
DOF for 3D Elements



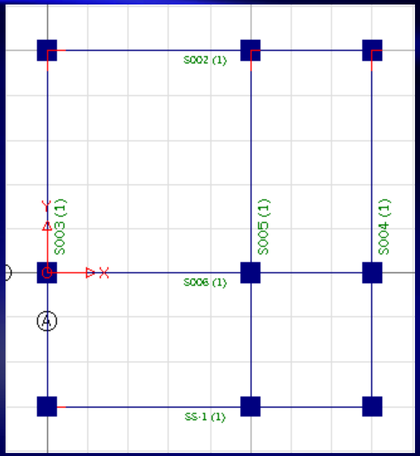
Solid/ Brick



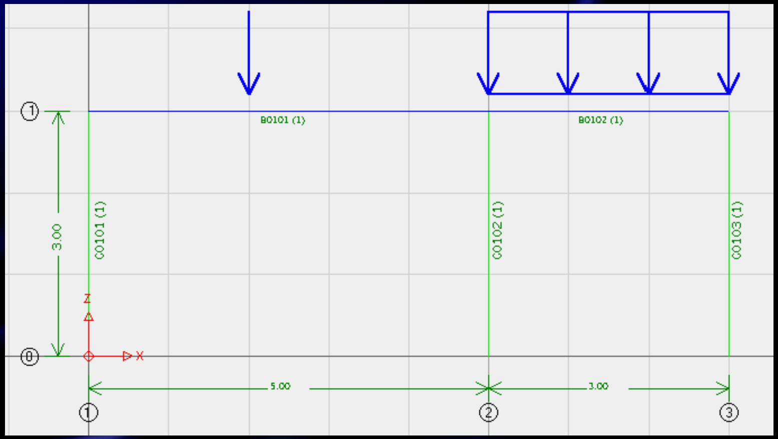
- The structure represented by rod or bar type elements
- Does not model the cross-section dimensions
- Suitable for skeletal structures
- The simplest and easiest model to construct, analyze and interpret
- Can be in 2D or in 3D space



3D Frame



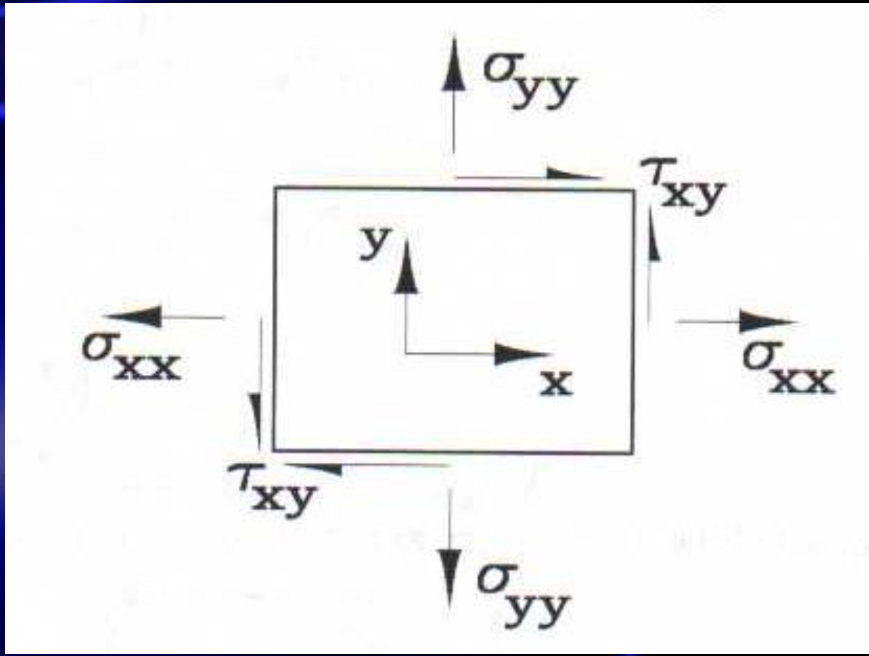
2D Grid



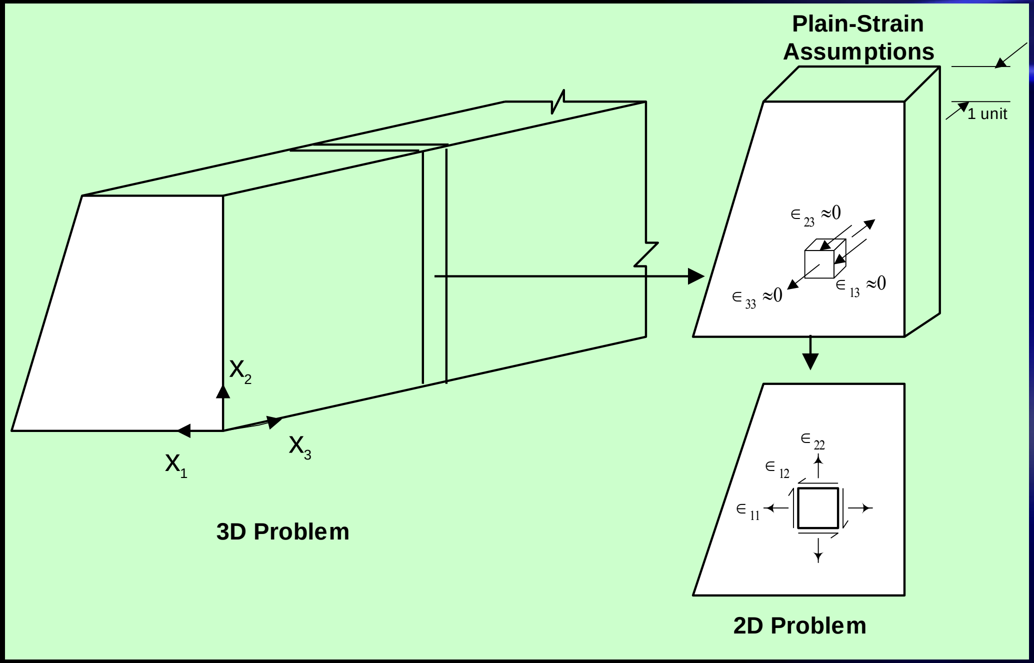
2D Frame

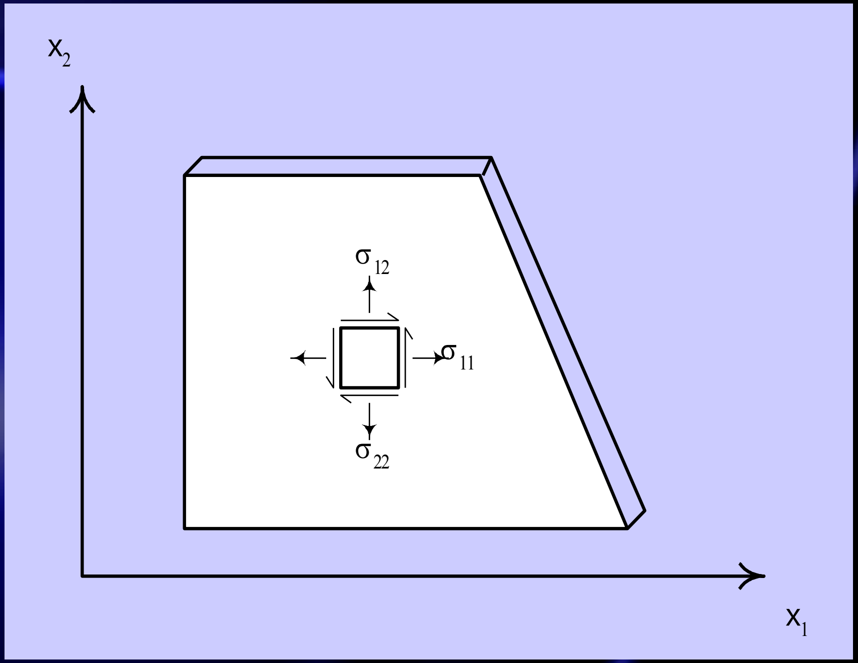
Membrane Model

- Ignore bending stiffness
- Tension / Compression
- In- plane Shear
- For in plane loads
- Principle Stresses
- Suitable for very thin structures / members
- Thin Walled Shells



