Credits: 3 + 0
CE – 842 - PG 2019
Spring 2020 Semester

Performance-based Seismic Design of Structures

Fawad A. Najam
Department of Structural Engineering
NUST Institute of Civil Engineering (NICE)
National University of Sciences and Technology (NUST)
H-12 Islamabad, Pakistan
Cell: 92-334-5192533, Email: fawad@nice.nust.edu.pk

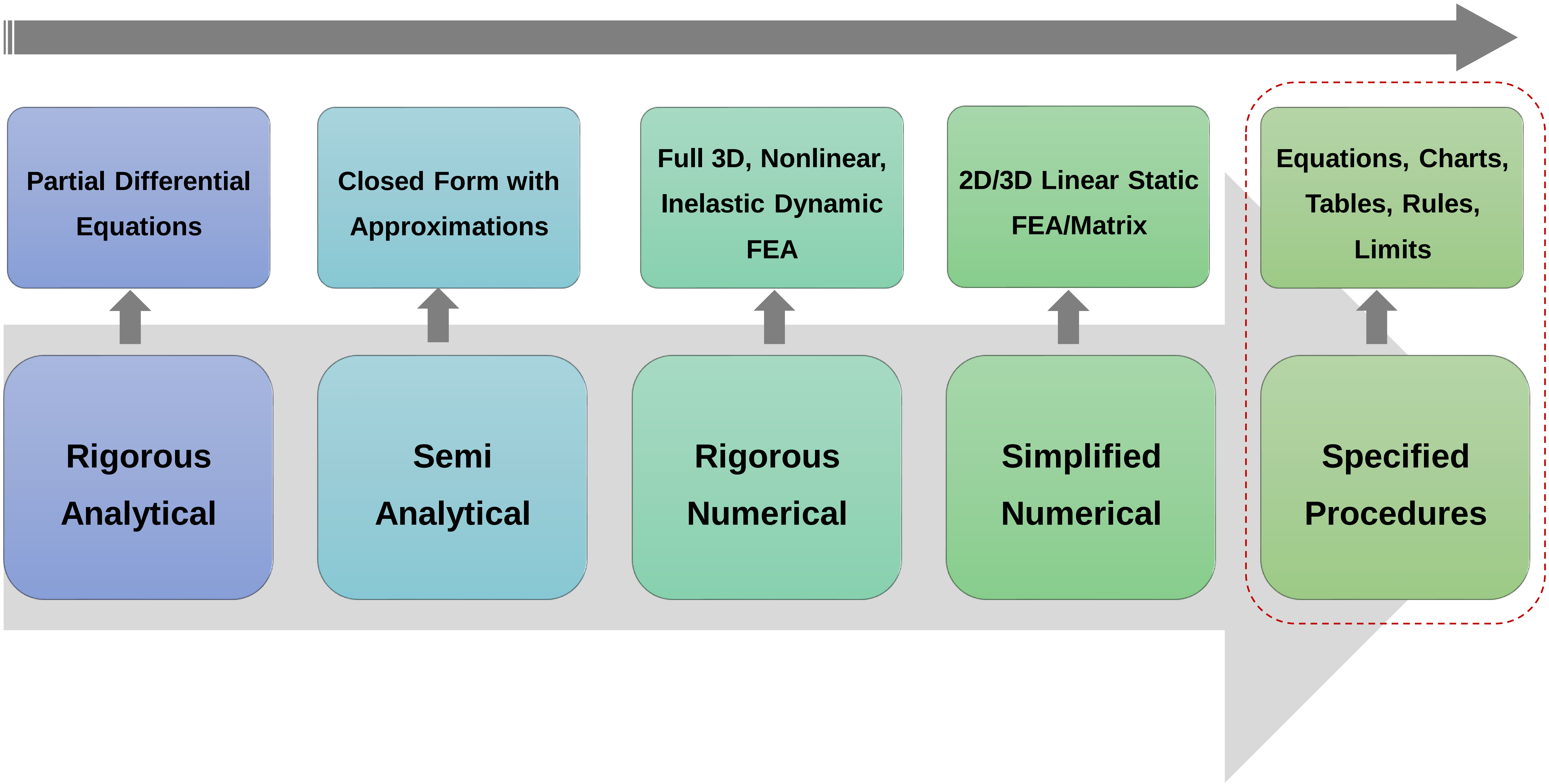
Acknowledgement

Dr. Naveed Anwar

- The material for the preparation of these lectures slides are taken from different sources.
- The primary source for these lecture slides are the lectures of Dr. Naveed Anwar at Asian Institute of Technology (AIT), Thailand
- Some other references of this training material include the following.
    - Class Notes of Prof. Dr. Worsak Kanok-Nukulchai at Asian Institute of Technology (AIT), Thailand
    - Training material developed by Mr. Thaung Htut Aung at AIT Solutions, AIT, Thailand
    - Notes from various workshops conducted by Dr. Naveed Anwar
    - Various International Codes and Guidelines
    - Lectures of Dr. Punchet Thammarak at Asian Institute of Technology (AIT), Thailand
    - Seminar notes from Computers and Structures Incorporated, USA
    - SAP2000 User and Technical Manuals
    - ETABS User and Technical Manuals
- The material is taken solely for educational purposes. **All sources are duly acknowledged.**

Lecture 4(b): Finite Element Modelling for Linear Elastic Analysis of Structures

- The Fundamental Principles of FE Analysis
- Structural Idealization
- Structural Discretization – The Element Menu
    - Joint Elements
    - One Dimensional Elements
    - Two Dimensional Elements
    - Three Dimensional Elements
- Modelling of Structural Materials
- Load Patterns, Load Cases and Load Combinations
- Post-processing and Results



Partial Differential Equations

Rigorous Analytical

Closed Form with Approximations

Semi Analytical

Full 3D, Nonlinear, Inelastic Dynamic FEA

Rigorous Numerical

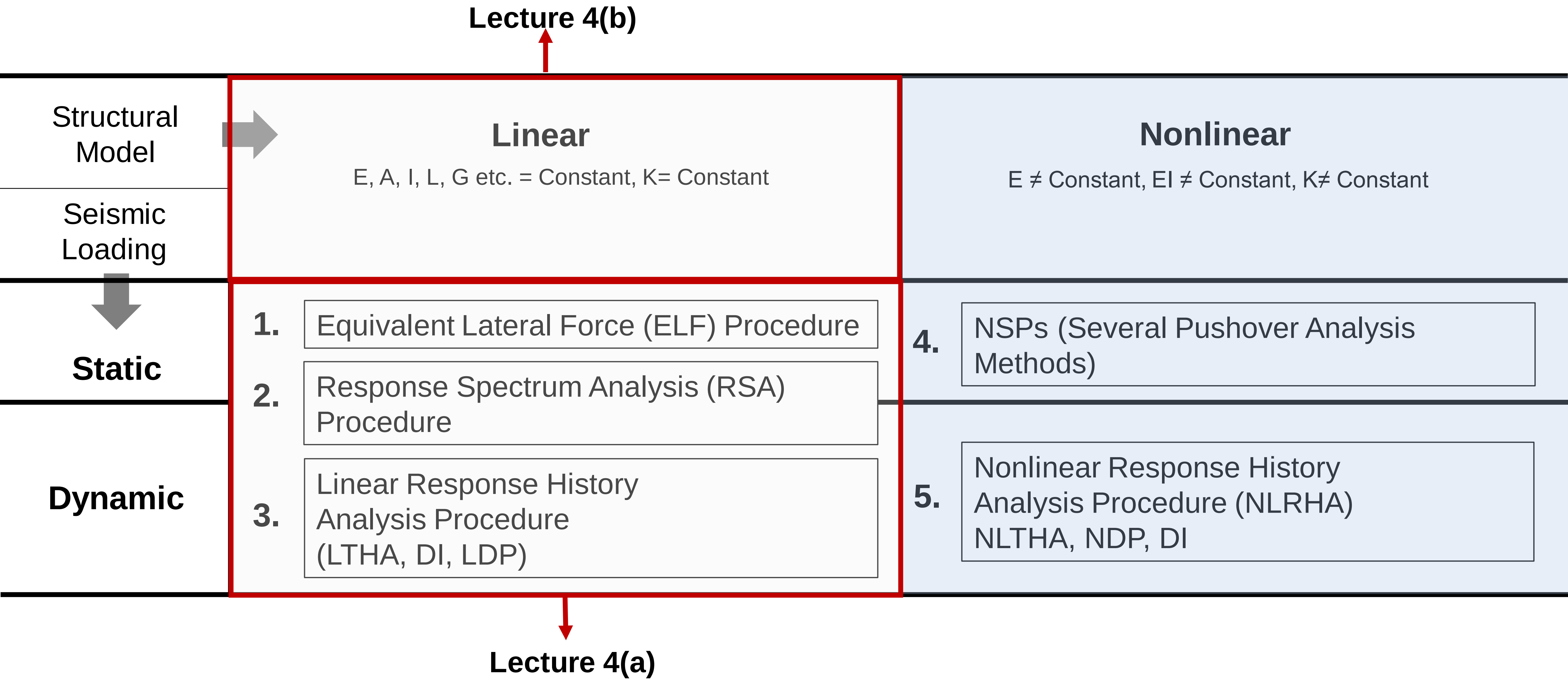
2D/3D Linear Static FEA/Matrix

Simplified Numerical

Equations, Charts, Tables, Rules, Limits

Specified Procedures

Seismic Analysis Procedures



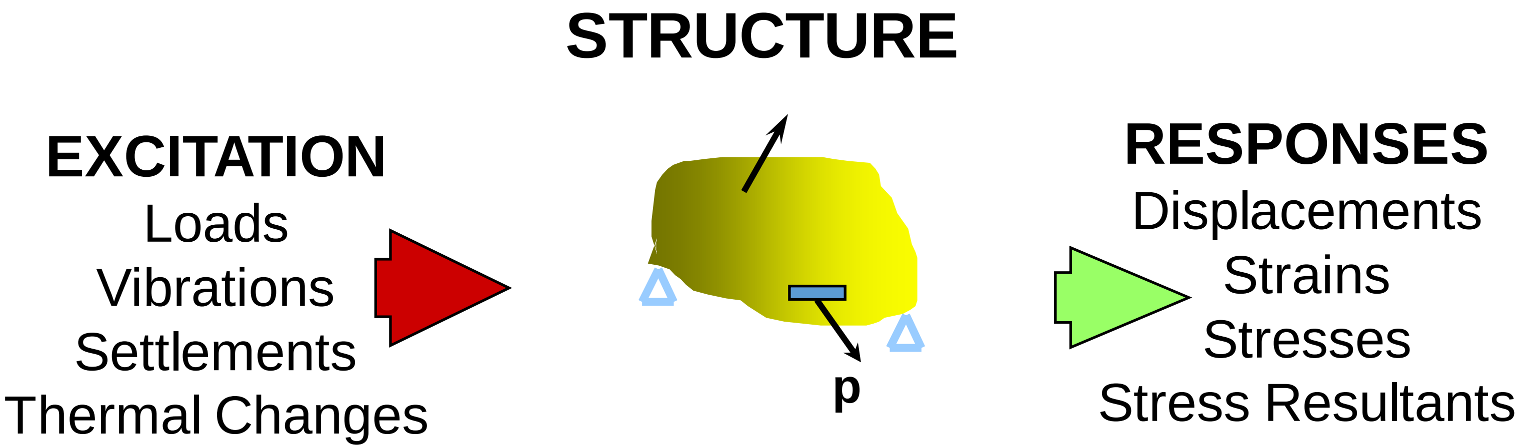
Lecture 4(b)

Structural Model Seismic Loading	**Linear** E, A, I, L, G etc. = Constant, K= Constant	**Nonlinear** E ≠ Constant, EI ≠ Constant, K≠ Constant
Static	**1.** Equivalent Lateral Force (ELF) Procedure **2.** Response Spectrum Analysis (RSA) Procedure	**4.** NSPs (Several Pushover Analysis Methods)
Dynamic	**3.** Linear Response History Analysis Procedure (LTHA, DI, LDP)	**5.** Nonlinear Response History Analysis Procedure (NLRHA) NLTHA, NDP, DI

Lecture 4(a)

Fundamental Principles of FE Analysis

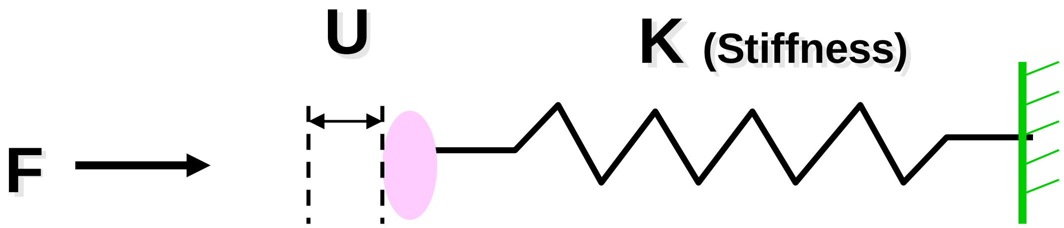
The Structural System



STRUCTURE

EXCITATION
Loads
Vibrations
Settlements
Thermal Changes

RESPONSES
Displacements
Strains
Stresses
Stress Resultants



Equilibrium Equation → $$F = K\,U$$

The Need For Analysis

- We need to determine the Response of the Structure to Excitations

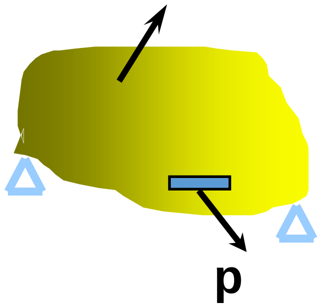
Analysis

so that:

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

Design

Analysis of Structures



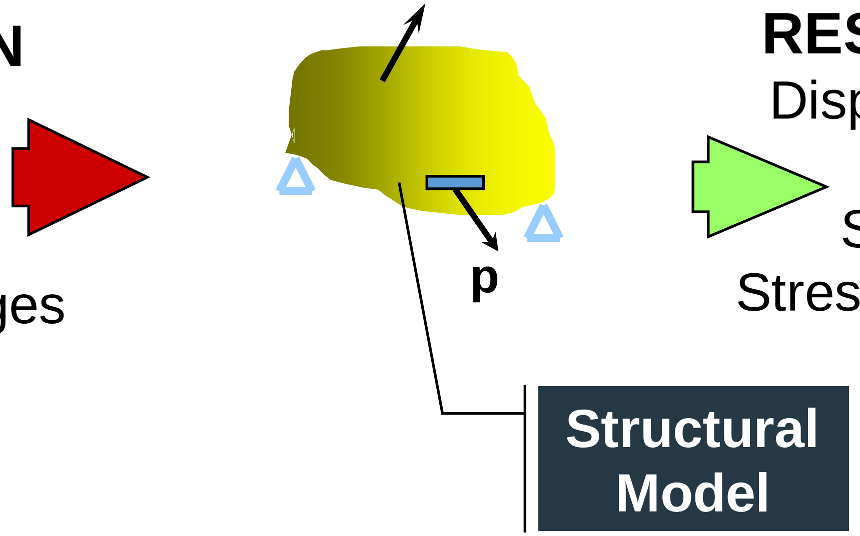
$$\overbrace{\frac{\partial \sigma_{xx}}{\partial x}+\frac{\partial \sigma_{yy}}{\partial y}+\frac{\partial \sigma_{zz}}{\partial z}+p_{vx}=0}^{\text{Equilibrium Equation: The Sum of Body Forces and Surface Tractions is equal to Zero}}$$

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 Structural Model



STRUCTURE

EXCITATION

Loads

Vibrations

Settlements

Thermal Changes

RESPONSES

Displacements

Strains

Stresses

Stress Resultants

Structural Model

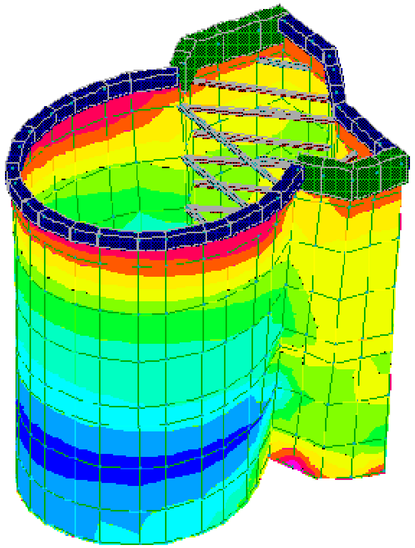
The Need for Modeling

- 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

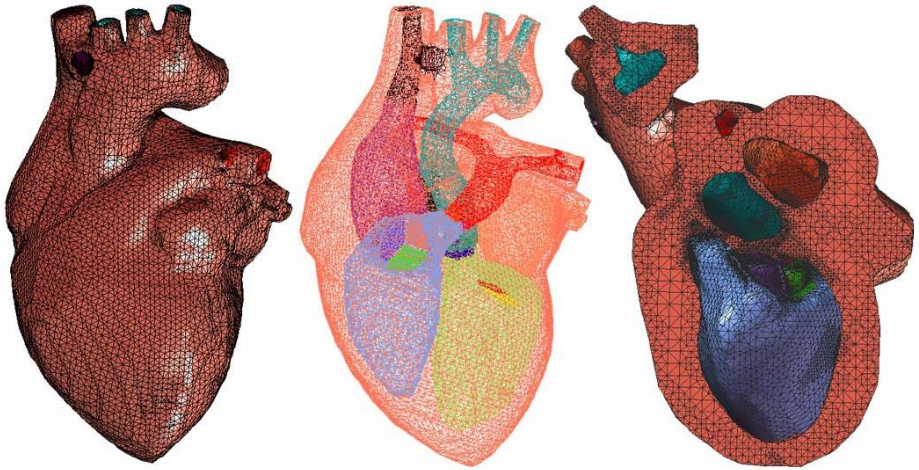
- C - We therefore need tools to <u>Model the Structure</u> and to <u>Analyze the Model</u>

Finite Element Method: The Analysis Tool



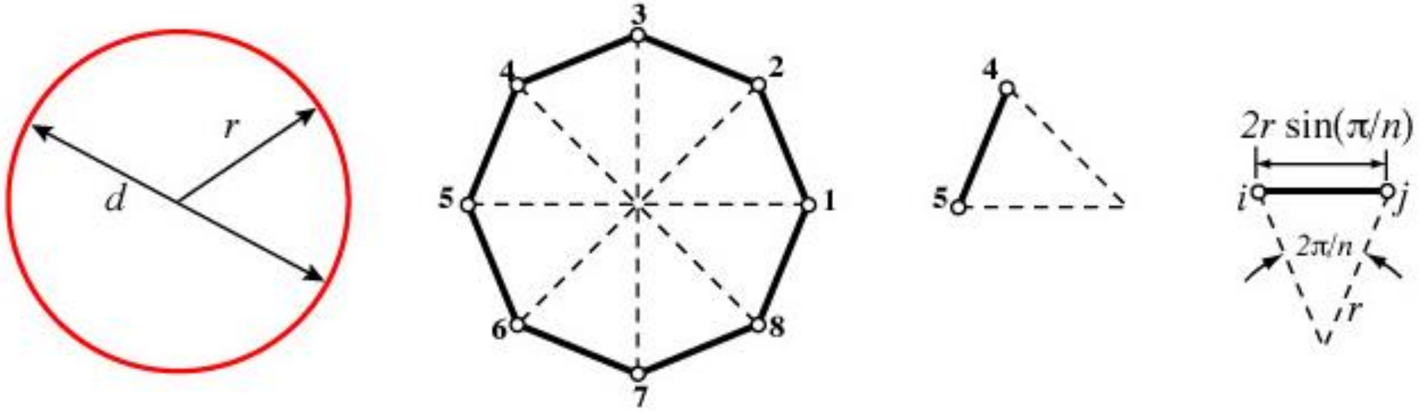
- Finite Element Analysis (FEA)
  - “A discretized solution to a continuum problem using FEM”