Plane Stress and Plane Strain



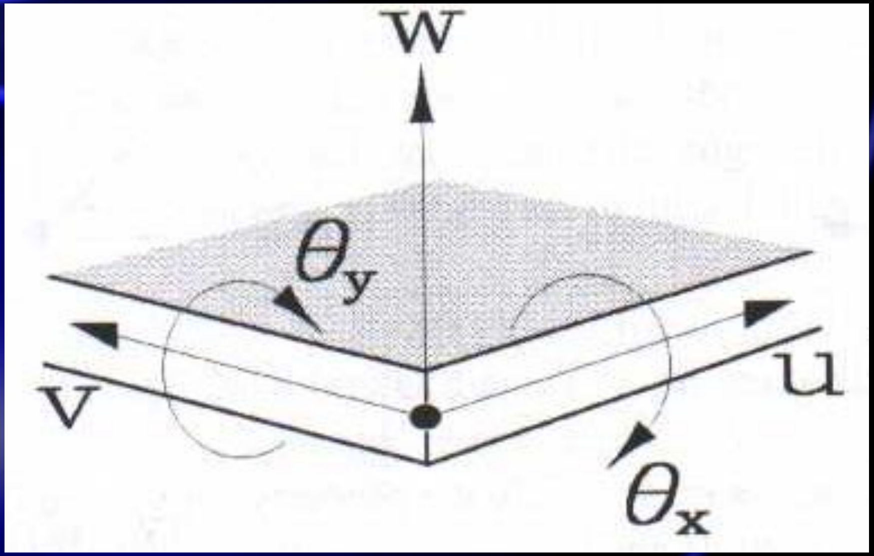


Plane Strain Problem

Plane Stress Problem

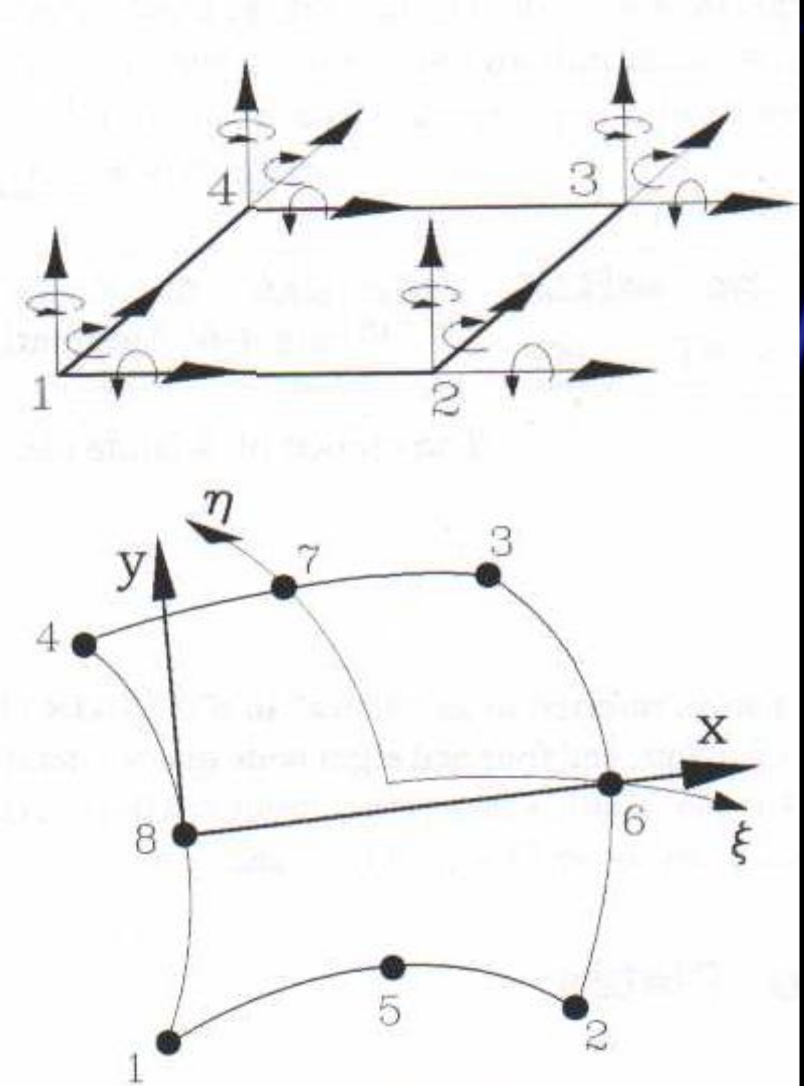
Plate Bending Model

- Primarily Bending mode
- Moment and Shear are predominant
- Suitable for moderately thick slabs and plates
- For Out-of-plane loads only
- Can be used in 3D or 2D models
- Suitable for planks and relatively flat structures



General Plate-Shell Model

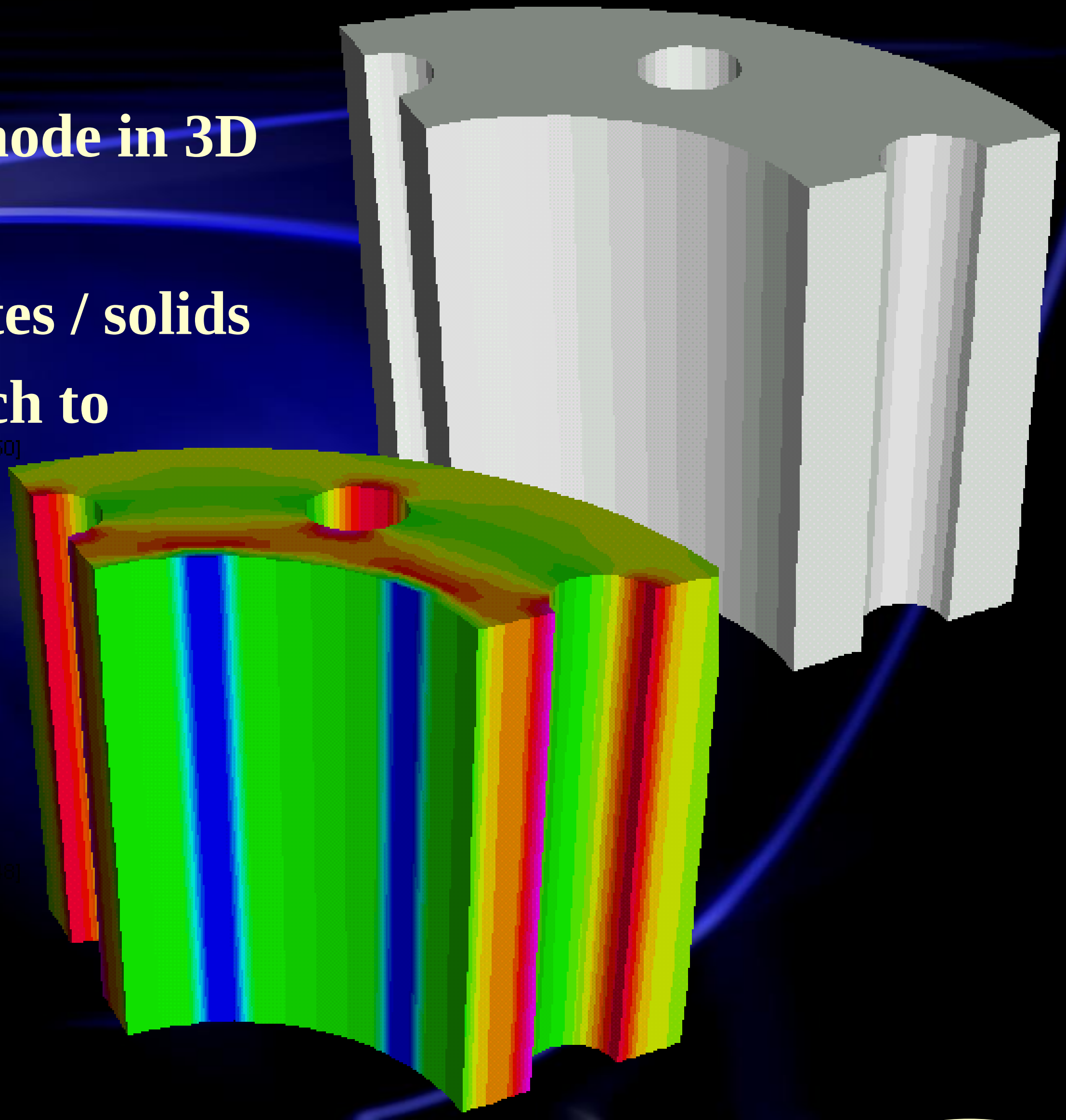
- Combined Membrane and Plate
- Suitable for general application to surface structures
- Suitable for curved structures
- Thick shell and thin shell implementations available
- Membrane thickness and plate thickness can be specified separately
- Numerous results generated. Difficult to design the section for combined actions



Solid Model

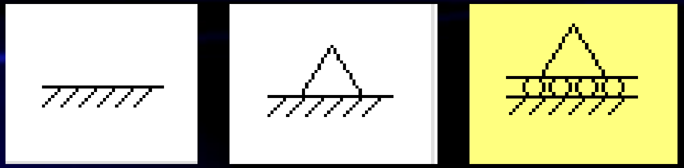
- Shear Axial deformation mode in 3D
- Suitable for micro-models
- Suitable for very thick plates / solids
- May not be applicable much to ferocement structures

- Use 6 to 20 node elements

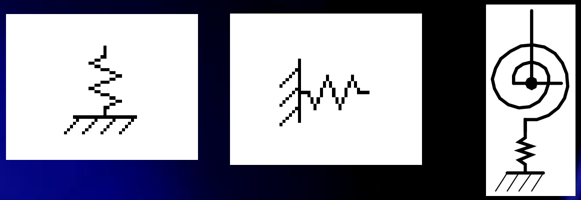


Soil-Structure Interaction

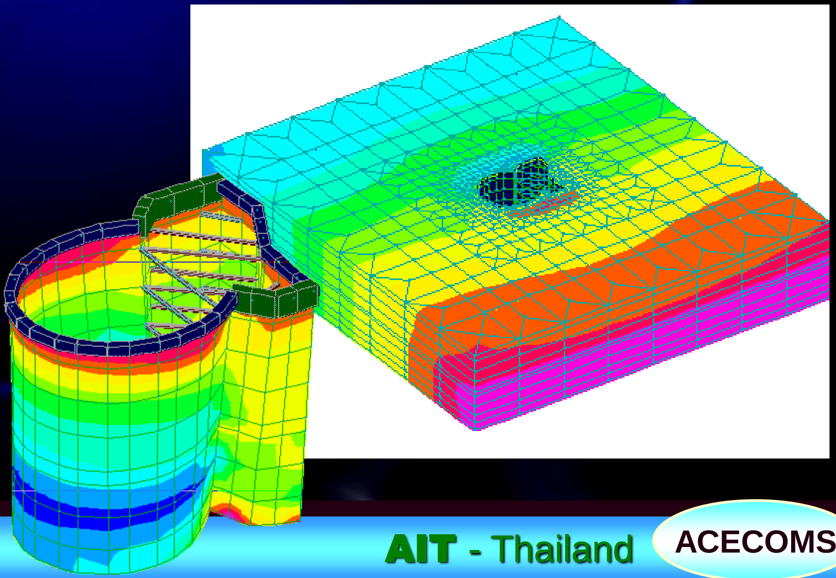
- Simple Supports
  - Fix, Pin, Roller etc.
  - Support Settlement



- Elastic Supports
  - Spring to represent soil
  - Using Modulus of Sub-grade reaction



- Full Structure-Soil Model
  - Use 2D plane stress elements
  - Use 3D Solid Elements



Connecting Different Types of Elements

	Truss	Frame	Membrane	Plate	Shell	Solid
Truss	OK	OK	Dz	OK	OK	OK
Frame	Rx, Ry, Rz	OK	Rx, Ry, Rz, Dz	Rx ? Dx, Dy	Rx ?	Rx, Ry, Rz
Membrane	OK	OK	OK	Dx, Dy	OK	OK
Plate	Rx, Rz	OK	Rx, Rz	OK	OK	Rx, Rz
Shell	Rx, Ry, Rz	OK	Rx, Ry, Rz, Dz	Dx, Dz	OK	Rx, Rz
Solid	OK	OK	Dz	Dx, Dz	OK	OK

What Type of Analysis should be Carried Out?

The type of Analysis to be carried out depends on the Structural System

- The Type of Excitation (Loads)
- The Type Structure (Material and Geometry)
- The Type Response

Basic Analysis Types

Excitation	Structure	Response	Basic Analysis Type
Static	Elastic	Linear	Linear-Elastic-Static Analysis
Static	Elastic	Nonlinear	Nonlinear-Elastic-Static Analysis
Static	Inelastic	Linear	Linear-Inelastic-Static Analysis
Static	Inelastic	Nonlinear	Nonlinear-Inelastic-Static Analysis
Dynamic	Elastic	Linear	Linear-Elastic-Dynamic Analysis
Dynamic	Elastic	Nonlinear	Nonlinear-Elastic-Dynamic Analysis
Dynamic	Inelastic	Linear	Linear-Inelastic-Dynamic Analysis
Dynamic	Inelastic	Nonlinear	Nonlinear-Inelastic-Dynamic Analysis

Some More Solution Types

- **Non-linear Analysis**
  - P-Delta Analysis
  - Buckling Analysis
  - Static Pushover Analysis
  - Fast Non-Linear Analysis (FNA)
  - Large Displacement Analysis