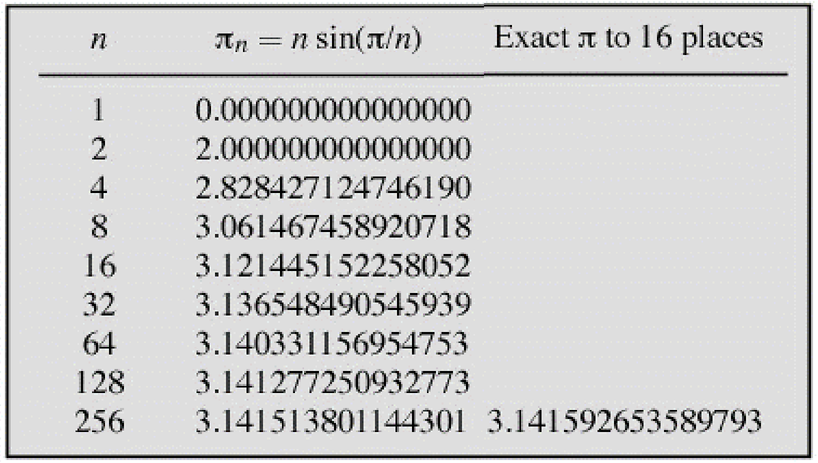
- Finite Element Method (FEM)
  - “A numerical procedure for solving (partial) differential equations associated with field problems, with an accuracy acceptable to engineers”



A Brief History - Archimedes FEM

Archimedes’ problem (*circa* 250 B.C.): rectification of the circle as limit of inscribed regular polygons





n	$\pi_n = n \sin(\pi/n)$	Exact π to 16 places
1	0.000000000000000	
2	2.000000000000000	
4	2.828427124746190	
8	3.061467458920718	
16	3.121445152258052	
32	3.136548490545939	
64	3.140331156954753	
128	3.141277250932773	
256	3.141513801144301	3.141592653589793

Computing π "by Archimedes FEM"

Brief History

- Grew out of aerospace industry
- Post-WW II jets, missiles, space flight
- Need for **light weight** structures
- Required **accurate stress analysis**
- Paralleled **growth of computers**

Developments

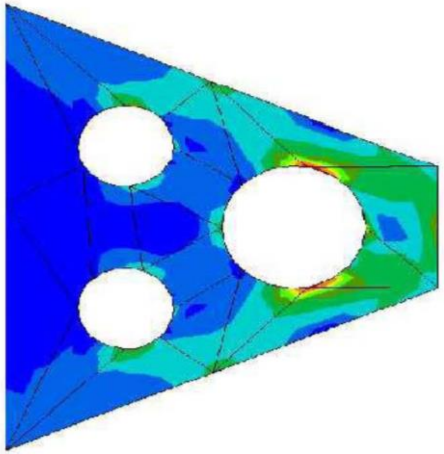
- 1940s
  - Hrennikoff (1941) - Lattice of 1D bars
  - McHenry (1943) - Model 3D solids
  - Courant (1943) - Variational form
  - Levy (1947, 1953) - Flexibility and Stiffness
- 1950-60s
  - Argryis and Kelsey (1954) - Energy Principle for Matrix Methods
  - Turner, Clough, Martin and Topp (1956) - 2D elements
  - Clough (1960) - Term “Finite Elements”
- 1980s - Wide applications due to:
  - Integration of CA/CAE – automated mesh generation and graphical display of analysis results
  - Powerful and low-cost computers
- 2000s – FEA in CAD; Design Optimization in FEA; Nonlinear FEA; Better CAD/CAE Integration

Current Applications

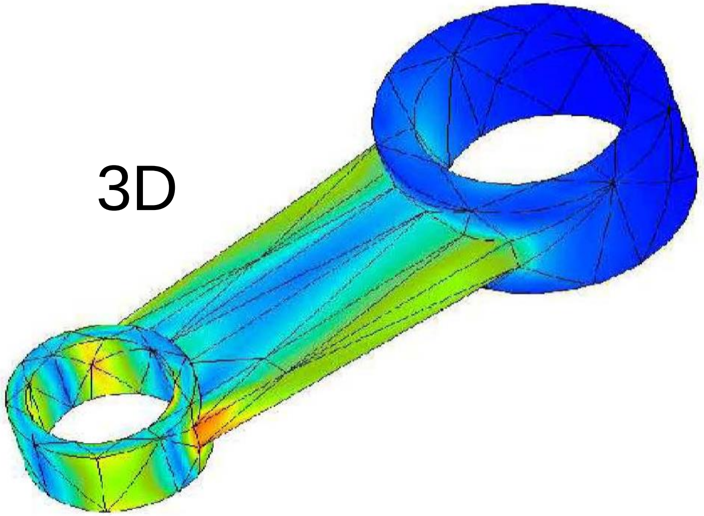
- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress Analysis
  - Static/Dynamic
  - Linear/Nonlinear
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

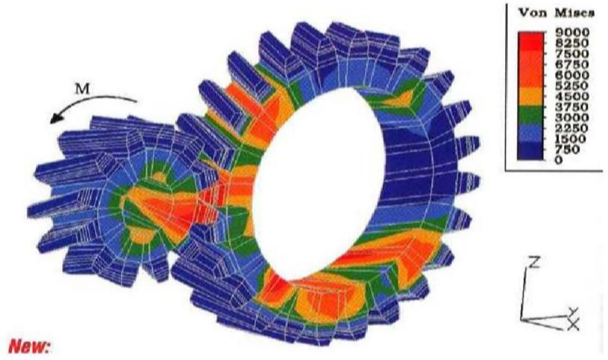
Finite Element Modelling - Examples

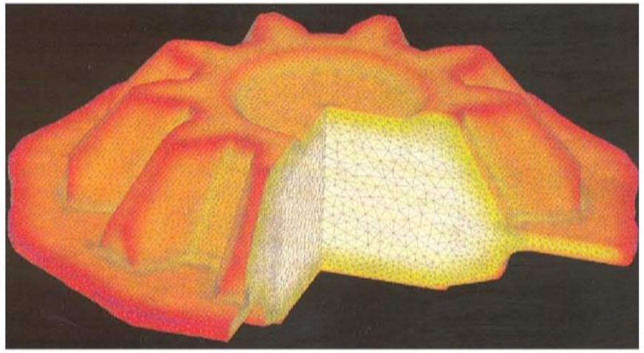
2D



3D



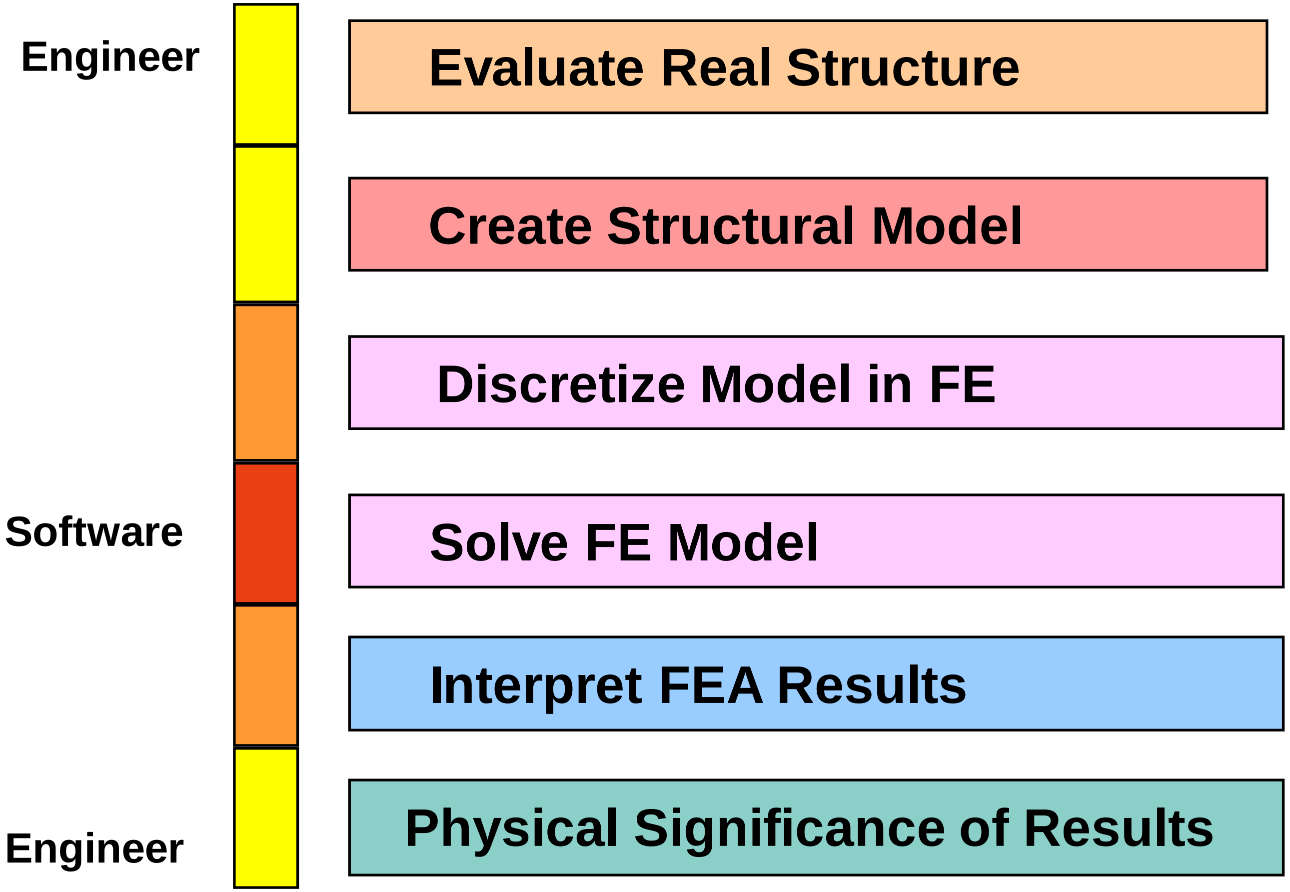




FEA Overall Process

- **Prepare the FE Model**
  - Discretize (mesh) the structure
  - Prescribe loads
  - Prescribe supports
- **Perform calculations (solve)**
  - Generate stiffness matric (k) for each element
  - Connect elements and assemble K
  - Assemble loads (into load vector F)
  - Impose supports conditions
  - Solve equations (KU = F) for displacements
- **Post-processing (stress recovery etc.)**

The Finite Element Analysis Process



Engineer

Software

Engineer

Evaluate Real Structure

Create Structural Model

Discretize Model in FE

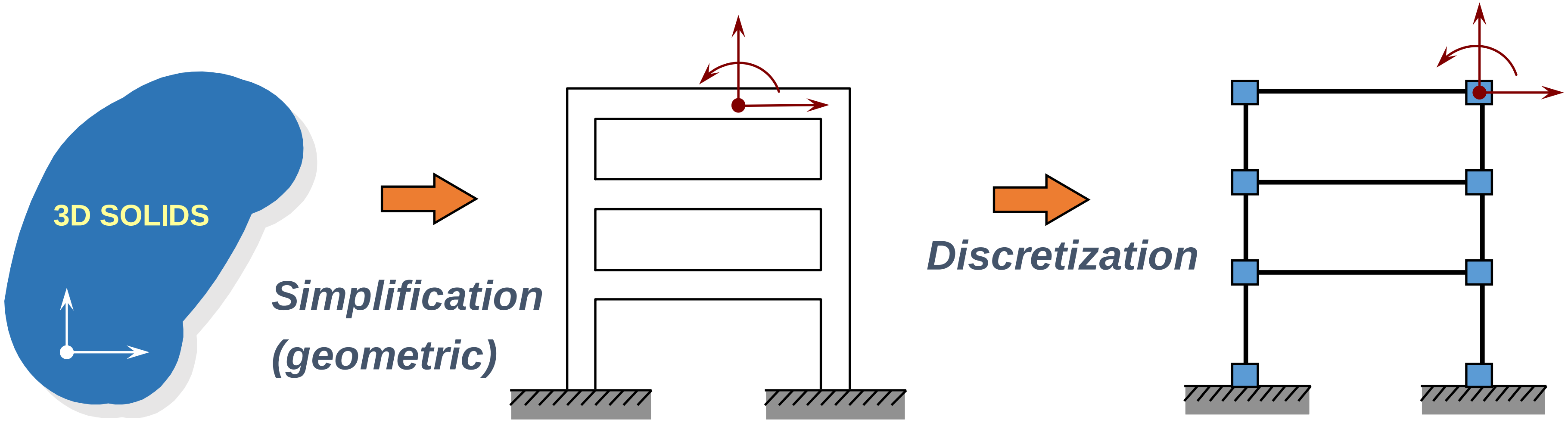
Solve FE Model

Interpret FEA Results

Physical Significance of Results

Structural Idealization

Solid – Structure - Model



Simplification (geometric)

Discretization

3D-CONTINUUM MODEL
(Governed by partial differential equations)

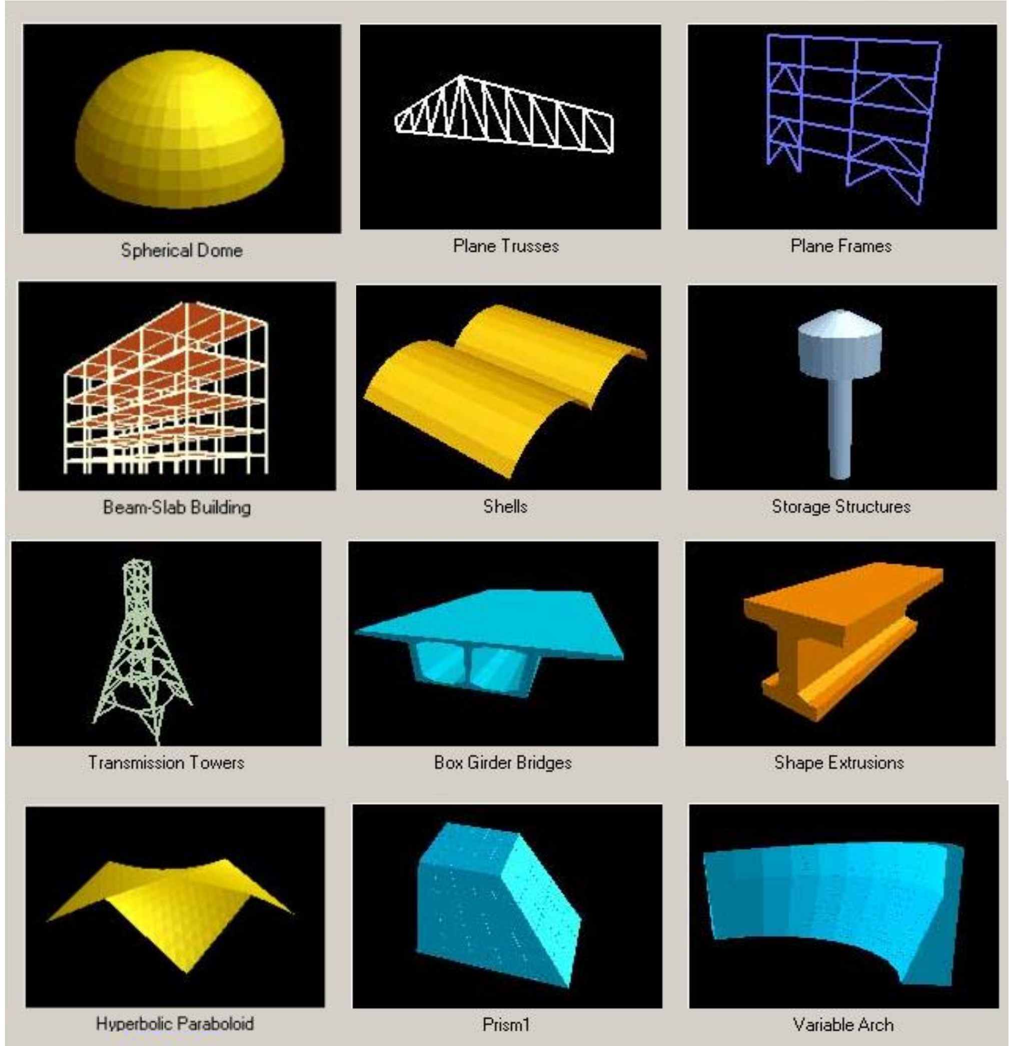
CONTINUOUS MODEL OF STRUCTURE
(Governed by either partial or ordinary differential equations)

DISCRETE MODEL OF STRUCTURE
(Governed by algebraic equations)

Continuum vs. Structure

- A **continuum** extends in all direction, has **infinite** particles, with continuous variation of material properties, deformation characteristics and stress states.
- A **structure** is of **finite** size and is made up of an assemblage of substructures, components and members.
- 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