- **Dynamic Analysis**
  - Free Vibration and Modal Analysis
  - Response Spectrum Analysis
  - Steady State Dynamic Analysis

Static Vs Dynamic

- **Static Excitation**
  - When the Excitation (Load) does not vary rapidly with Time
  - When the Load can be assumed to be applied "Slowly"
- **Dynamic Excitation**
  - When the Excitation varies rapidly with Time
  - When the "Inertial Force" becomes significant

- **Most Real Excitation are Dynamic but are considered "Quasi Static"**
- **Most Dynamic Excitation can be converted to "Equivalent Static Loads"**

Elastic Vs Inelastic

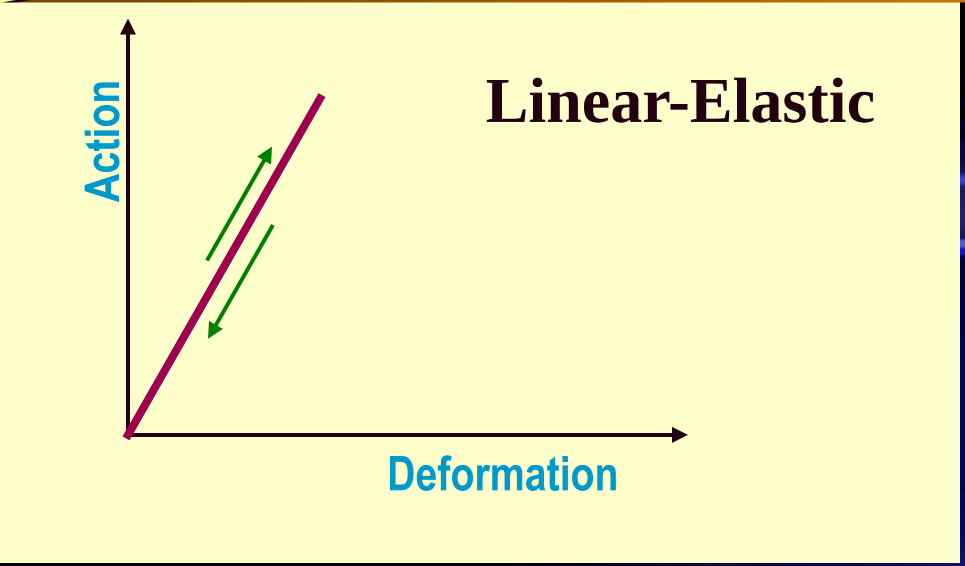
- **Elastic Material**
  - Follows the same path during loading and unloading and returns to initial state of deformation, stress, strain etc. after removal of load/ excitation
- **Inelastic Material**
  - Does not follow the same path during loading and unloading and may not returns to initial state of deformation, stress, strain etc. after removal of load/ excitation

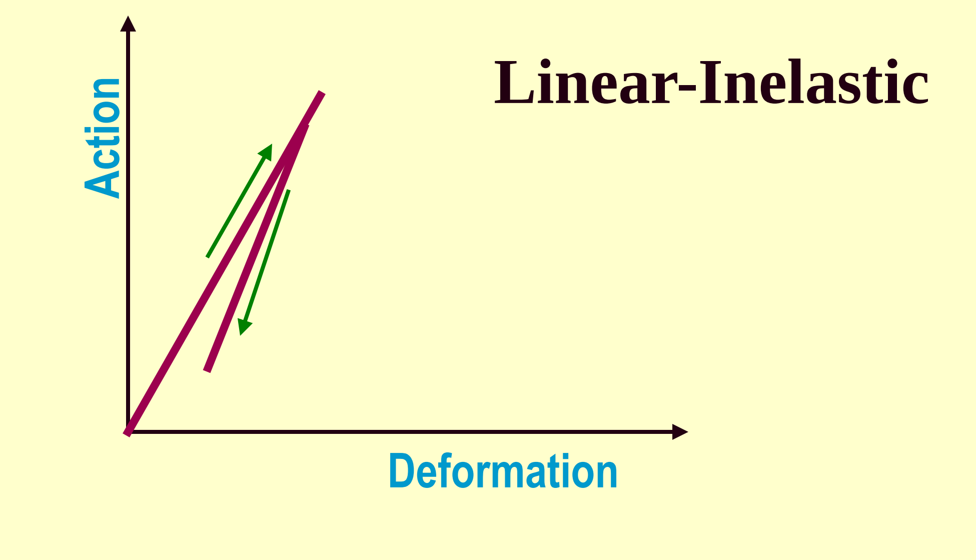
- **Most materials exhibit both, elastic and inelastic behavior depending upon level of loading.**

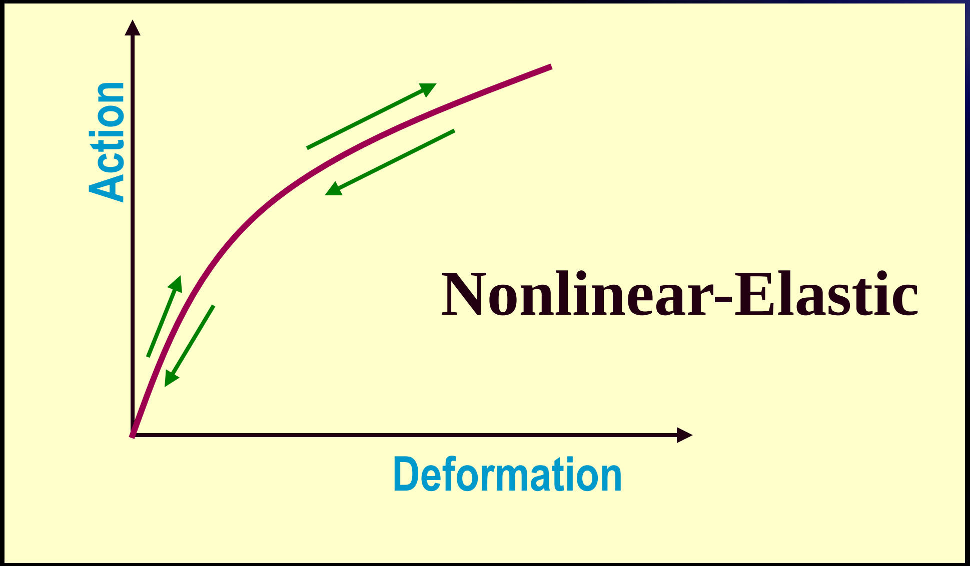
Linear Vs Nonlinear

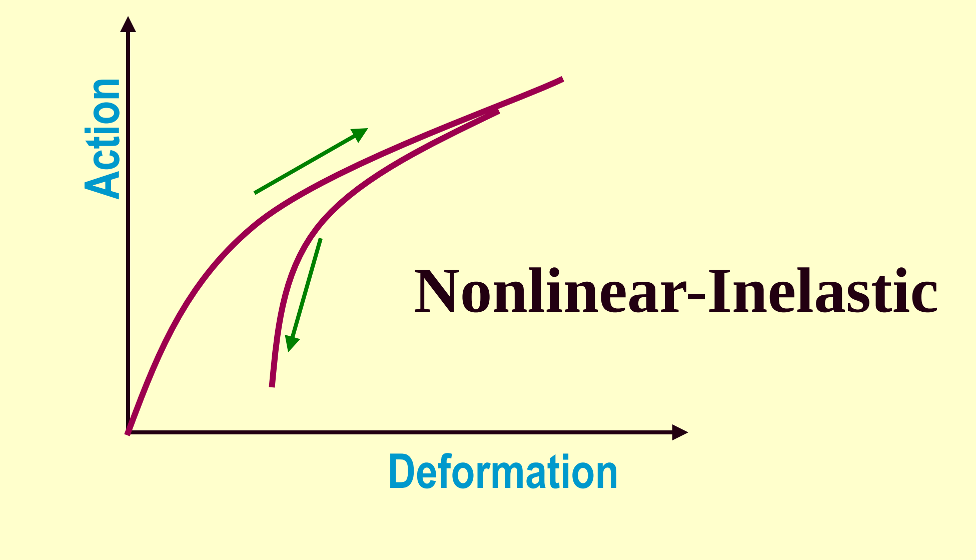
- **Linearity**
  - The response is directly proportional to excitation
  - (Deflection doubles if load is doubled)
- **Non-Linearity**
  - The response is not directly proportional to excitation
  - (deflection may become 4 times if load is doubled)
- **Non-linear response may be produced by:**
  - Geometric Effects (Geometric non-linearity)
  - Material Effects (Material non-linearity)
  - Both

Elasticity and Linearity









Physical Object Based Modeling, Analysis and Design

Continuum Vs Structure

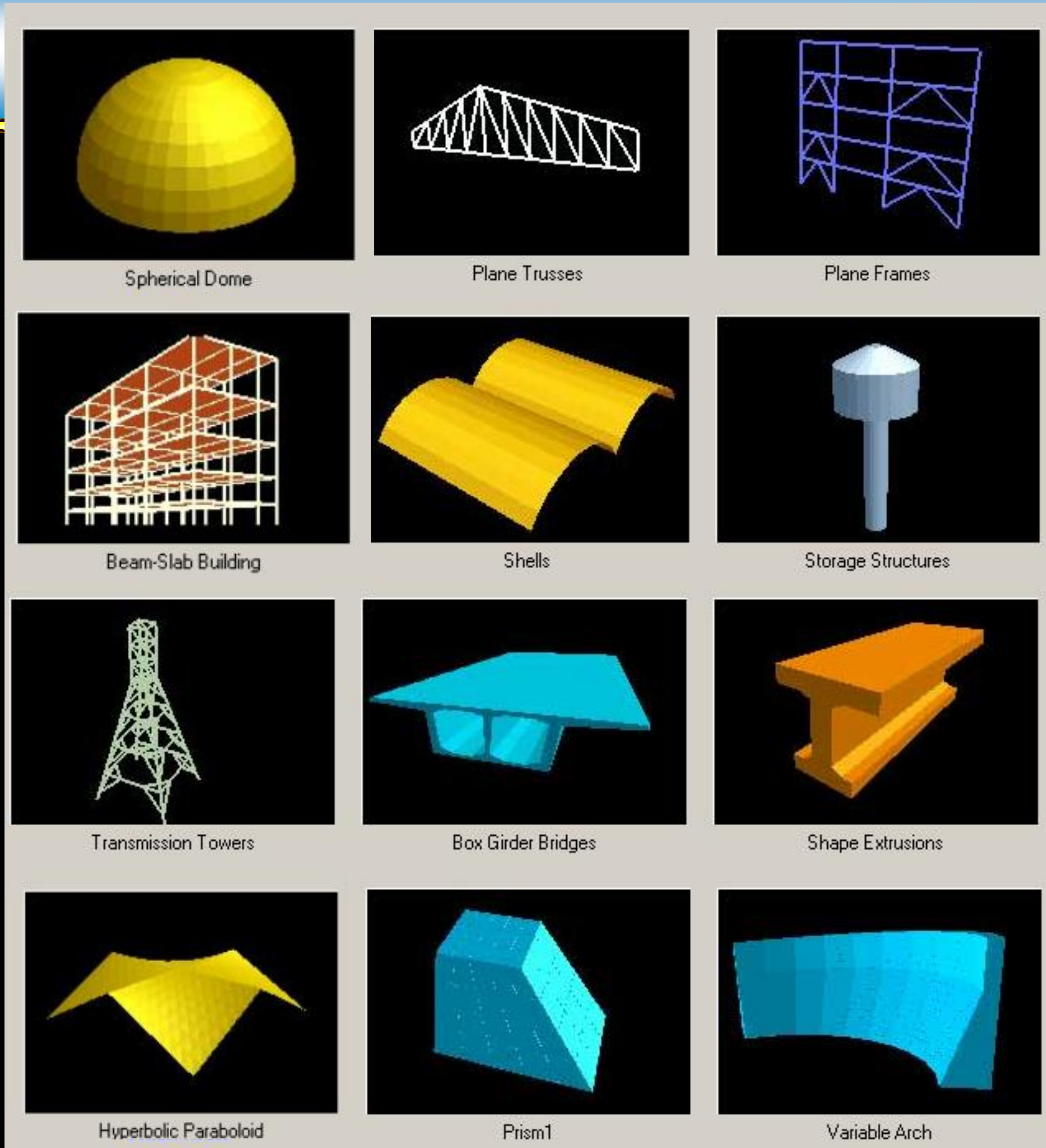
- A continuum extends in all direction, has infinite particles, with continuous variation of material properties, deformation characteristics and stress state
- A Structure is of finite size and is made up of an assemblage of substructures, components and members
- Dicretization process is used to convert Structure to Finite Element Models for determining response

Physical Categorization of Structures

- **Structures can be categorized in many ways.**
- **For modeling and analysis purposes, the overall physical behavior can be used as basis of categorization**

  - Cable or Tension Structures
  - Skeletal or Framed Structures
  - Surface or Spatial Structures
  - Solid Structures
  - Mixed Structures