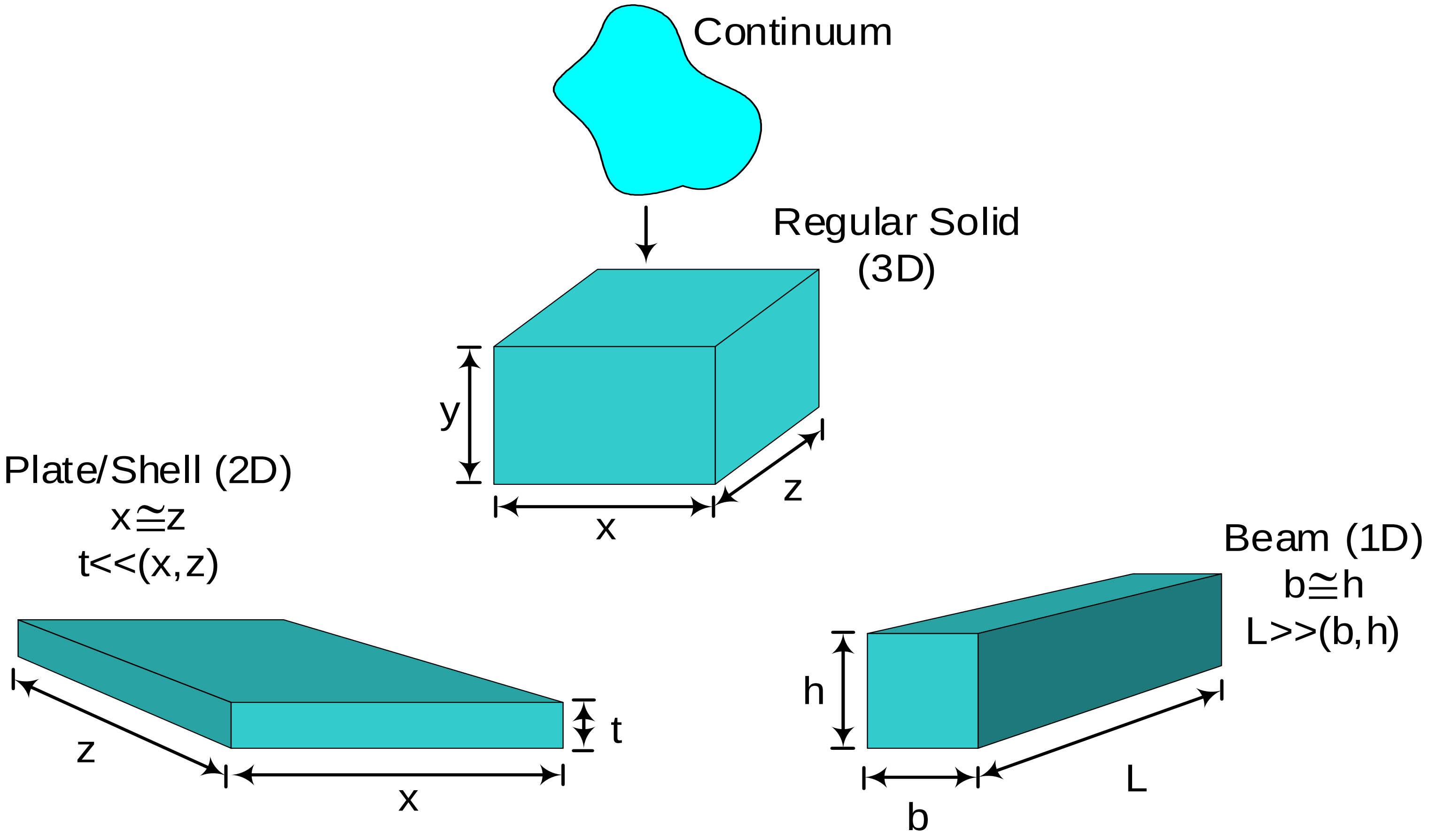
- Cable Structures
  - Cable Nets
  - Cable Stayed

- Bar Structures
  - 2D/3D Trusses
  - 2D/3D Frames, Grids

- Surface Structures
  - Plate, Shell
  - In-Plane, Plane Stress

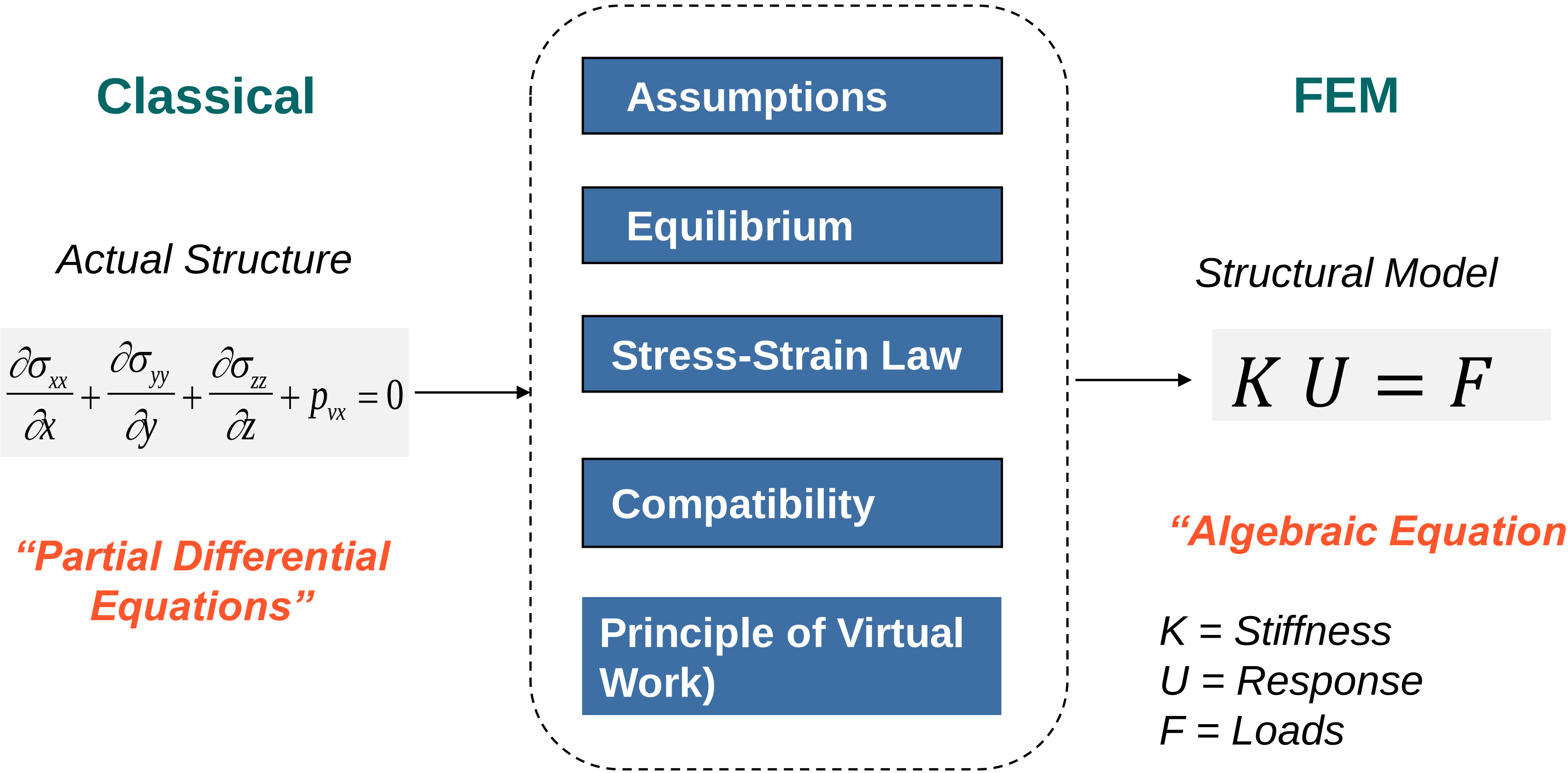
- Solid Structures

Structural Idealization



Dimensional Hierarchy of Structural Members

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

Principle of Virtual Work)

FEM

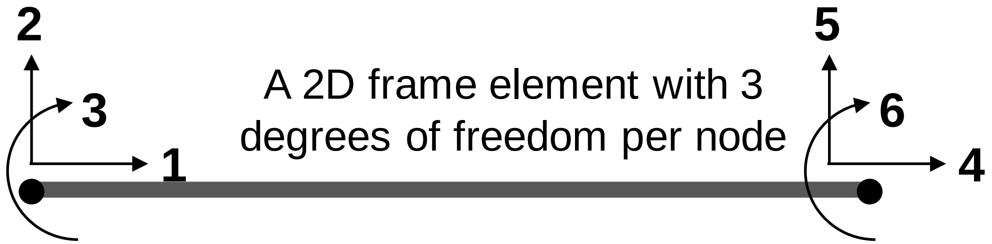
Structural Model

$$K\ U = F$$

"Algebraic Equations"

K = Stiffness
U = Response
F = Loads

What is Stiffness (K) “made off”?



A 2D frame element with 3 degrees of freedom per node

$$K = f(\underset{\swarrow}{E}, \underbrace{A, I}_{\downarrow}, \underset{\searrow}{L})$$

Material Property

Cross-section Properties

Member Property

$$K = \begin{pmatrix} EA/L & 0 & 0 & -EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 4EI/L & 0 & -6EI/L^2 & 2EI/L \\ -EA/L & 0 & 0 & EA/L & 0 & 0 \\ & -12EI/L^3 & -6EI/L^2 & 0 & 12EI/L^3 & -6EI/L^2 \\ 0 & 6EI/L^2 & 2EI/L & 0 & -6EI/L^2 & 4EI/L \end{pmatrix}$$

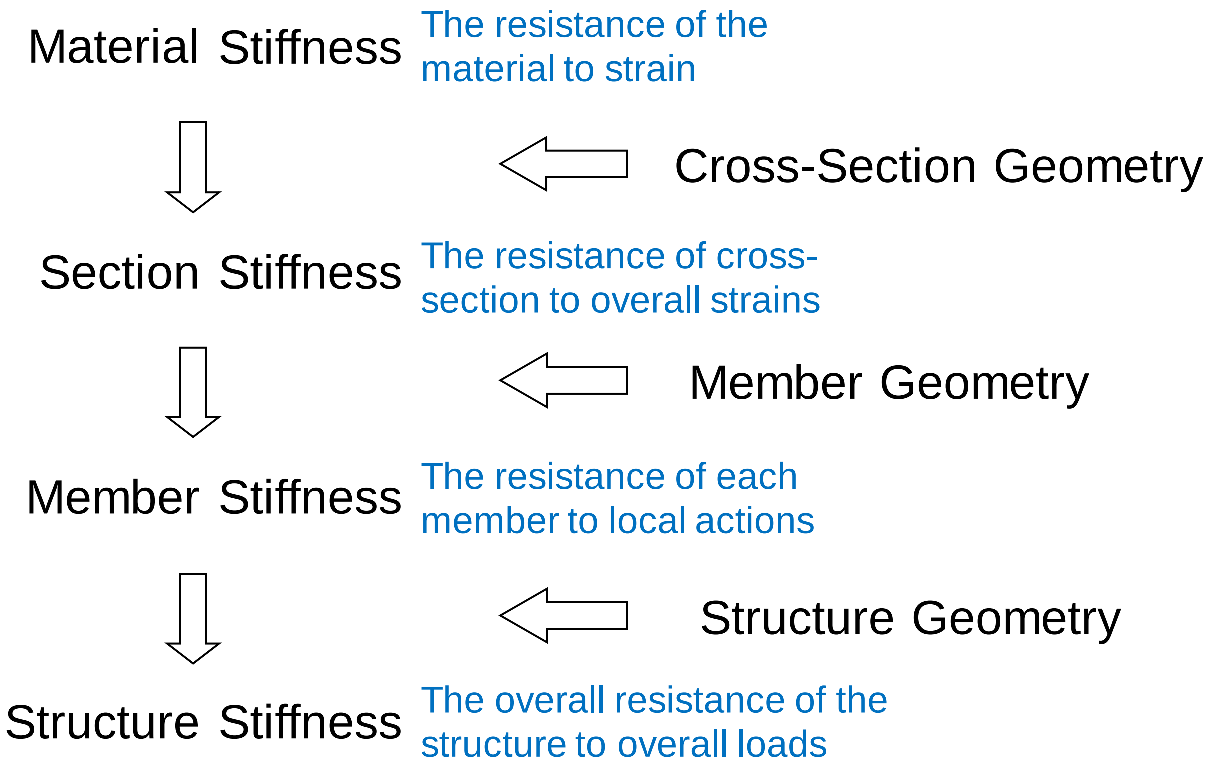
What is Stiffness (K) “made off”?

- The **overall stiffness** of the structure is derived from the overall geometry and connectivity of the members, their stiffnesses, and the boundary conditions.

- The **member stiffness** is derived from the **cross-section stiffness**, and member geometry.

- The **cross-section stiffness** is derived from the **material stiffness** and the cross-section geometry.

- All of these stiffness relationships may be linear or nonlinear.



Material Stiffness — The resistance of the material to strain

⇩ ⇦ Cross-Section Geometry

Section Stiffness — The resistance of cross-section to overall strains

⇩ ⇦ Member Geometry

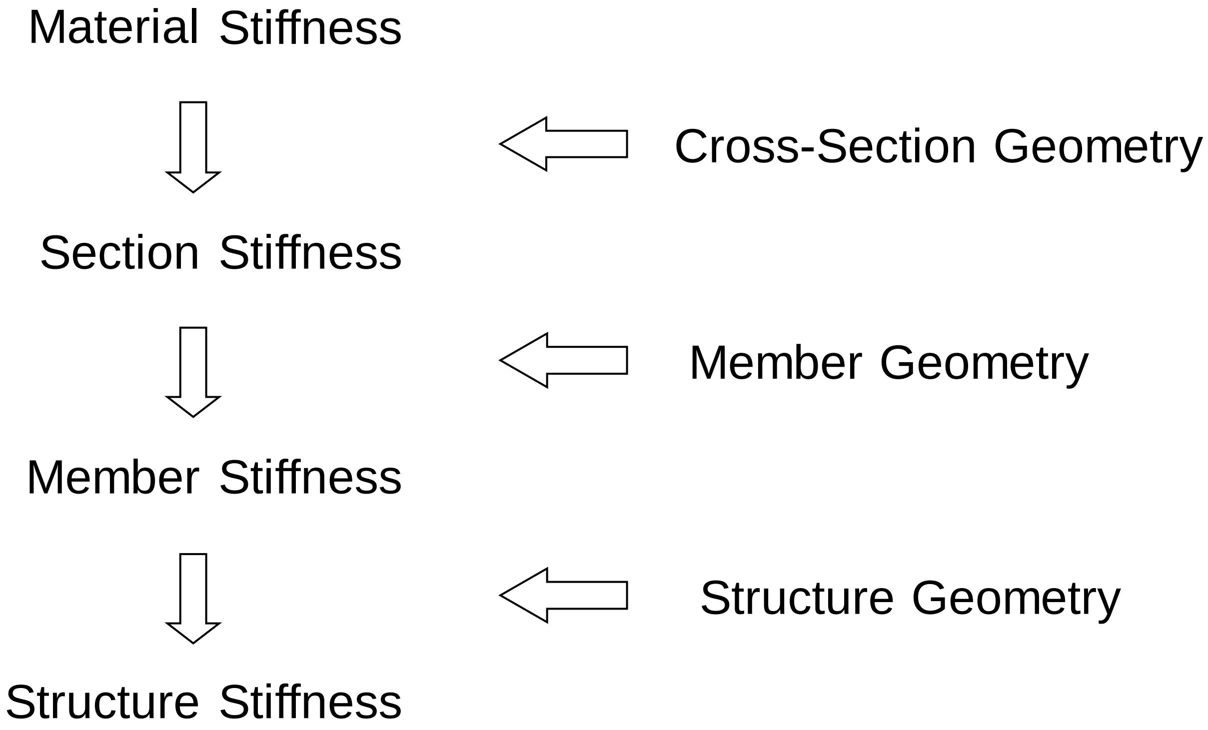
Member Stiffness — The resistance of each member to local actions

⇩ ⇦ Structure Geometry

Structure Stiffness — The overall resistance of the structure to overall loads

“Actual” Stiffness Estimation Influenced by

- The state of the structure at any given time
  - Damage
  - Deformation
  - Cracking
  - Creep/Shrinkage
  - Stress-state