Structure Types

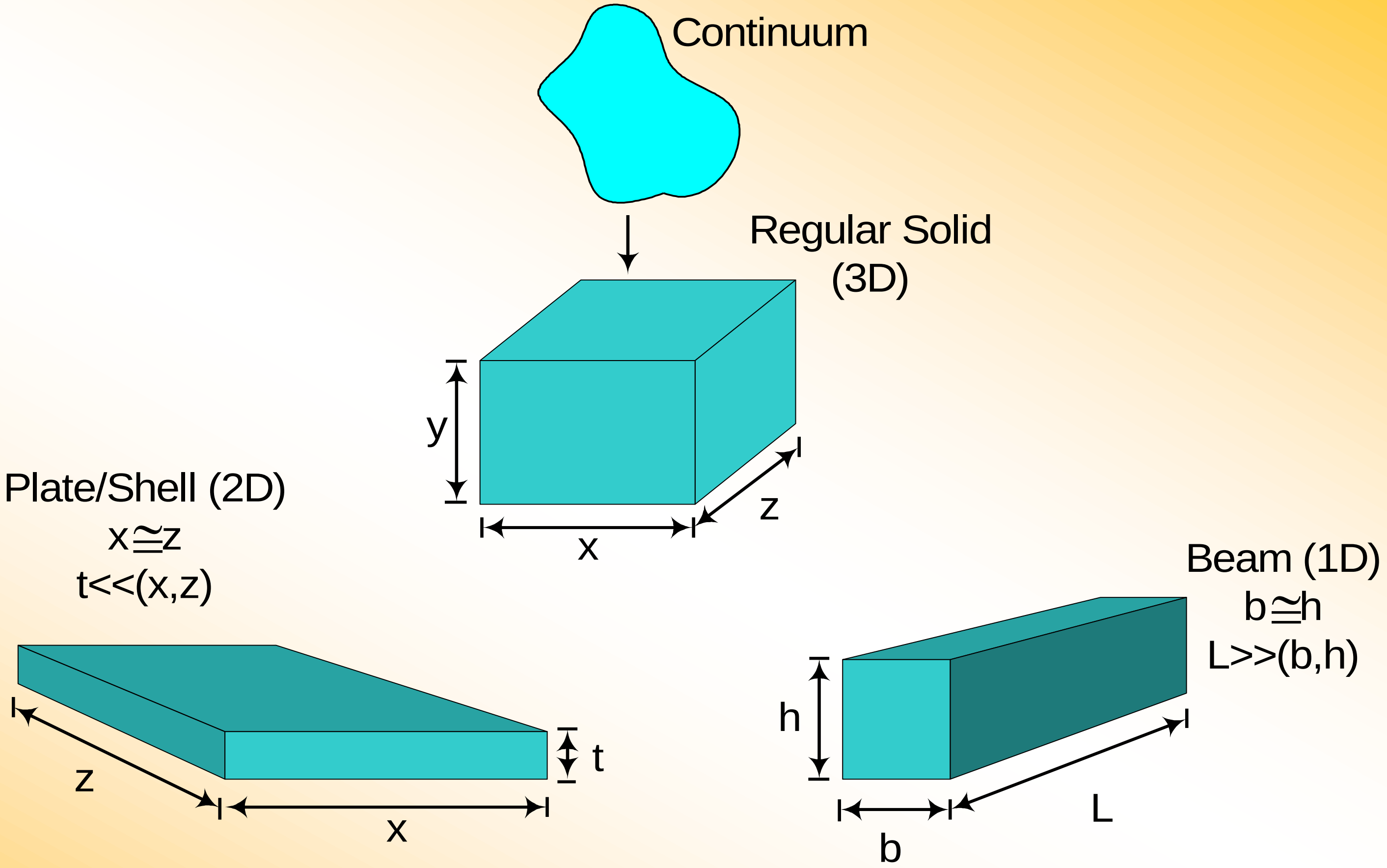


- Cable Structures
  - Cable Nets
  - Cable Stayed
- Bar Structures
  - 2D/3D Trusses
  - 2D/3D Frames, Grids
- Surface Structures
  - Plate, Shell
  - In-Plane, Plane Stress
- Solid Structures

Structure, Member, Element

- **Structure can 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, 2D element, 3D elements
  - Frame element, plate element, shell element, solid element, etc.
- **Modeling in terms Graphical Objects to represent Physical Components relieves the engineers from intricacies and idiosyncrasy of finite element discretization**

Structural Members



Dimensional Hierarchy of Structural Members

Load Transfer Path For Gravity Loads

- **Most loads are basically "Volume Loads" generated due to mass contained in a volume**

- **Mechanism and path must be found to transfer these loads to the "Supports" through a Medium**

- **All types of Static Loads can be represented as:**
  - Point Loads
  - Line Loads
  - Area Loads
  - Volume Loads

The Load Transfer Path

- **The Load is transferred through a medium which may be:**
  - A Point
  - A Line
  - An Area
  - A Volume
  - A system consisting of combination of several mediums

- **The supports may be represented as:**
  - Point Supports
  - Line Supports
  - Area Supports
  - Volume Supports

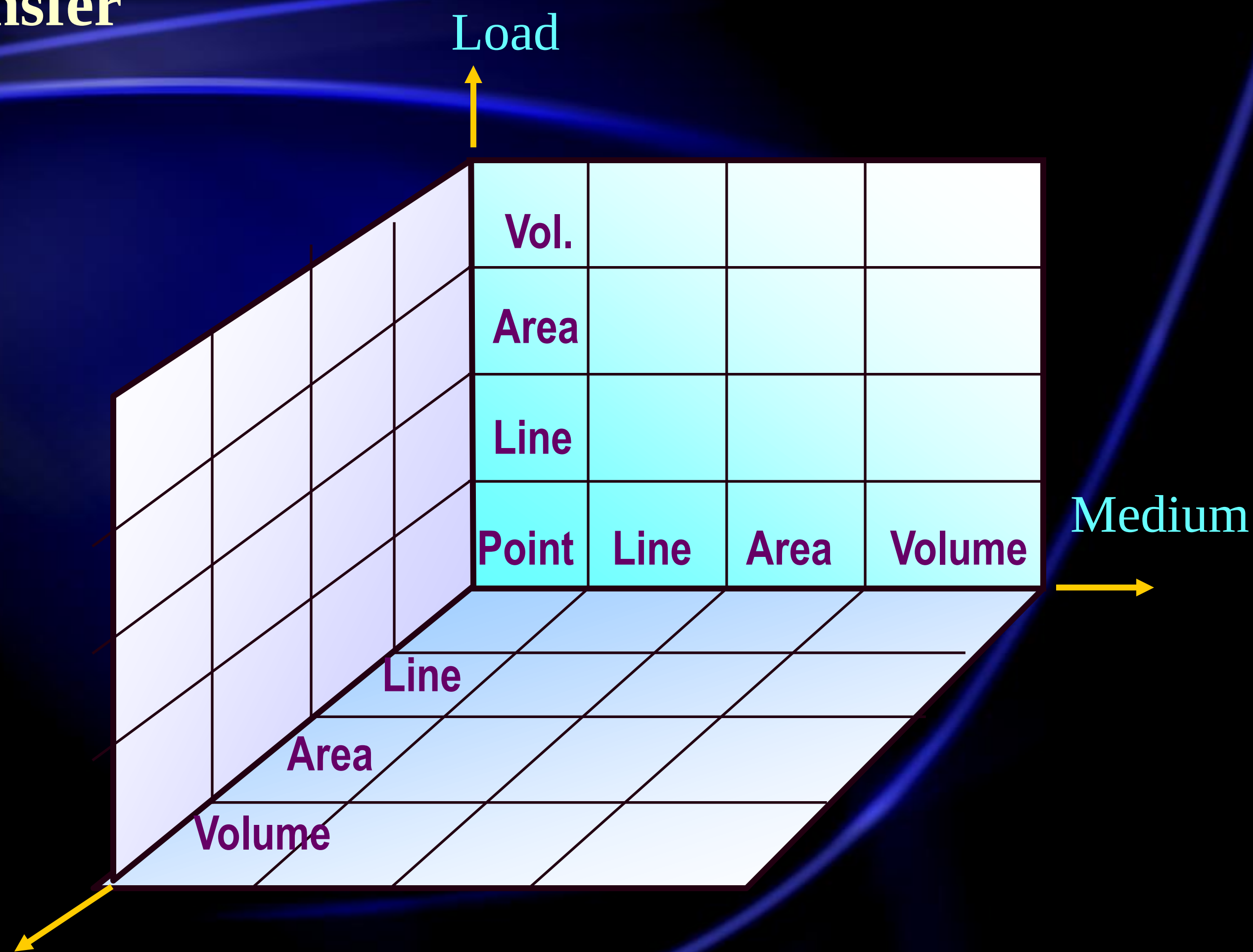
Graphic Object Representation

Object	Load	Geometry Medium	Support Boundary
Point	Point Load Concentrated Load	Node	Point Support Column Support
Line	Beam Load Wall Load Slab Load	Beam / Truss Connection Element Spring Element	Line Support Wall Support Beam Support
Area	Slab Load Wind Load	Plate Element Shell Element Panel/ Plane	Soil Support
Volume	Seismic Load Liquid Load	Solid Element	Soil Support