Material Stiffness

Section Stiffness

Member Stiffness

Structure Stiffness

Cross-Section Geometry

Member Geometry

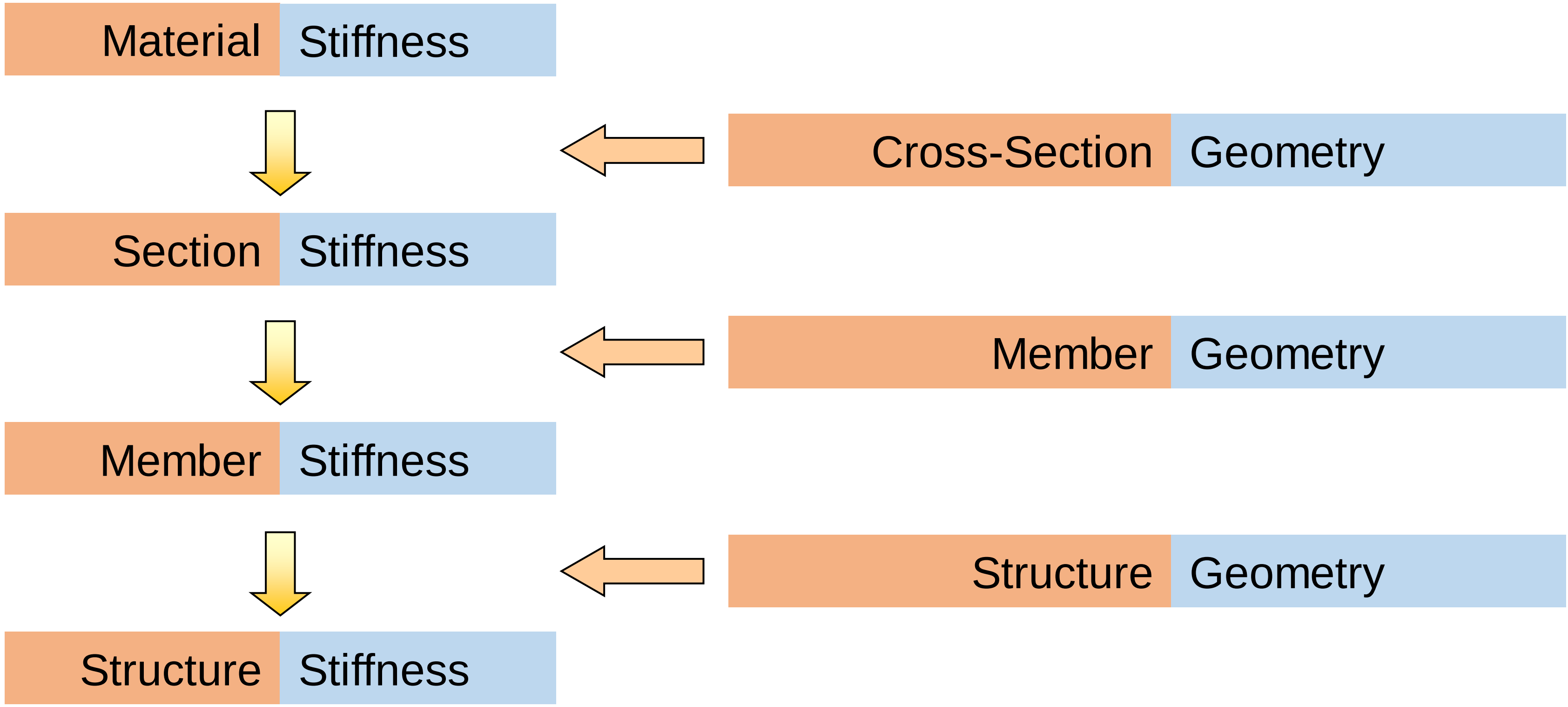
Structure Geometry

The Global Structure Stiffness - K

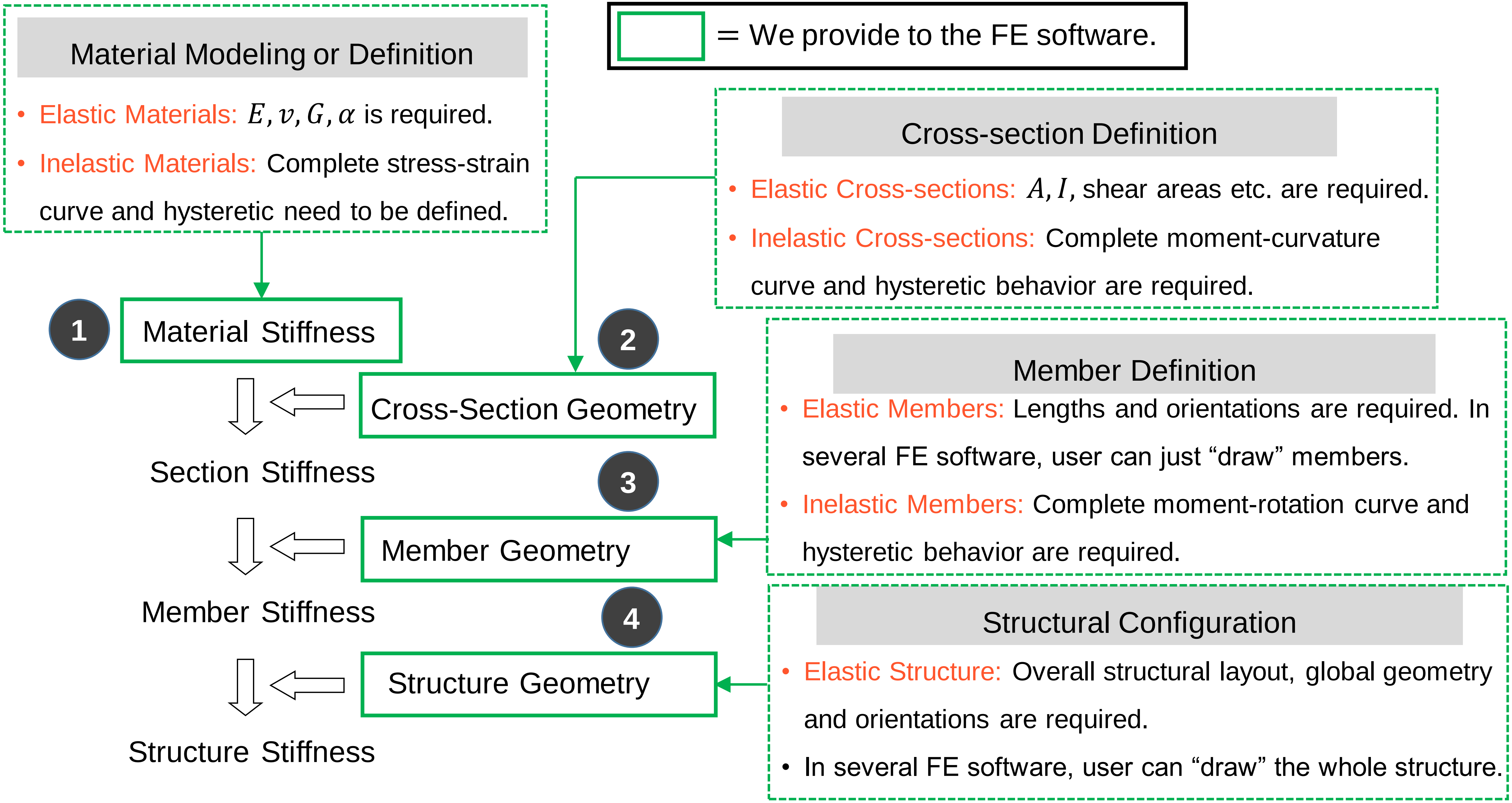
$$M\ddot{u} + C\dot{u} + Ku = F$$

🟦 Linear

🟧 Non-Linear



Nonlinear Modeling is a topic for Lecture 6 (b)



🟩 = We provide to the FE software.

Material Modeling or Definition

- Elastic Materials: E, v, G, α is required.
- Inelastic Materials: Complete stress-strain curve and hysteretic need to be defined.

Cross-section Definition

- Elastic Cross-sections: $A, I,$ shear areas etc. are required.
- Inelastic Cross-sections: Complete moment-curvature curve and hysteretic behavior are required.

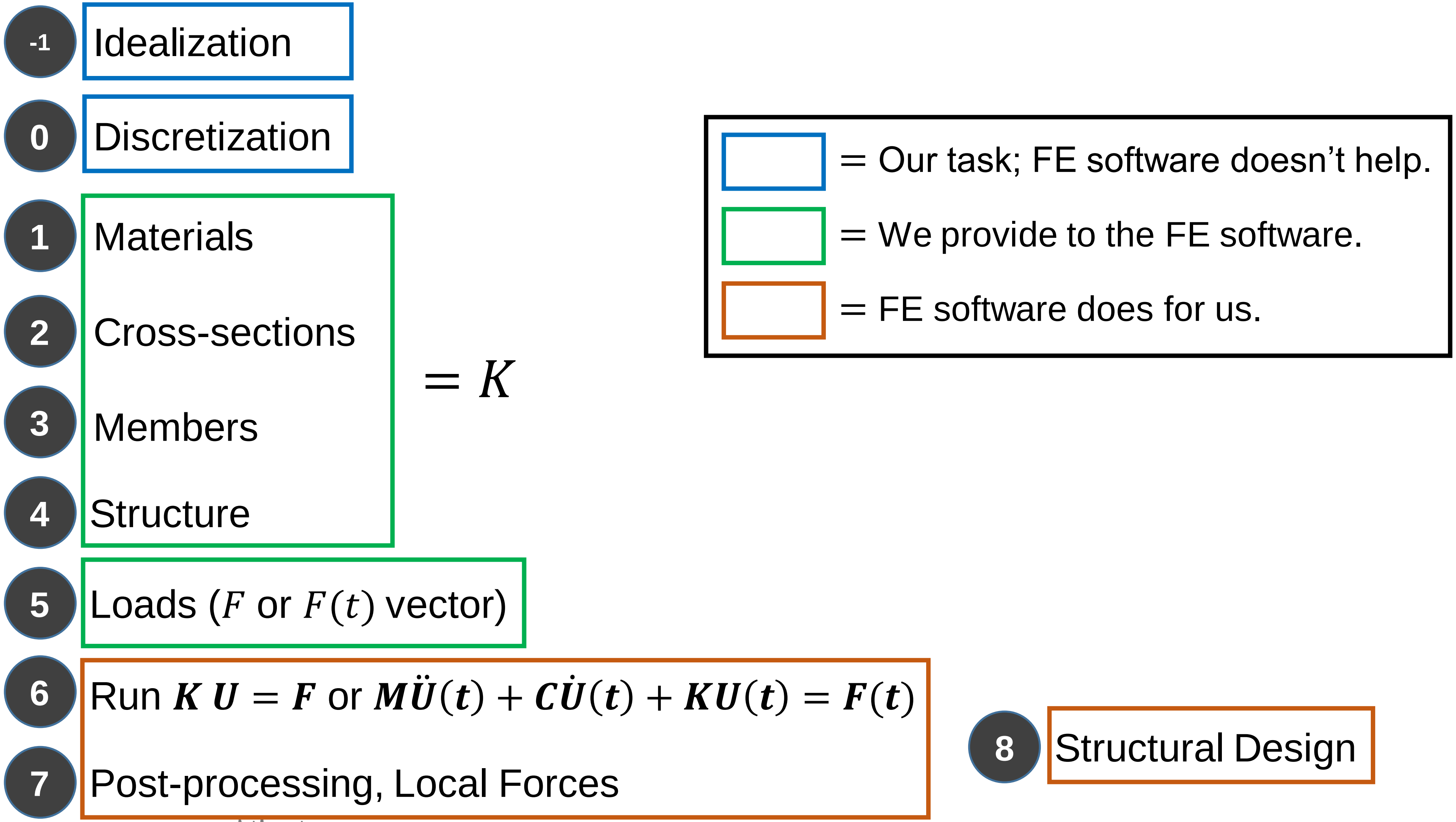
Member Definition

- Elastic Members: Lengths and orientations are required. In several FE software, user can just “draw” members.
- Inelastic Members: Complete moment-rotation curve and hysteretic behavior are required.

Structural Configuration

- Elastic Structure: Overall structural layout, global geometry and orientations are required.
- In several FE software, user can “draw” the whole structure.

Finite Element Modeling, Analysis and Design Process



-1 Idealization

0 Discretization

1 Materials

2 Cross-sections

3 Members

4 Structure

$= K$

5 Loads (F or $F(t)$ vector)

6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$

7 Post-processing, Local Forces

8 Structural Design

🟦 = Our task; FE software doesn't help.

🟩 = We provide to the FE software.

🟧 = FE software does for us.

Structure, Members, Elements

- Structure can be considered as an assemblage of “**Physical Components**” called Members.
  - Slabs, Beams, Columns, Footings, etc.

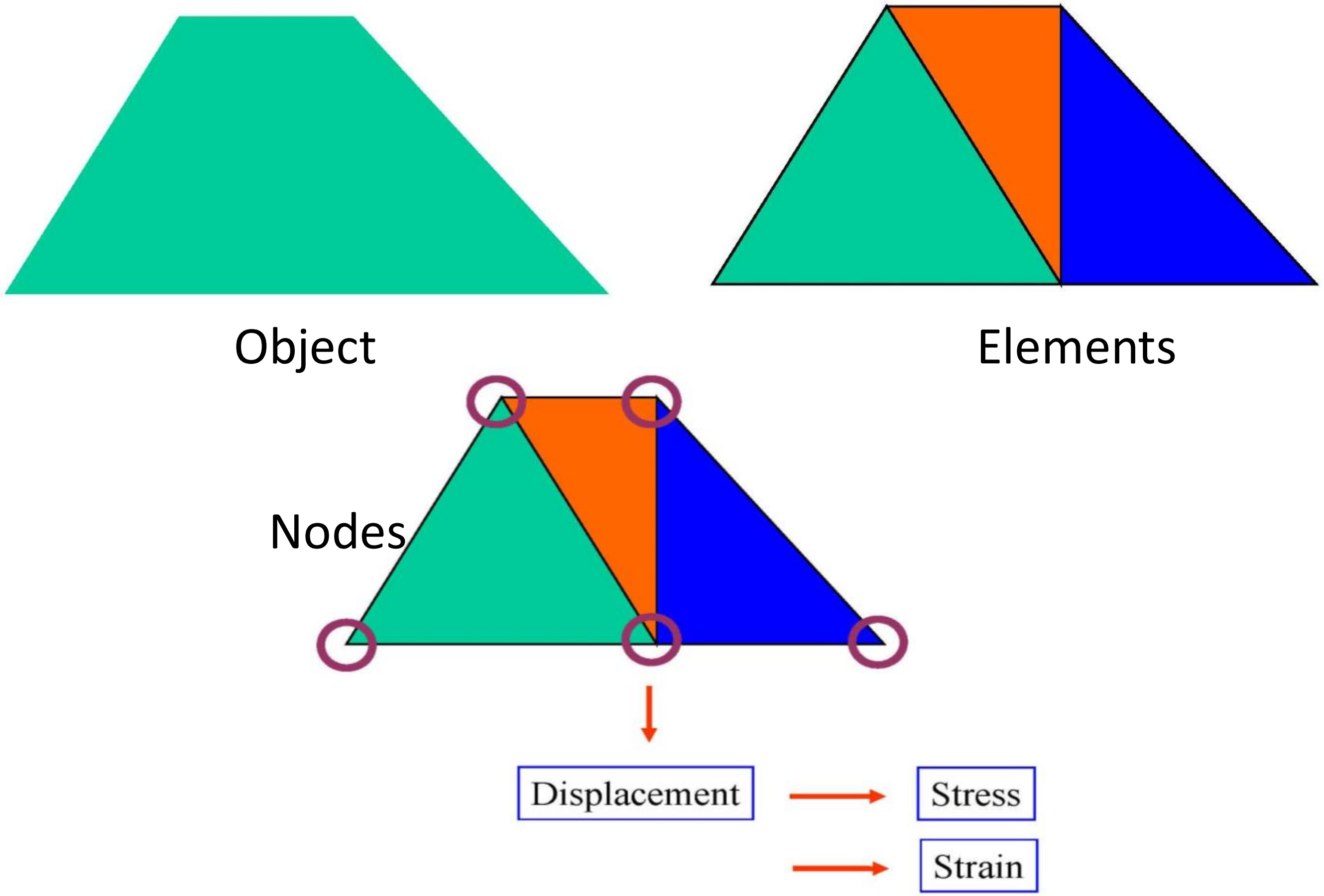
- Physical Members can be modeled by using one or more “**Conceptual Components**” called Elements.
  - 1D elements, 2D element, 3D elements
  - Frame element, plate element, shell element, solid element, etc.

Nodes and Finite Elements

- The **Finite Elements** are discretized representation of the continuous structure.
- Generally they correspond to the physical structural components but sometimes dummy or idealized elements may also be used.
- Elements behavior is completely defined within its boundaries and is not directly related to other elements.
- **Nodes** are imaginary points which serve to provide connectivity across element boundaries.

Solid – Structure - Model

- Discretization



Object

Elements

Nodes

Displacement

Stress

Strain

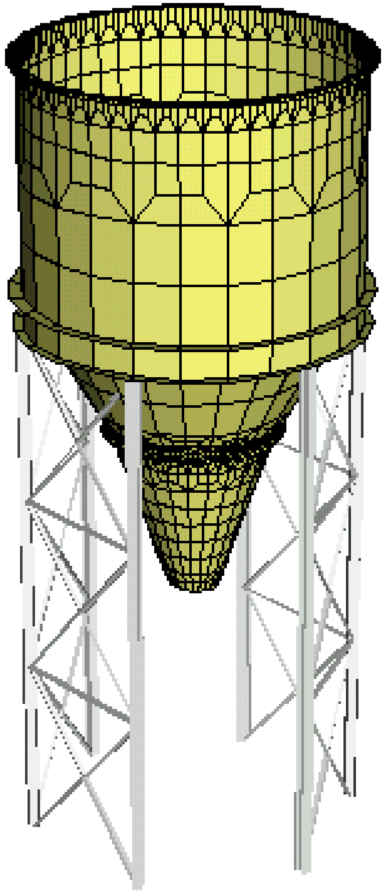
Basic Categories of Finite Elements

- **1 D Elements**
  - 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 Model
- **2 D Elements**
  - 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**
  - All three dimensions are modeled as a solid
  - Can be used in 3D Model

Global Modelling or “Macro Model”

- A model of the **Whole Structure**
- Objective is to get **Overall Structural Response**
- Results in the form of member forces and stress patterns

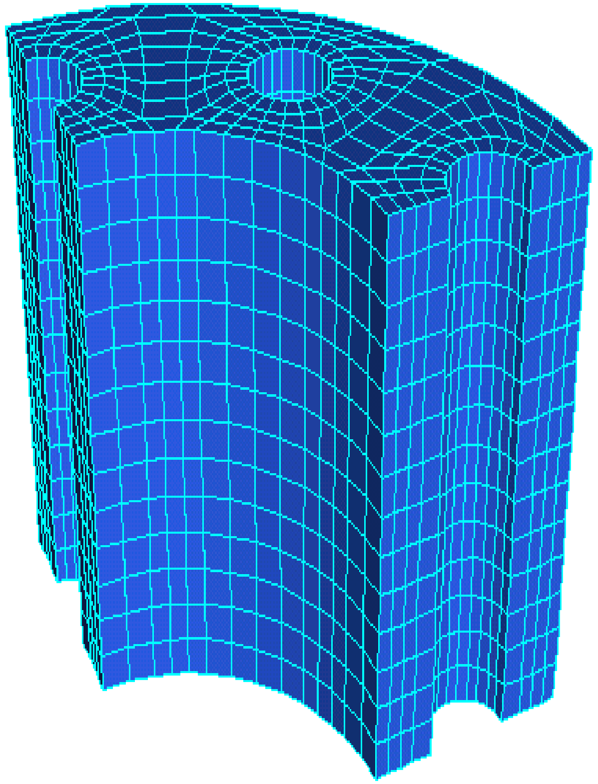
- Global Modeling is same for nearly all materials and material distinction is made by using specific material properties
- Global Model may be a simple 2D beam/ frame model or a sophisticated full 3D finite element model.
- Generally adequate for design of usual structures.



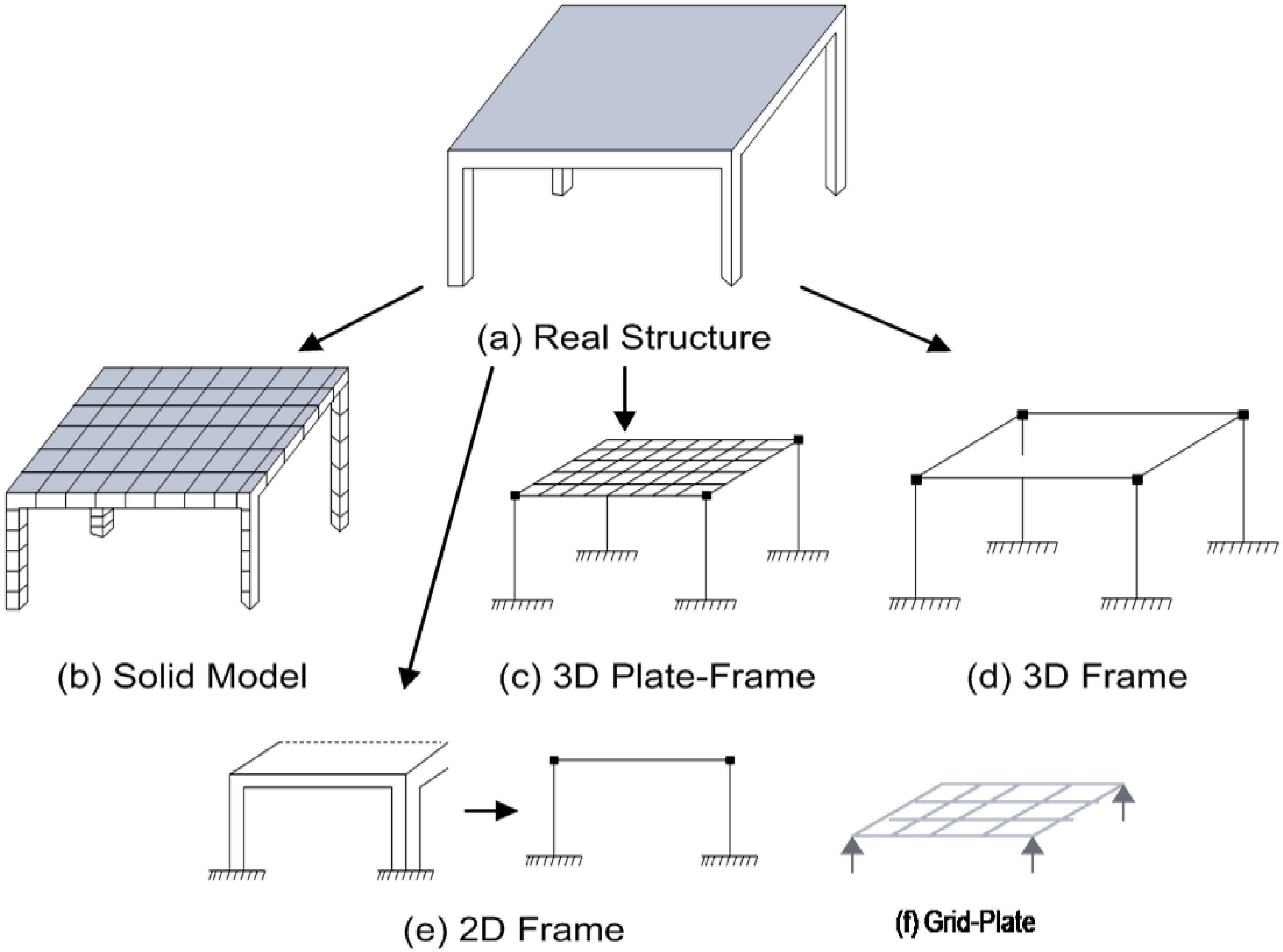
Local Model or “Micro Model”

- Model of **Single Member** or part of a Member
- Model of the Cross-section, Opening, Joints, connections etc.