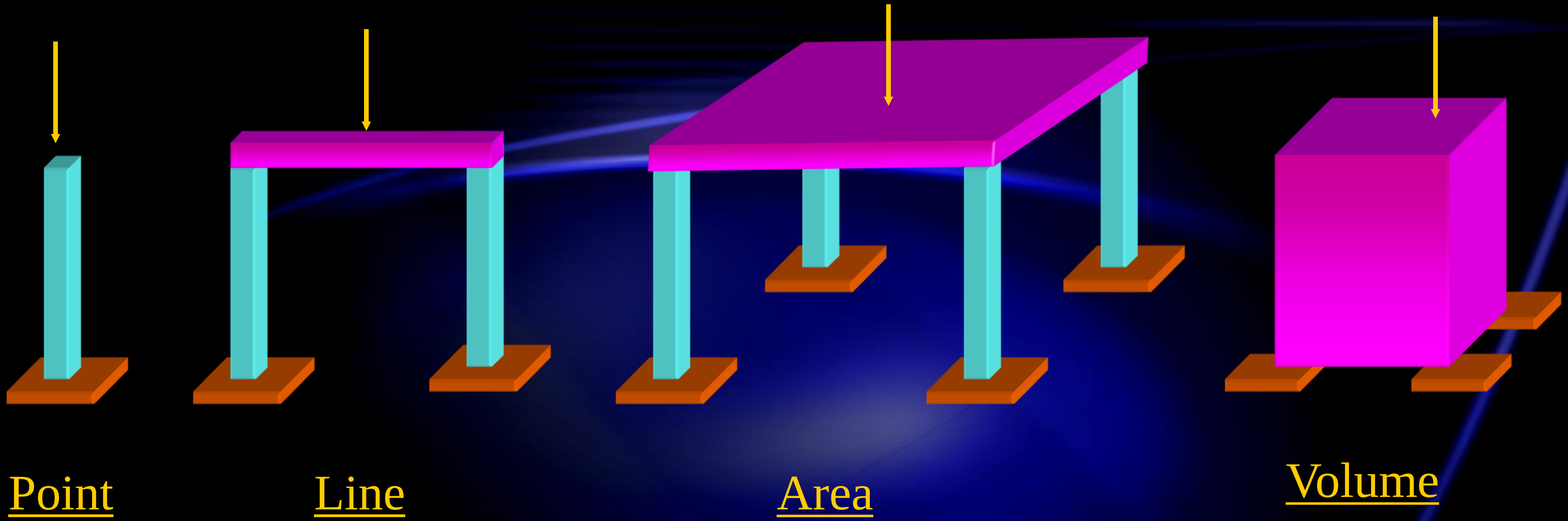
ETABS uses graphic object modeling concept

Load Transfer Path is difficult to Determine

- **Complexity of Load Transfer Mechanism depend on:**
  - Complexity of Load
  - Complexity of Medium
  - Complexity of Boundary



Load Transfer Path is difficult to Determine



Transfer of a Point Load to Point Supports Through Various Mediums

Objects in ETABS

- **Building Object Specific Classification**
  - Plank – One way slabs
  - Slab – One way or Two way slabs
  - Deck – Special one way slabs
  - Wall – Shear Walls, Deep Beams, In-Fill Panel
  - Frame – Column, Beam or Brace
- **Finite Elements**
  - Shell
  - Plate
  - Membrane
  - Beam
  - Node

Shell Element

General

- •Total DOF per Node = 6 (or 5)
- •Total Displacements per Node = 3
- •Total Rotations per Node = 3
- •Used for curved surfaces

Application

- •For Modeling surface elements carrying general loads

Building Specific Application

- •May be used for modeling of general slabs systems. But not used generally

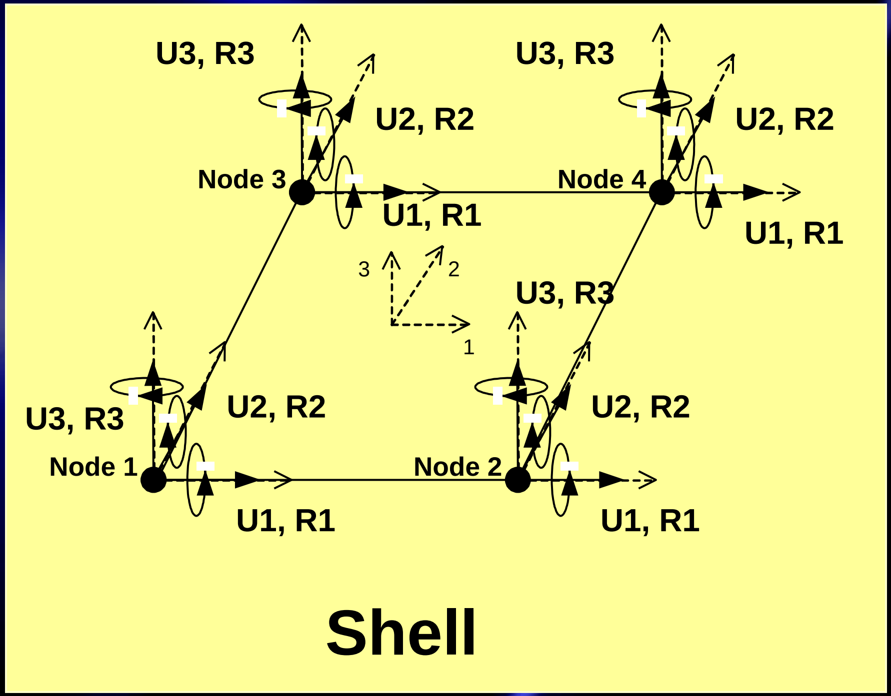


Plate Element

General

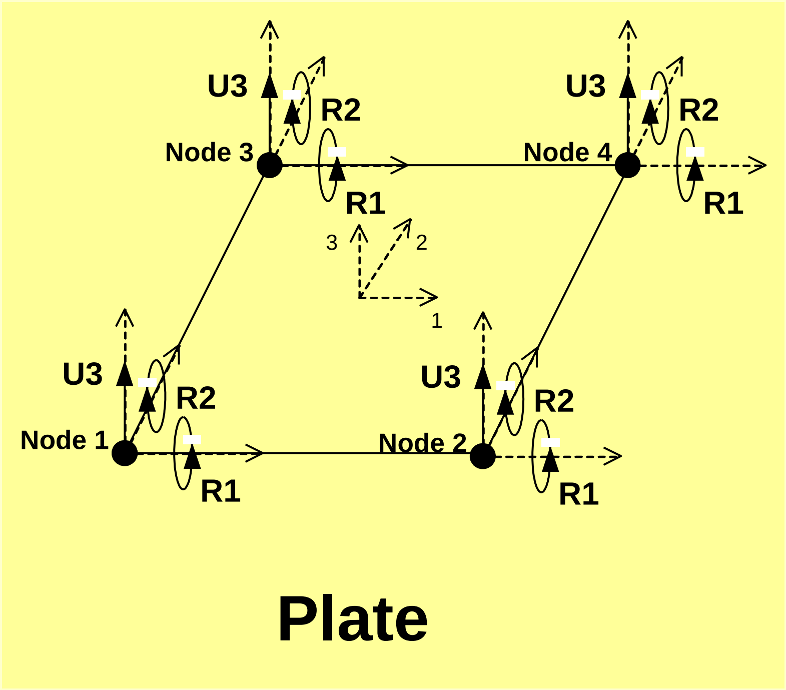
- Total DOF per Node = 3
- Total Displacements per Node = 1
- Total Rotations per Node = 2
- Plates are for flat surfaces