- Objective is to determine **local stress concentration**, **cross-section behavior, modeling of cracking, bond, anchorage** etc.
- Needs finite element modeling using very fine mesh, advance element features, non-linear analysis etc.
- Mostly suitable for research, simulation, experiment verification and theoretical studies.



Global Modelling of Structural Geometry - Various Ways to Model a Real Structure



The Selection of Elements

- Identify basic issue and need for modeling
- See how materials are represented and modeled
- Explore the types and properties of various elements
- Select appropriate elements for representing different structural members

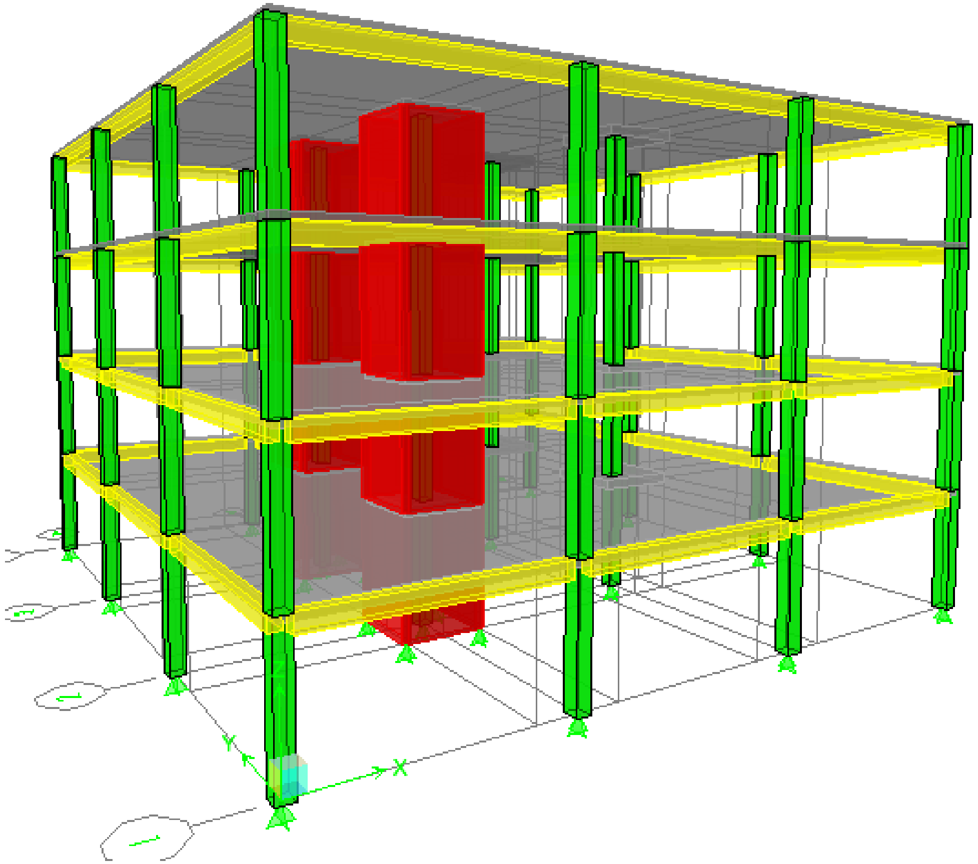
Current Modelling Trends – Object Based Modelling

- In several software, the **Graphic Objects** representing the Structural Members are automatically divided into **Finite Elements** for analysis → Object-based Modeling

- This involves
  - Auto Meshing
  - Auto Load Computation
  - Auto Load Transfer

Object Based Modelling

- Level-1
  - The Nodes are defined first by coordinates and then Elements are defined that connect the nodes
- Level-2
  - The Elements are defined directly, either numerically or graphically and the Nodes are created automatically
- Level-3
  - The structure is represented by generic Objects and the elements and Nodes are created automatically



SAP/ETABS (Objects → Elements)

- The **physical structural members** in a structural model are represented by **objects**.
- Using the graphical user interface, you “draw” the geometry of an object, then “assign” properties and loads to the object to completely define the model of the physical member.
- For analysis purposes, SAP2000 **converts each object into one or more (finite) elements** and develop an “**analysis model**”.

SAP/ETABS (Objects → Elements)

The following object types are available, listed in order of geometrical dimension:

- **Point objects**, are of two types:
  - Joint objects: Automatically created at the corners or ends of all other types of objects; also can be explicitly added to represent supports or to capture other localized behavior.
  - Grounded (one-joint) link/support objects: Used to model special member behavior
- **Line objects**, are of four types
  - Frame objects: Used to model beams, columns, braces, and trusses
  - Cable objects: Used to model slender cables under self weight and tension
  - Tendon objects: Used to prestressing ten dons within other objects
  - Connecting (two-joint) link/support objects: Used to model special member behavior
- **Area objects**: Shell elements (plate, membrane, and full-shell) used to model walls, floors, and other thin-walled members)
- **Solid objects**: Used to model three-dimensional solids.

SAP/ETABS (Objects → Elements)

- When you run an analysis, SAP/ETABS automatically converts your **object-based model** into an **element-based model** that is used for analysis.

- This finite element-based model is called the **analysis model**, and it consists of traditional finite elements and joints (nodes).

- Results of the analysis are reported back on the **object-based model**.

- **Group:**

  A group is a named collection of objects that you define. For each group, you must provide a unique name, then select the objects that are to be part of the group. You can include objects of any type or types in a group. Each object may be part of one of more groups.

Structural Discretization – The Element Menu

Finite Element Modeling, Analysis and Design Process

✓ -1 Idealization

→ ✓ 0 Discretization

1 Materials

2 Cross-sections

3 Members

4 Structure

$= K$

5 Loads (F or $F(t)$ vector)

6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$

7 Post-processing, Local Forces

8 Structural Design

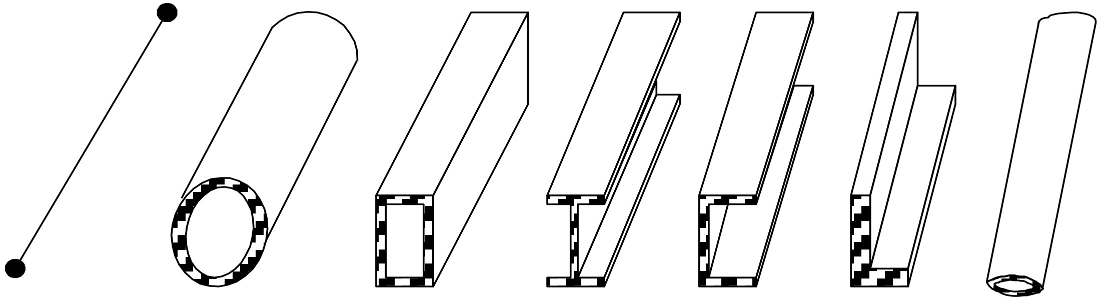
🟦 = Our task; FE software doesn't help.

🟩 = We provide to the FE software.

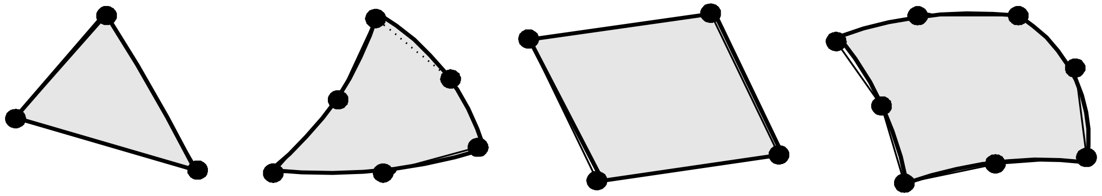
🟧 = FE software does for us.

Some Finite Elements

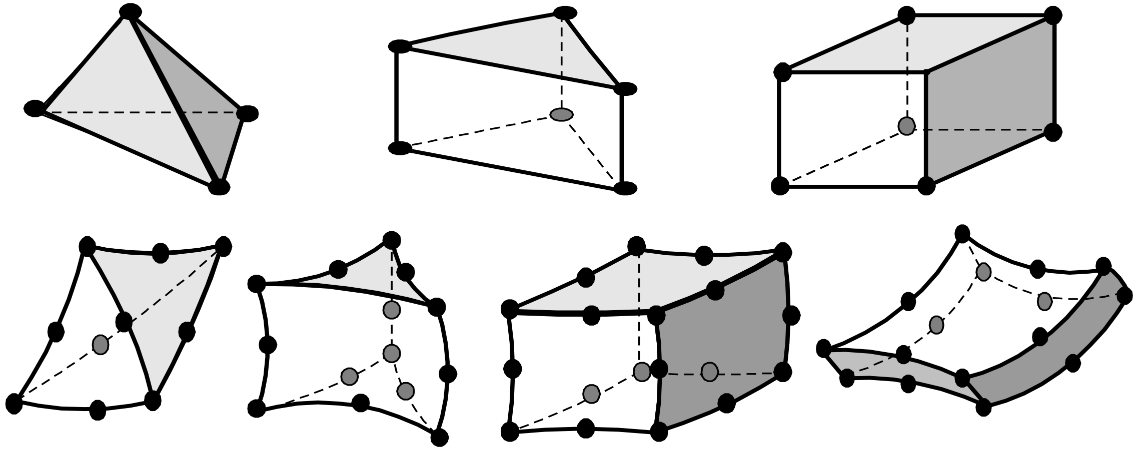
- Not all of these finite elements are available in SAP/ETABS.
- ABAQUS CAE
- ANSYS
- DIANA FEA
- OPENSEES
- Agros2D
- DUNE
- Femap
- GOMA
- GetFEM++
- Robot
- STAAD Pro
- ADINA
- LS DYNA
- Nastran
- Visual FEA



Truss and Beam Elements (1D,2D,3D)



Plane Stress, Plane Strain, Axisymmetric, Plate and Shell Elements (2D,3D)

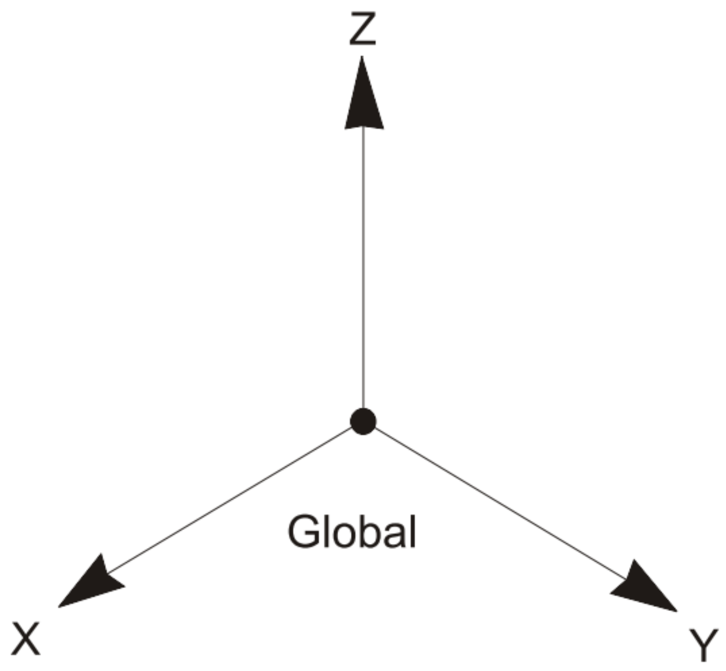


Brick Elements

Joints, Degrees of Freedom, Boundary Conditions and Constraints

Global Coordinate System in SAP/ETABS

- SAP2000/ETABS always assumes that **Z is the vertical axis**, with +Z being upward.
- Local coordinate systems for joints, elements, and ground-acceleration loading are defined with respect to this upward direction.
- Self-weight loading always acts downward, in the –Z direction.
- **The X-Y plane is horizontal.**



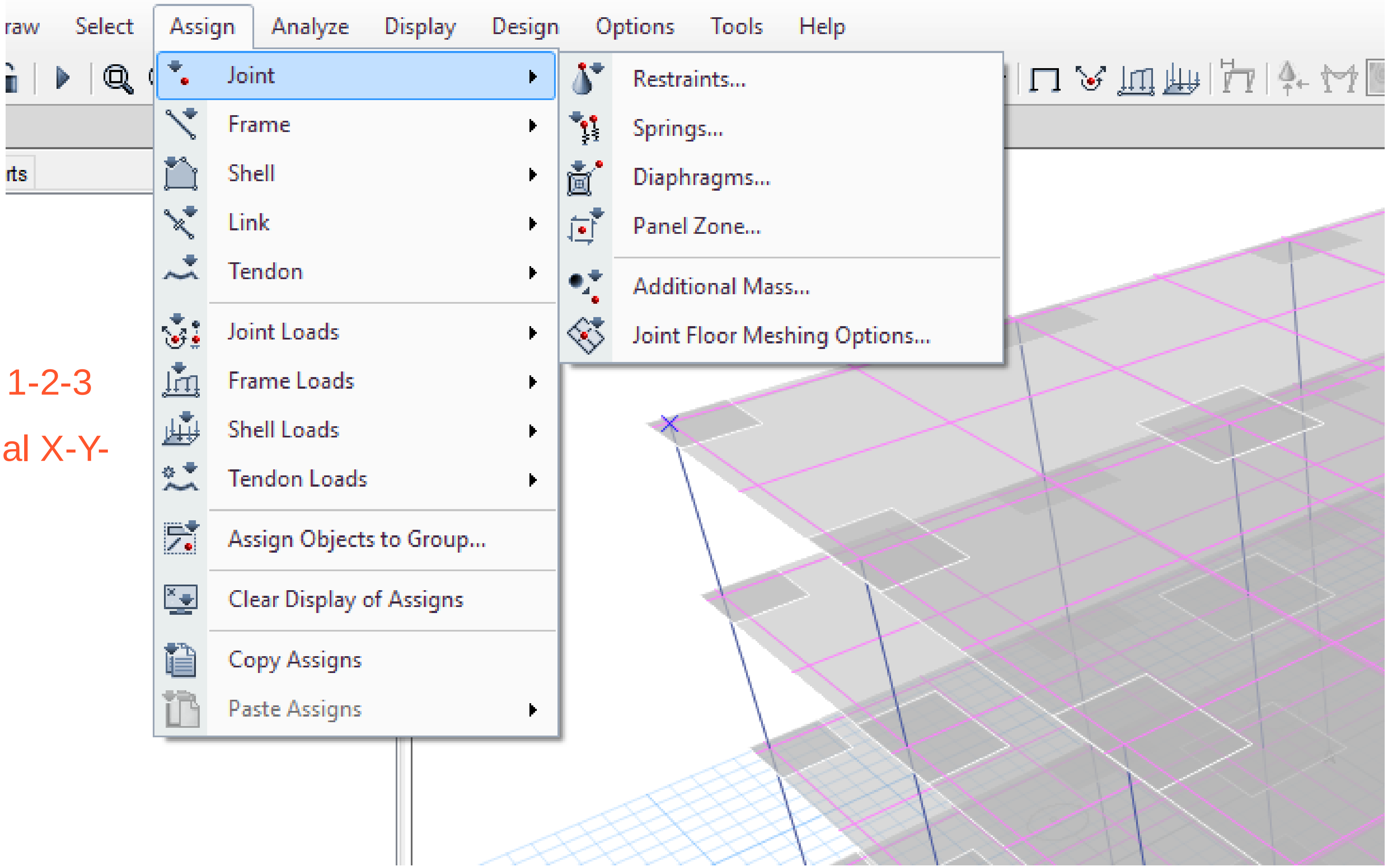
Joint Elements

- **Joints,** also known as nodal points or nodes, are a fundamental part of every structural model.
- Joints are the primary locations in the structure at which the displacements are known (i.e. the supports) or are to be determined.
- In SAP/ETABS, a joint is defined by specifying its label and three spatial coordinates that locate the joint in space.
  - All elements are connected to the structure at the joints.
  - The structure is supported at the joints using Restraints and/or Springs.
  - Rigid-body behavior and symmetry conditions can be specified using Constraints that apply to the joints.
  - Concentrated loads may be applied at the joints.
  - Lumped masses and rotational inertia may be placed at the joints.
  - Panel Zones can be assigned to joints.
  - Loads and masses applied to the elements are transferred to the joints.

Joint Elements

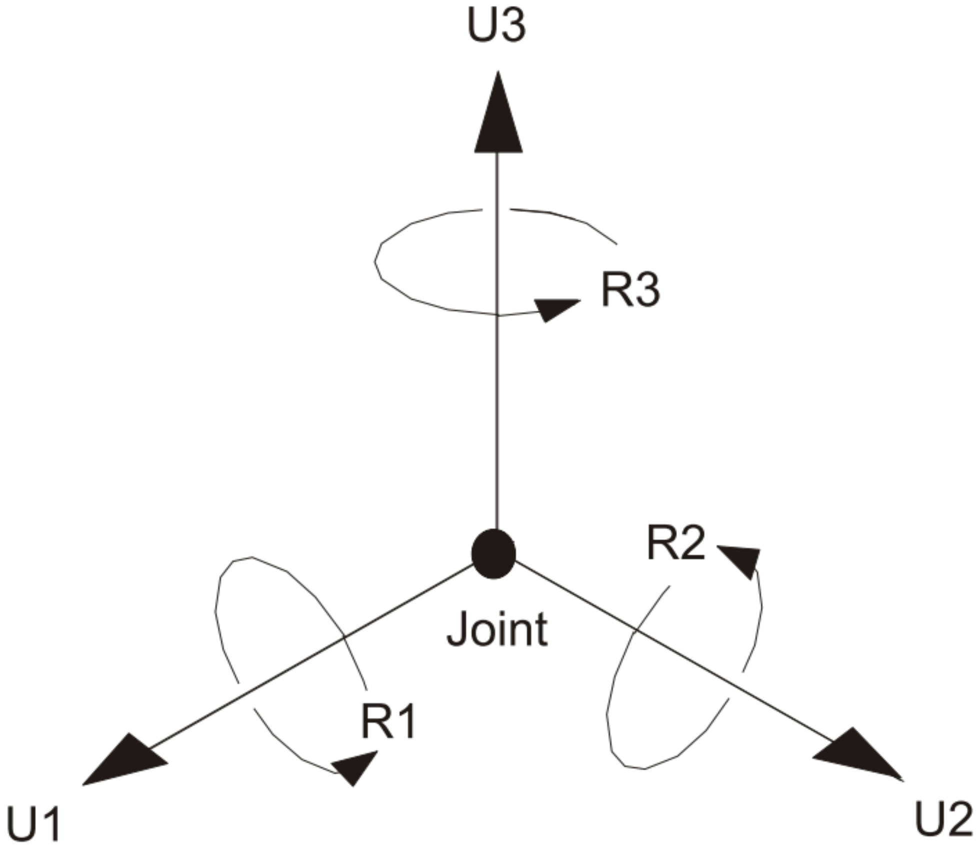
Local Coordinates of Joint Elements:

In SAP/ETABS, by default, the joint local 1-2-3 coordinate system is identical to the global X-Y-Z coordinate system



Degrees of Freedom

- Every joint of the structural model may have up to **six displacement components or degrees of freedom.**
  - The joint may translate along its three local axes. These translations are de-noted U1, U2, and U3.
  - The joint may rotate about its three local axes. These rotations are denoted R1, R2, and R3.