Application

- For Modeling surface elements carrying out of plane loads

Building Specific Application

- For representing floor slabs for Vertical Load Analysis
- Model slabs



Membrane Element

General

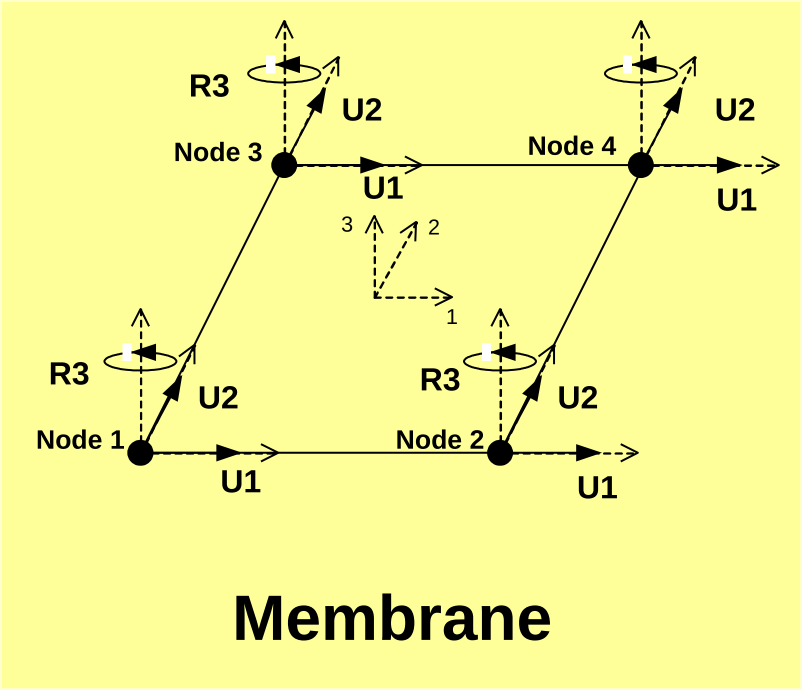
- Total DOF per Node = 3 (or 2)
- Total Displacements per Node = 2
- Total Rotations per Node = 1 (or 0)
- Membranes are modeled for flat surfaces

Application

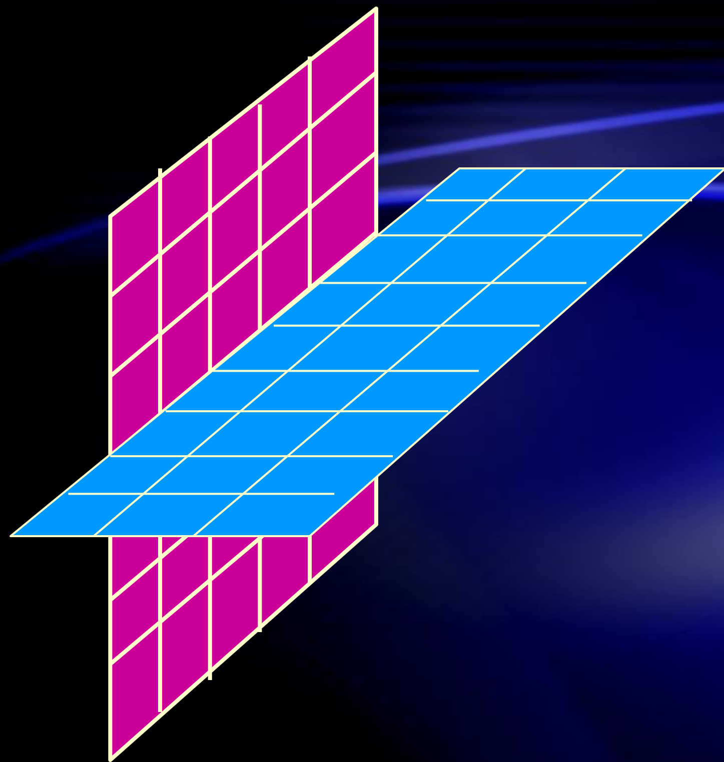
- For Modeling surface elements carrying in-plane loads

Building Specific Application

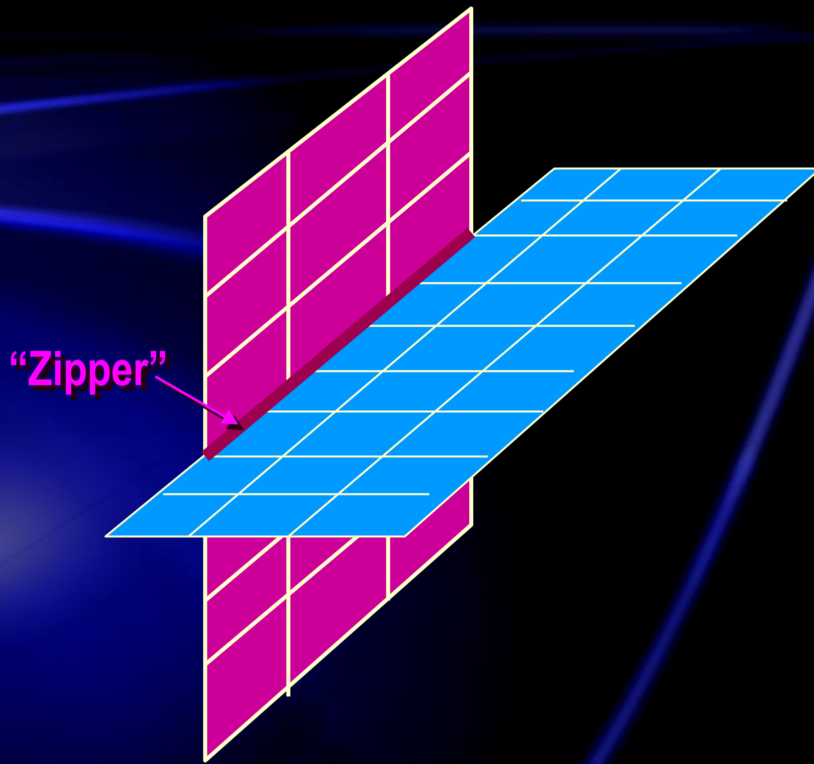
- For representing floor slabs for Lateral Load Analysis.
- Model Shear walls, Floor Diaphragm etc



Meshing Slabs and Walls



In general the mesh in the slab should match with mesh in the wall to establish connection



Some software automatically establishes connectivity by using constraints or "Zipper" elements