Three Translations and Three Rotations

Degrees of Freedom

- Each degree of freedom in the structural model must be one of the following types:
  - **Active** — the displacement is computed during the analysis
  - **Restrained** — the displacement is specified, and the corresponding reaction is computed during the analysis
  - **Constrained** —the displacement is determined from the displacements at other degrees of freedom
  - **Null** — the displacement does not affect the structure and is ignored by the analysis
  - **Unavailable** —the displacement has been explicitly excluded from the analysis

Degrees of Freedom - Availability

- By default, **all six degrees of freedom are available** to every joint. This default should generally be used for all three-dimensional structures.
- The degrees of freedom that are not specified as being available are called **unavailable** degrees of freedom.
- Any stiffness, loads, mass, restraints, or constraints that are applied to the unavailable degrees of freedom are **ignored by the analysis**.

Constrained Degrees of Freedom

- Any joint that is part of a **Constraint** may have one or more of its available degrees of freedom constrained.
- The program automatically creates a **master joint** to govern the behavior of each Constraint.
- The displacement of a constrained degree of freedom is then computed as a linear combination of the displacements along the degrees of freedom at the corresponding master joint.
- **A degree of freedom may not be both constrained and restrained.**

Restrained Degrees of Freedom

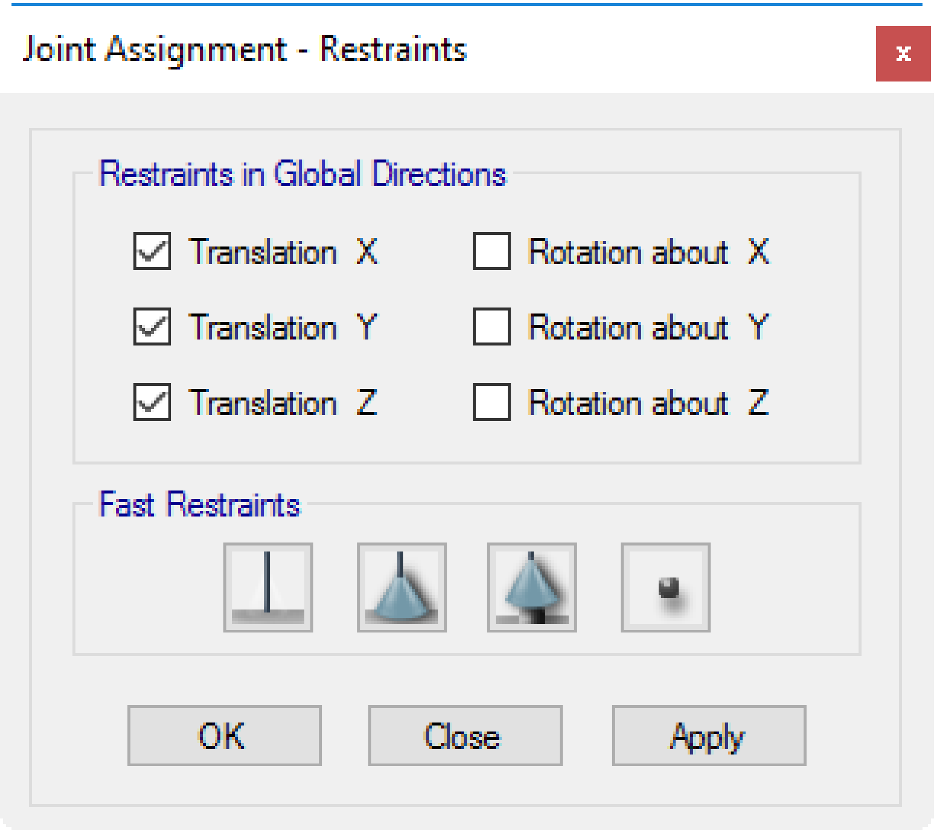
- If the displacement of a joint along any one of its available degrees of freedom is known, such as at a **support point**, that degree of freedom is restrained.
- The known value of the displacement may be zero or non-zero, and may be different in different Load Cases.
- The force along the restrained degree of freedom that is required to impose the specified restraint displacement is called the **reaction**, and is determined by the analysis.
- **Unavailable degrees of freedom are essentially restrained.**

Modelling of Boundary Conditions

- Restraint Supports
- Spring Supports
- Nonlinear Supports
- Distributed Spring Supports

Modelling of Boundary Conditions – Restraint Supports

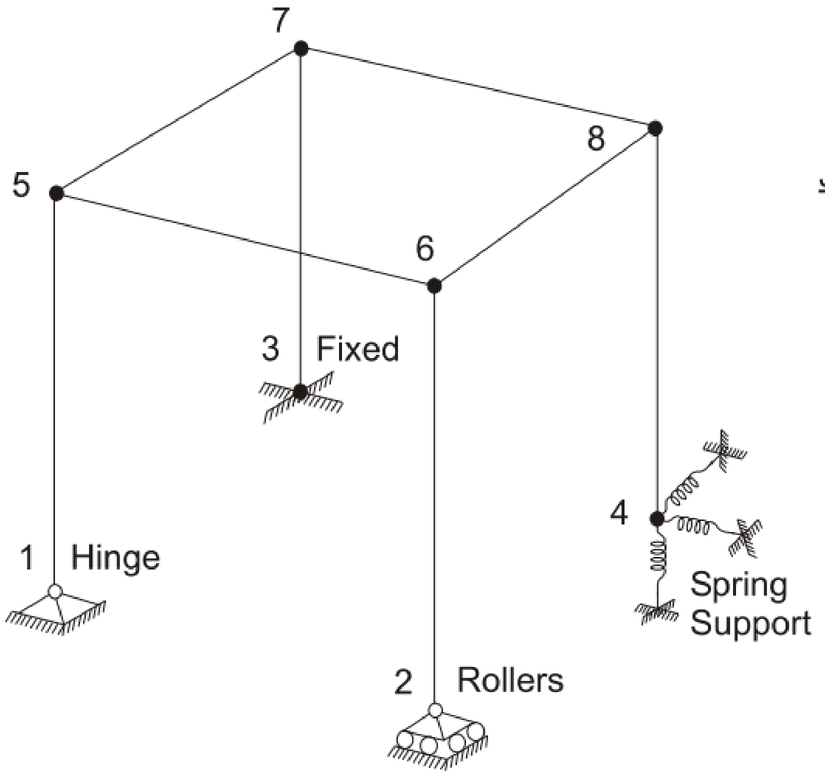
- Restraints are always applied to the joint local degrees of freedom **U1, U2, U3, R1, R2,** and **R3**.
- If a restraint is applied to an unavailable degree of freedom, it is ignored. The displacement will be zero, but no reaction will be computed.



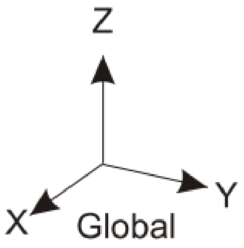
Joint Assignment - Restraints

Restraints in Global Directions

☑ Translation X
☑ Translation Y
☑ Translation Z
☐ Rotation about X
☐ Rotation about Y
☐ Rotation about Z



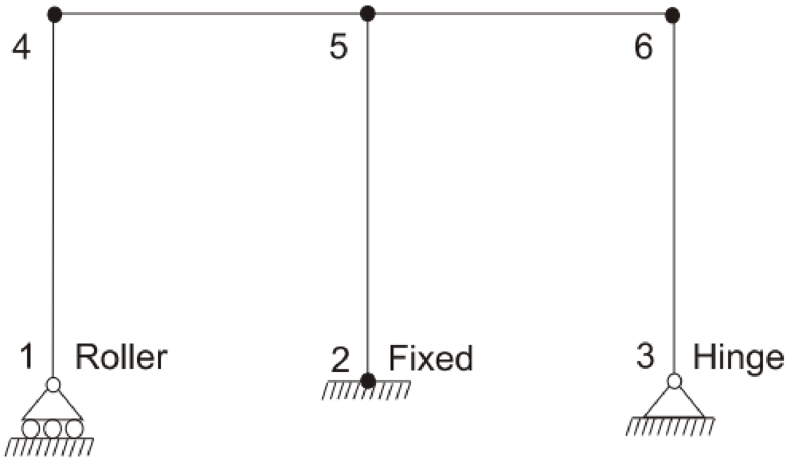
Joint	Restraints
1	U1, U2, U3
2	U3
3	U1, U2, U3, R1, R2, R3
4	None



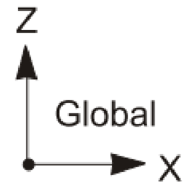
3-D Frame Structure

Notes: Joints are indicated with dots:
- ● Solid dots indicate moment continuity
- ○ Open dots indicate hinges

All joint local 1-2-3 coordinate systems are identical to the global X-Y-Z coordinate system



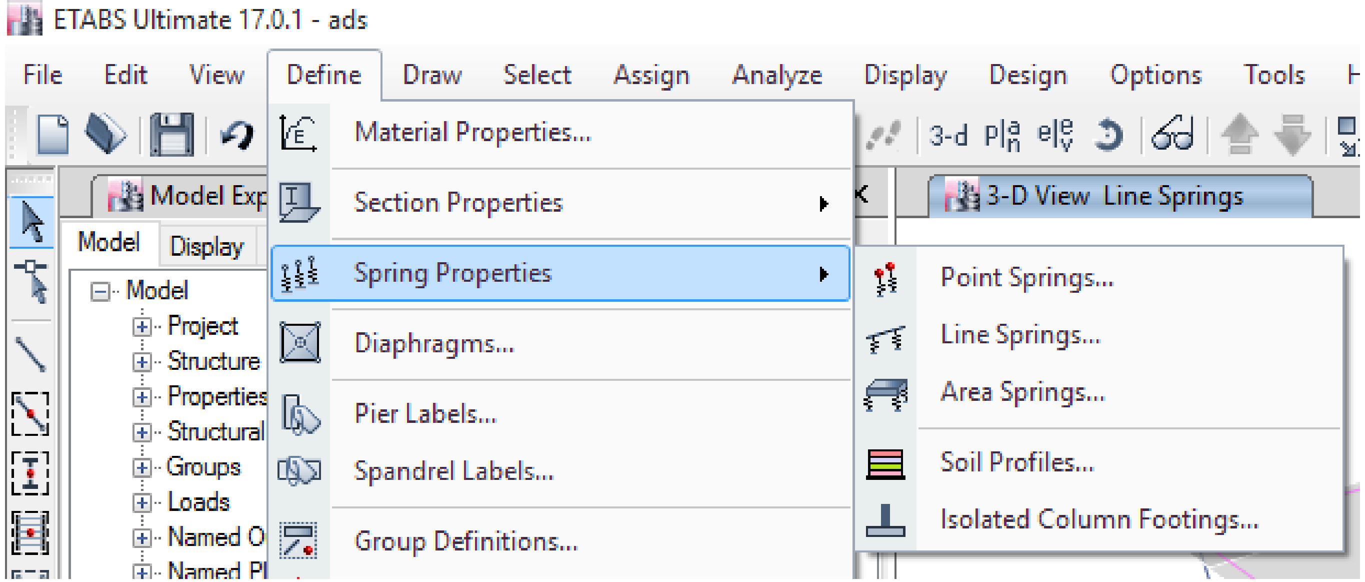
Joint	Restraints
All	U2, R1, R3
1	U3
2	U1, U3, R2
3	U1, U3



2-D Frame Structure, X-Z plane

Modelling of Boundary Conditions – Spring Supports

- Any of the six degrees of freedom at any of the joints in the structure can have **translational or rotational spring support conditions**. These springs **elastically** connect the joint to the ground.
- Springs may be specified that couple the degrees of freedom at a joint. The spring forces that act on a joint are related to the displacements of that joint by a **6x6 symmetric matrix** of spring stiffness coefficients. **These forces tend to oppose the displacements.**



Stiffness Matrix for Spring Element

- In a joint local coordinate system, the spring forces and moments F_1, F_2, F_3, M_1, M_2 and M_3 at a joint are given by:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = -\begin{bmatrix} \mathbf{u1} & \mathbf{u1u2} & \mathbf{u1u3} & \mathbf{u1r1} & \mathbf{u1r2} & \mathbf{u1r3} \\ & \mathbf{u2} & \mathbf{u2u3} & \mathbf{u2r1} & \mathbf{u2r2} & \mathbf{u2r3} \\ & & \mathbf{u3} & \mathbf{u3r1} & \mathbf{u3r2} & \mathbf{u3r3} \\ & & & \mathbf{r1} & \mathbf{r1r2} & \mathbf{r1r3} \\ & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{r2r3} \\ & & & & & \mathbf{r3} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ r_1 \\ r_2 \\ r_3 \end{Bmatrix}$$

- where u_1 ,u_2 ,u_3 ,r_1 ,r_2 and r_3 are the joint displacements and rotations, and the terms $u1, u1u2, u2$, ... are the specified spring stiffness coefficients.

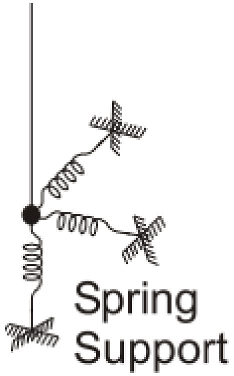
Modelling of Boundary Conditions – Spring Supports

- For springs that **do not couple** the degrees of freedom in a particular coordinate system, **only the six diagonal terms** need to be specified since the off-diagonal terms are all zero.
- When **coupling is present, all 21 coefficients** in the upper triangle of the matrix must be given; the other 15 terms are then known by symmetry.

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = -\begin{bmatrix} \mathbf{u1} & \mathbf{u1u2} & \mathbf{u1u3} & \mathbf{u1r1} & \mathbf{u1r2} & \mathbf{u1r3} \\ & \mathbf{u2} & \mathbf{u2u3} & \mathbf{u2r1} & \mathbf{u2r2} & \mathbf{u2r3} \\ & & \mathbf{u3} & \mathbf{u3r1} & \mathbf{u3r2} & \mathbf{u3r3} \\ & & & \mathbf{r1} & \mathbf{r1r2} & \mathbf{r1r3} \\ & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{r2r3} \\ & & & & & \mathbf{r3} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ r_1 \\ r_2 \\ r_3 \end{Bmatrix}$$

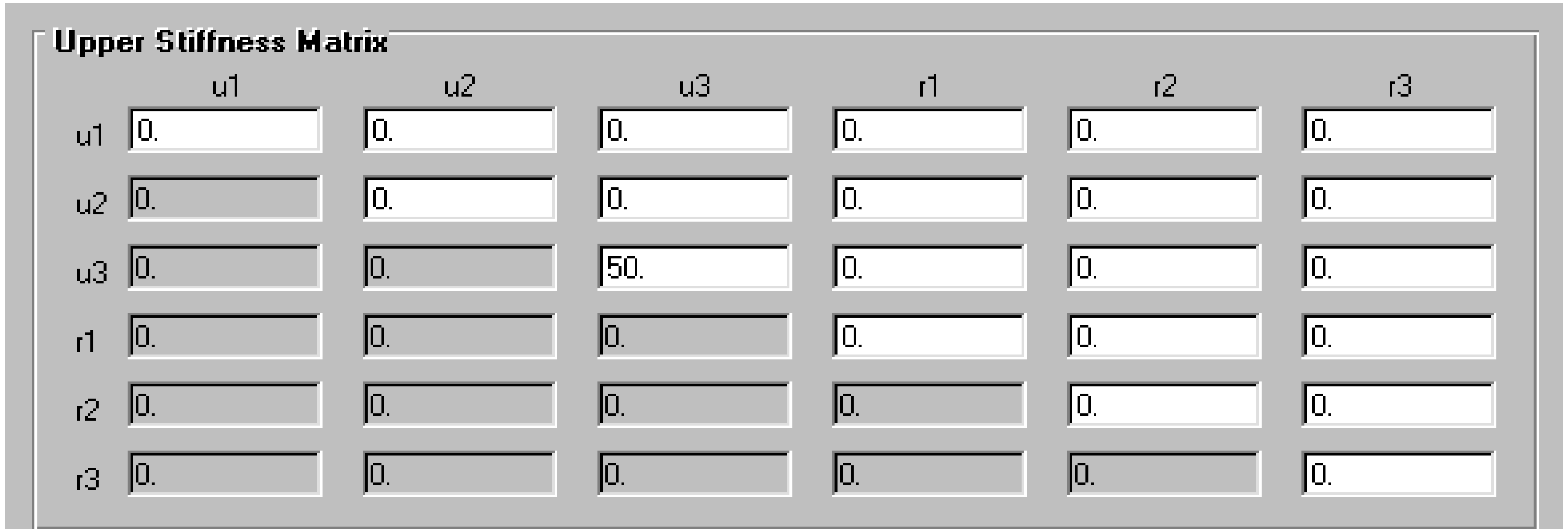
Coupled Spring Restraints

- Coupled 6x6 user-defined spring stiffness option

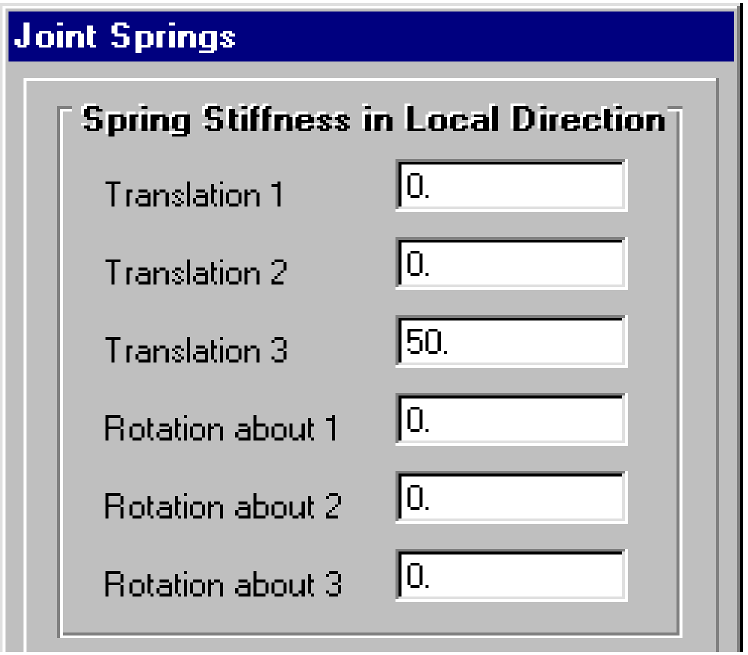


Uncoupled Spring Restraints

- Independent springs stiffness in each DOF


Upper Stiffness Matrix

	u1	u2	u3	r1	r2	r3
u1	0.	0.	0.	0.	0.	0.
u2	0.	0.	0.	0.	0.	0.
u3	0.	0.	50.	0.	0.	0.
r1	0.	0.	0.	0.	0.	0.
r2	0.	0.	0.	0.	0.	0.
r3	0.	0.	0.	0.	0.	0.

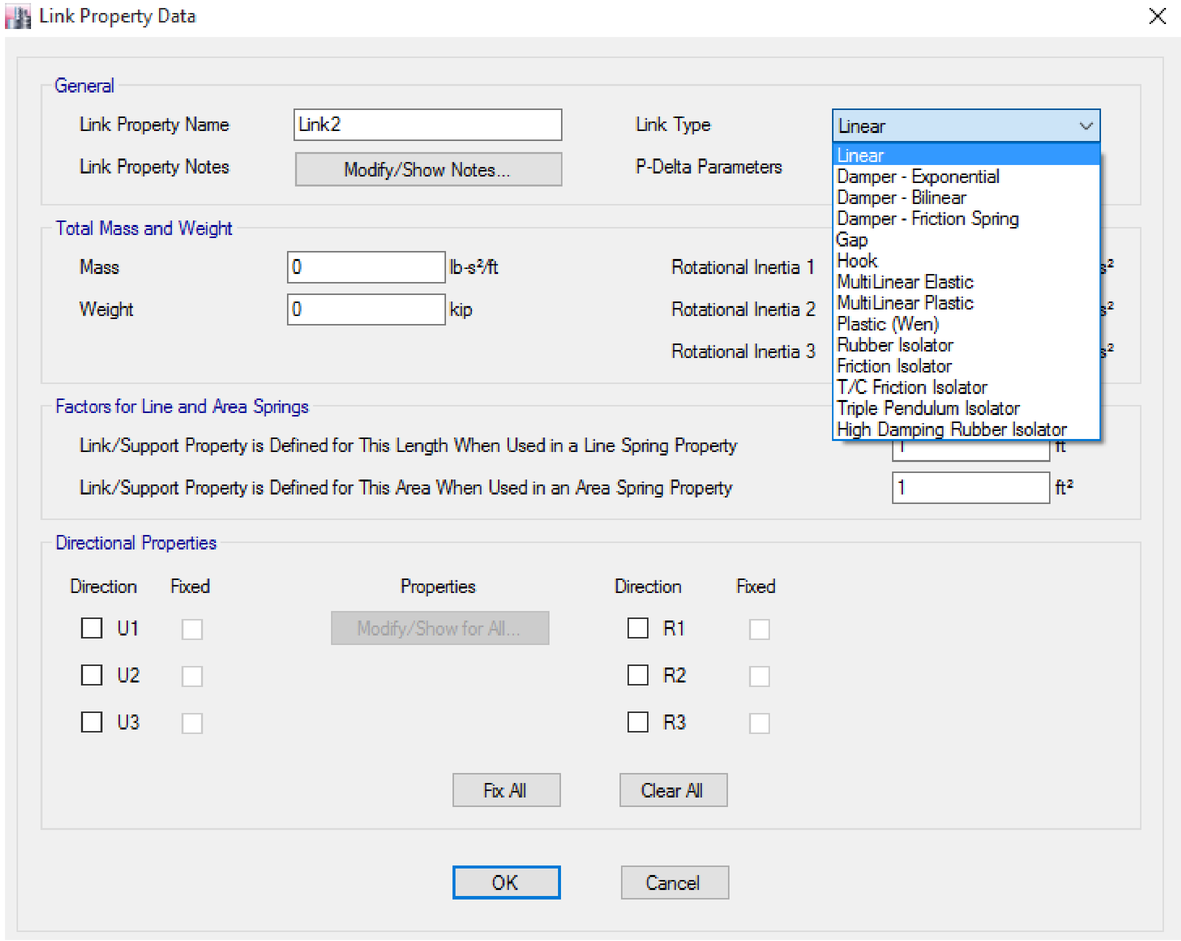

Joint Springs

Spring Stiffness in Local Direction

Translation 1	0.
Translation 2	0.
Translation 3	50.
Rotation about 1	0.
Rotation about 2	0.
Rotation about 3	0.

Modelling of Boundary Conditions – Nonlinear Supports

- **Nonlinear support conditions** that can be modeled include gaps (compression only), multi-linear elastic or plastic springs, viscous dampers, base isolators, and more.
- This Link/Support can be used in two ways:
    - You can add (draw) a **one-joint object**, in which case it is considered a Support object, and it connects the joint directly to the ground.
    - The object can also be drawn with two joints, in which case it is considered **Link object**. You can use a Link object as a support if you connect one end to the structure, and fully restrain the other end.



Link Property Data

General

Link Property Name: Link2

Link Type: Linear

Total Mass and Weight

Mass: 0 lb-s²/ft

Weight: 0 kip

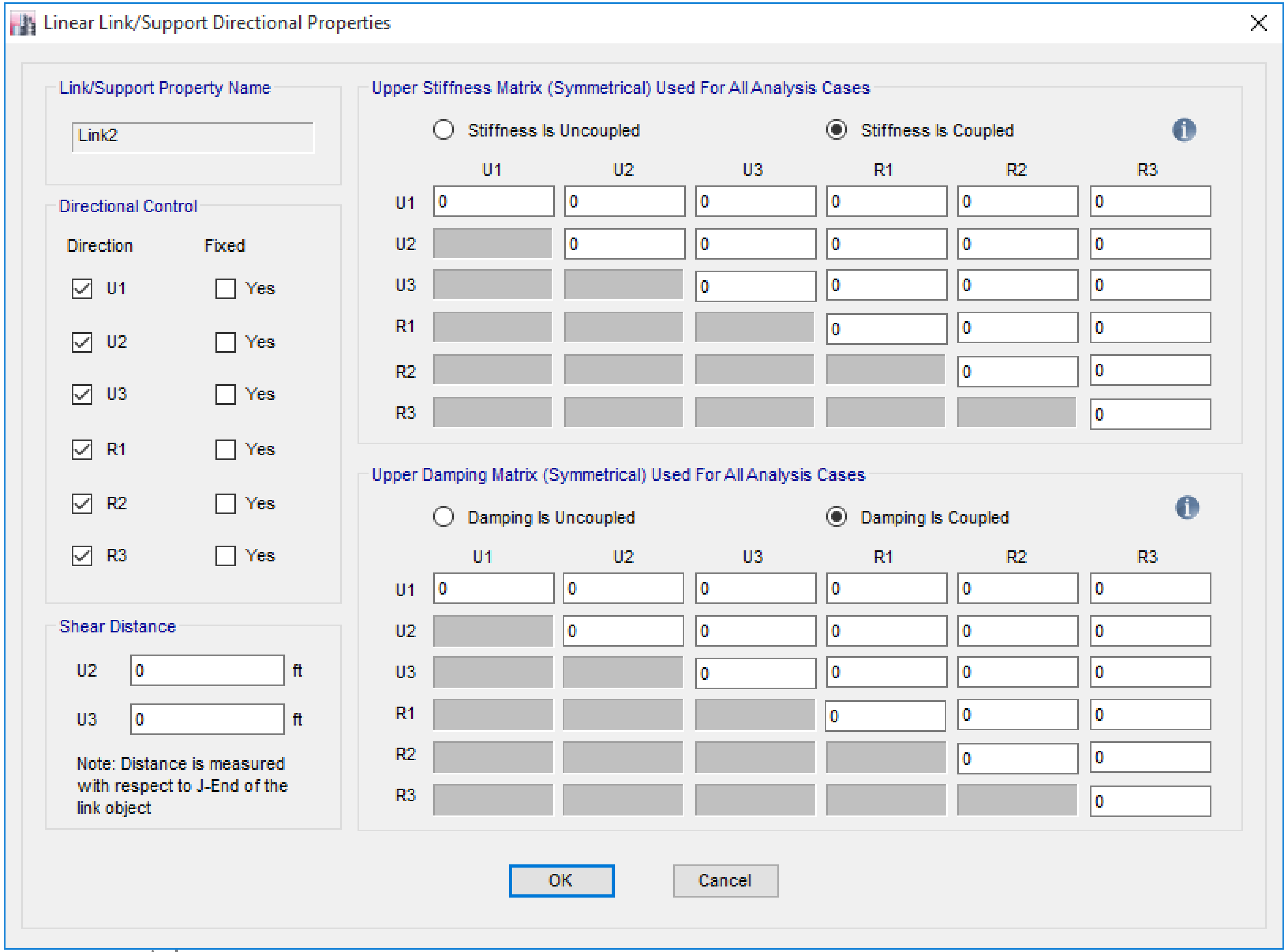
Factors for Line and Area Springs

Link/Support Property is Defined for This Length When Used in a Line Spring Property: 1 ft

Link/Support Property is Defined for This Area When Used in an Area Spring Property: 1 ft²

Directional Properties

Modelling of Boundary Conditions – Link Supports



Linear Link/Support Directional Properties

Link/Support Property Name

Link2

Directional Control

Direction	Fixed
☑ U1	☐ Yes
☑ U2	☐ Yes
☑ U3	☐ Yes
☑ R1	☐ Yes
☑ R2	☐ Yes
☑ R3	☐ Yes

Shear Distance

U2 0 ft

U3 0 ft

Note: Distance is measured with respect to J-End of the link object

Upper Stiffness Matrix (Symmetrical) Used For All Analysis Cases

○ Stiffness Is Uncoupled ● Stiffness Is Coupled

	U1	U2	U3	R1	R2	R3
U1	0	0	0	0	0	0
U2		0	0	0	0	0
U3			0	0	0	0
R1				0	0	0
R2					0	0
R3						0

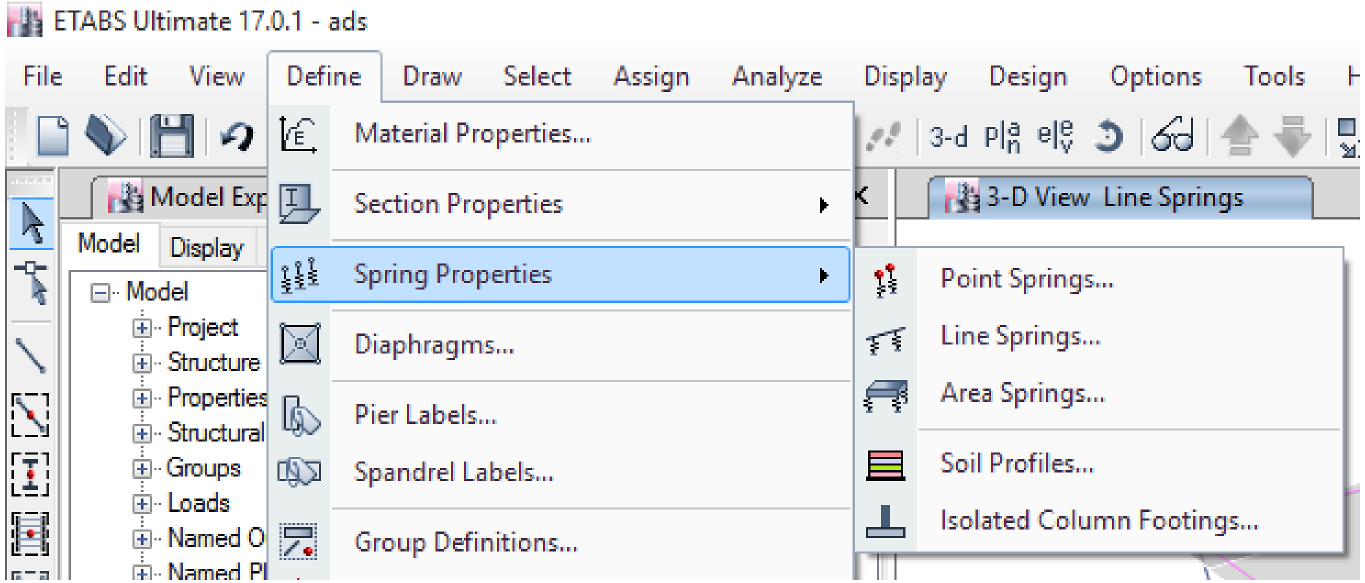
Upper Damping Matrix (Symmetrical) Used For All Analysis Cases

○ Damping Is Uncoupled ● Damping Is Coupled

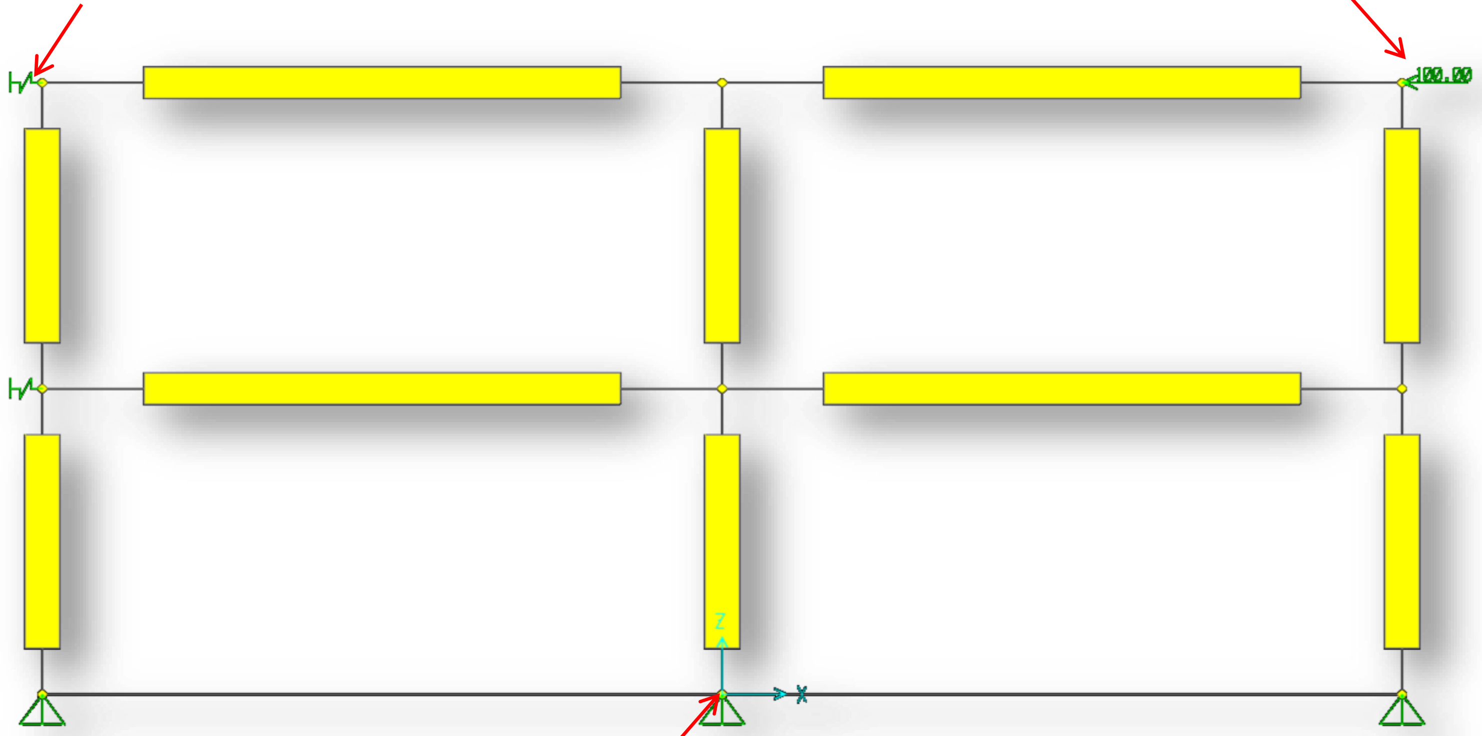
	U1	U2	U3	R1	R2	R3
U1	0	0	0	0	0	0
U2		0	0	0	0	0
U3			0	0	0	0
R1				0	0	0
R2					0	0
R3						0

Modelling of Boundary Conditions – Distributed Spring Supports

- You may assign distributed spring supports along the length of a Frame element, or over the any face of an area object (Shell, Plane, Asolid) or Solid element.
- These springs may be linear, multi-linear elastic, or multi-linear plastic.
- These springs are converted to **equivalent two-joint Link/Support elements acting at the joints of the element**, after accounting for the tributary length or area of the element. The generated Link/Support elements are of zero length, with one end connected to the parent object, and the other end connected to a generated joint that is fully restrained.



Joints in FE Model



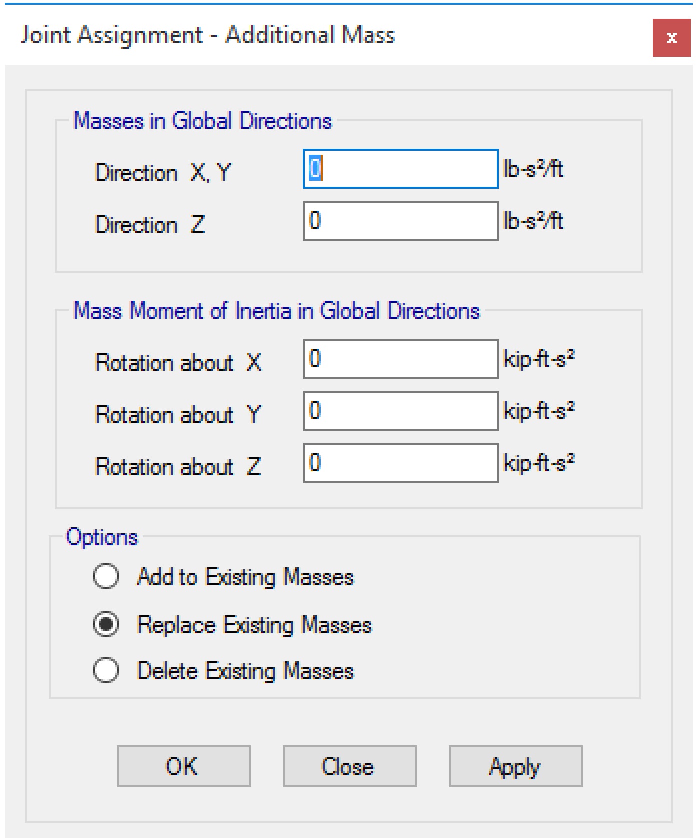
Spring Support

Joint Load

Hinge Support

Joint Masses

- In a dynamic analysis, the mass of the structure ($\boldsymbol{M}$ Matrix) is used to compute **inertial forces**.
- Normally, the mass is obtained from the elements using the mass density of the material and the volume of the element. This automatically produces lumped (uncoupled) masses at the joints. The element mass values are equal for each of the three translational degrees of freedom. No mass moments of inertia are produced for the rotational degrees of freedom. This approach is adequate for most analyses.
- It is often necessary to place **additional concentrated masses and/or mass moments of inertia** at the joints. These can be applied to any of the six degrees of freedom at any of the joints in the structure.
- For computational efficiency and solution accuracy, SAP2000 always uses lumped masses. This means that there is **no mass coupling between degrees of freedom** at a joint or between different joints. These uncoupled masses are always referred to the local coordinate system of each joint. Mass values along restrained degrees of freedom are ignored.



Joint Assignment - Additional Mass

Masses in Global Directions

Direction X, Y: 0 lb-s^2/ft

Direction Z: 0 lb-s^2/ft

Mass Moment of Inertia in Global Directions

Rotation about X: 0 kip-ft-s^2

Rotation about Y: 0 kip-ft-s^2

Rotation about Z: 0 kip-ft-s^2

Options

○ Add to Existing Masses

● Replace Existing Masses

○ Delete Existing Masses

Joint Masses

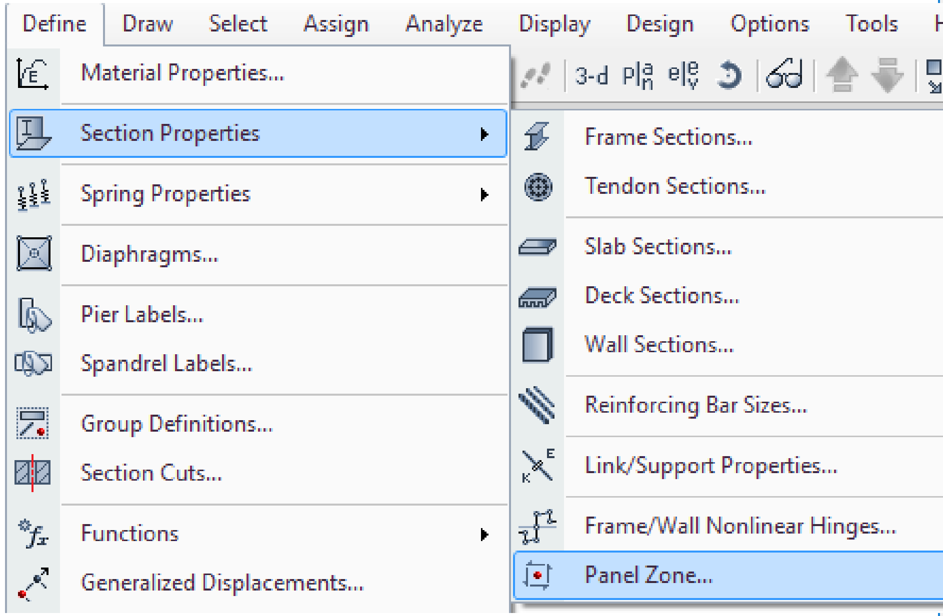
- Inertial forces acting on the joints are related to the accelerations at the joints by a 6x6 matrix of mass values. These forces tend to oppose the accelerations. In a joint local coordinate system, the inertia forces and moments F_1, F_2, F_3, M_1, M_2 and M_3 at a joint are given by:

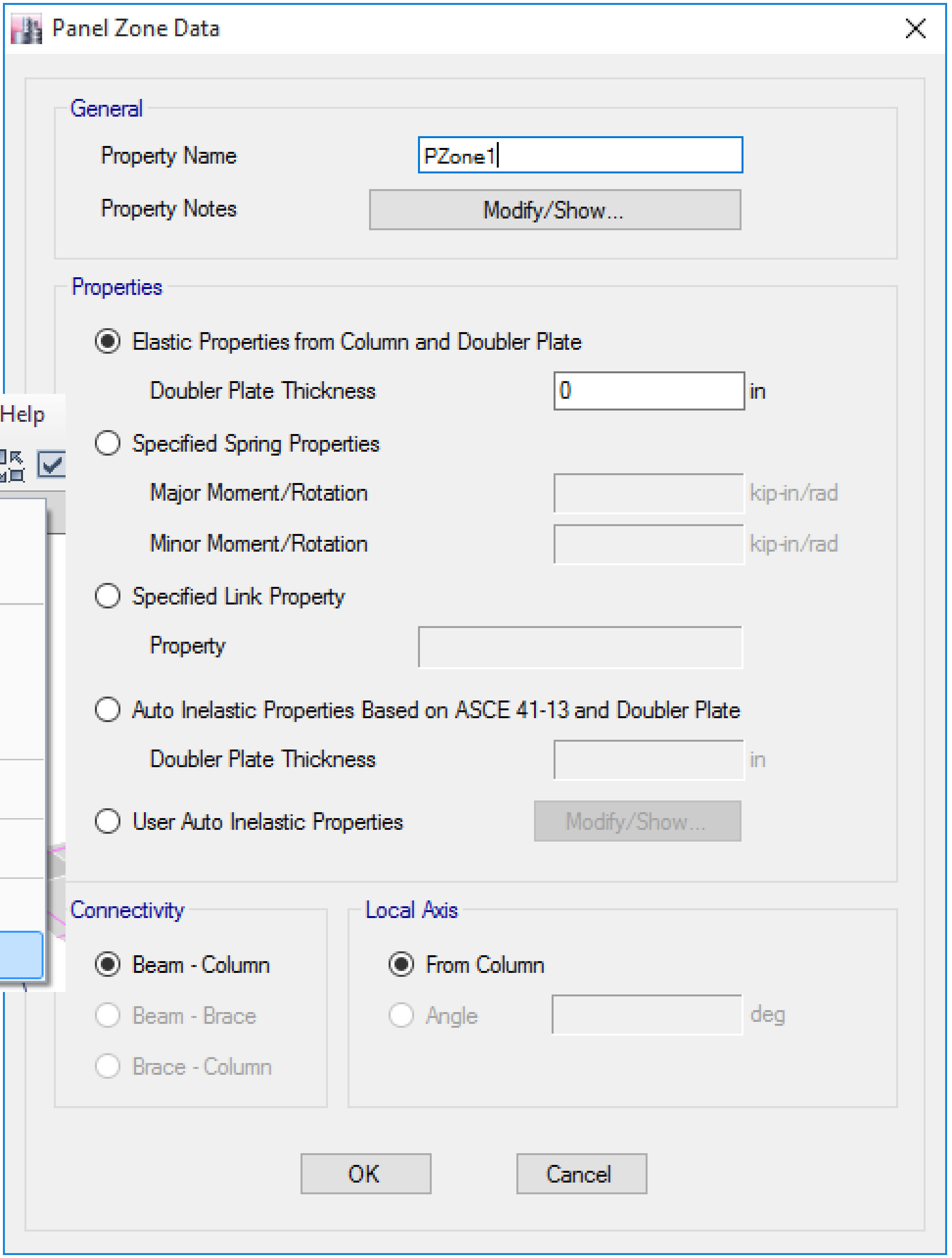
$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = -\begin{bmatrix} \mathbf{u1} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ & \mathbf{u2} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ & & \mathbf{u3} & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ & & & \mathbf{r1} & \mathbf{0} & \mathbf{0} \\ & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{0} \\ & & & & & \mathbf{r3} \end{bmatrix} \begin{Bmatrix} \ddot{u}_1 \\ \ddot{u}_2 \\ \ddot{u}_3 \\ \ddot{r}_1 \\ \ddot{r}_2 \\ \ddot{r}_3 \end{Bmatrix}$$

where $\ddot{u}_1, \ddot{u}_2, \ddot{u}_3, \ddot{r}_1, \ddot{r}_2, \ddot{r}_3$ are the translational and rotational accelerations at the joint, and the terms **u1**, **u2, u3, r1, r2**, and **r3** are the specified mass values.

Panel Zones

- Panel zones are assigned to joints to model the **flexibility of beam-column connections**. Panel zones may be **linear or nonlinear**, and may affect the following performance measures:
  - Structural drift
  - Structural period
  - Moment distribution





Property Name: PZone1

Property Notes

● Elastic Properties from Column and Doubler Plate

Doubler Plate Thickness: 0 in

○ Specified Spring Properties

Major Moment/Rotation: kip-in/rad

Minor Moment/Rotation: kip-in/rad

○ Specified Link Property

Property

○ Auto Inelastic Properties Based on ASCE 41-13 and Doubler Plate

Doubler Plate Thickness: in

○ User Auto Inelastic Properties

● Beam - Column

○ Beam - Brace

○ Brace - Column

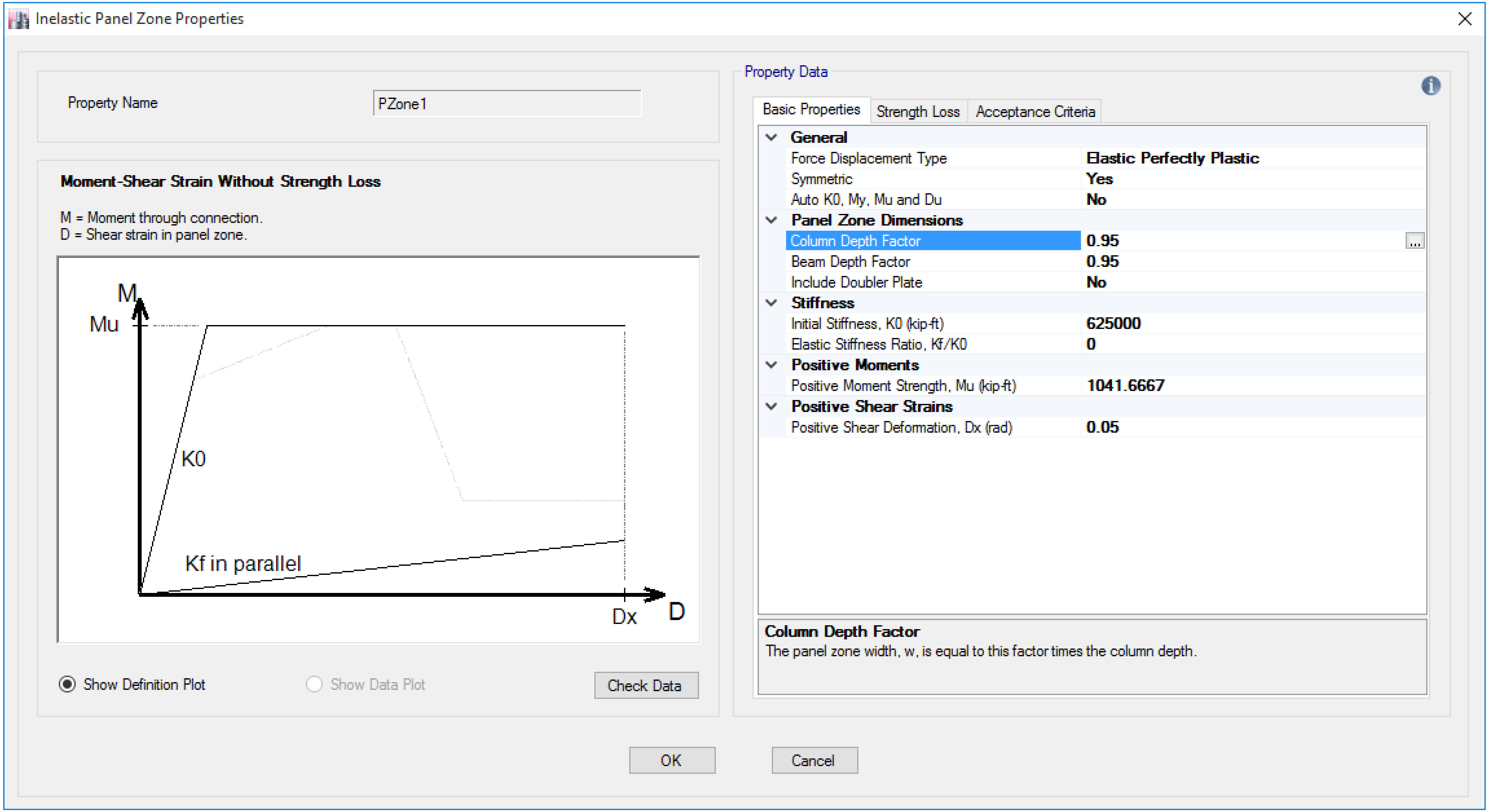
● From Column

○ Angle: deg

- Panel-zone results are available through Display > Show Tables > Analysis Results > Element Output > Panel Zone Output

Panel Zones

- Studies have shown that not accounting for the deformation within a beam-column panel zone in a model may cause a significant discrepancy between the analytical results and the physical behavior of the building.
- ETABS allows for the explicit incorporation of panel zone shear behavior and it is believed to have an appreciable impact on the deformation at the beam-to-column connection.
- Mathematically, panel zone deformation is modeled using **springs attached to rigid bodies geometrically the size of the panel zone**. ETABS allows the assignment of a panel zone "property" to a point object at the beam-column intersection. The properties of the panel zone may be determined in one of the following four ways:
  1) Automatically by the program from the elastic properties of the column.
  2) Automatically by the program from the elastic properties of the column in combination with any doubler plates that are present.
  3) User-specified spring values.
  4) Users-specified link properties, in which case it is possible to have inelastic panel zone behavior if performing a nonlinear time history analysis. Link properties may also be used to specify panel zone behavior for beam to brace and brace to column connections.



Property Name: PZone1

Moment-Shear Strain Without Strength Loss

M = Moment through connection.
D = Shear strain in panel zone.

● Show Definition Plot ○ Show Data Plot

Property Data

General

Force Displacement Type	**Elastic Perfectly Plastic**
Symmetric	**Yes**
Auto K0, My, Mu and Du	**No**

Panel Zone Dimensions

Column Depth Factor	**0.95**
Beam Depth Factor	**0.95**
Include Doubler Plate	**No**

Stiffness

Initial Stiffness, K0 (kip-ft)	**625000**
Elastic Stiffness Ratio, Kf/K0	**0**

Positive Moments

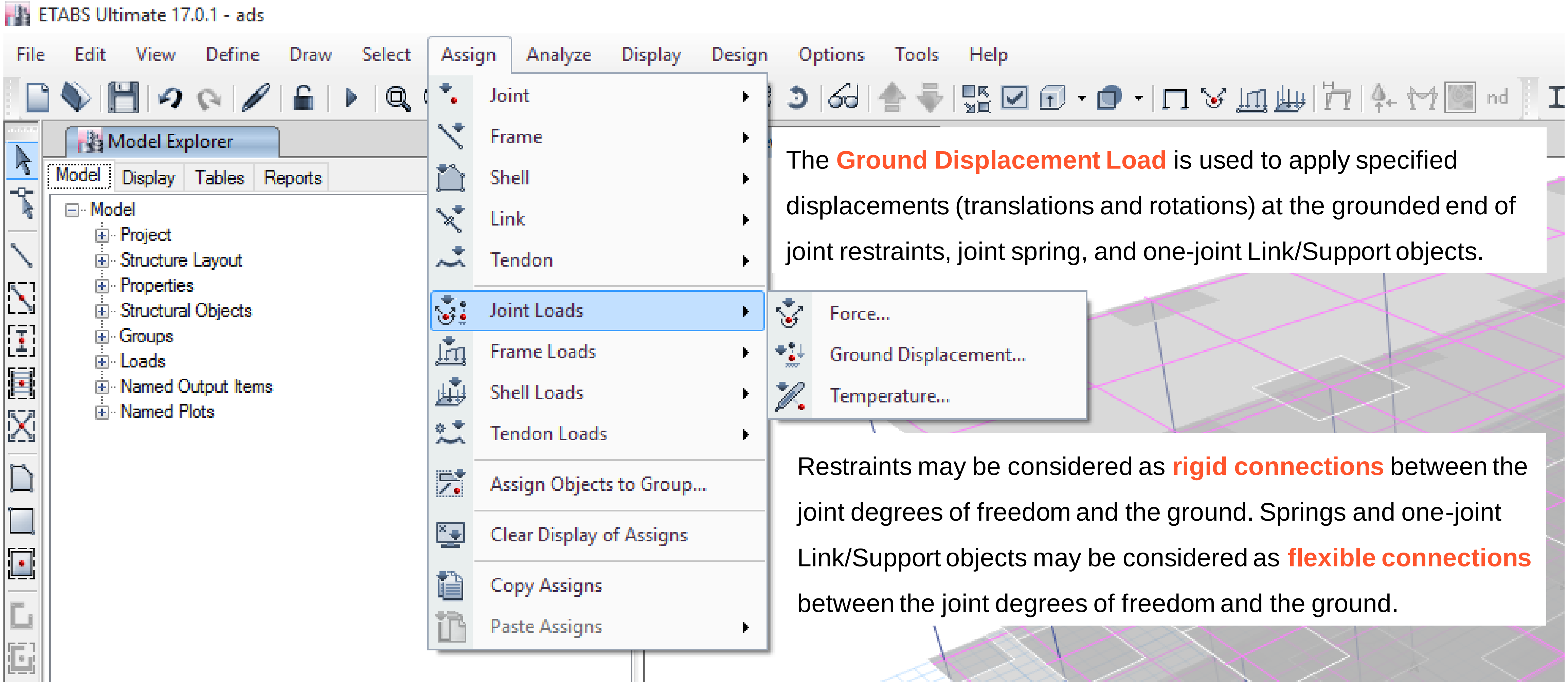
Positive Moment Strength, Mu (kip-ft)	**1041.6667**

Positive Shear Strains

Positive Shear Deformation, Dx (rad)	**0.05**

Column Depth Factor
The panel zone width, w, is equal to this factor times the column depth.

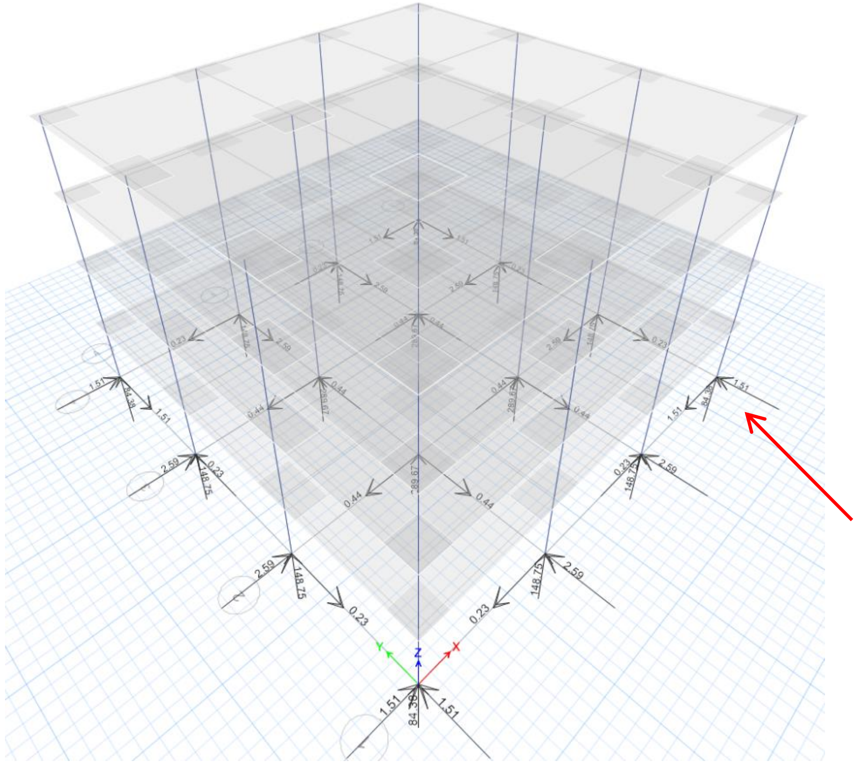
Joint Loads



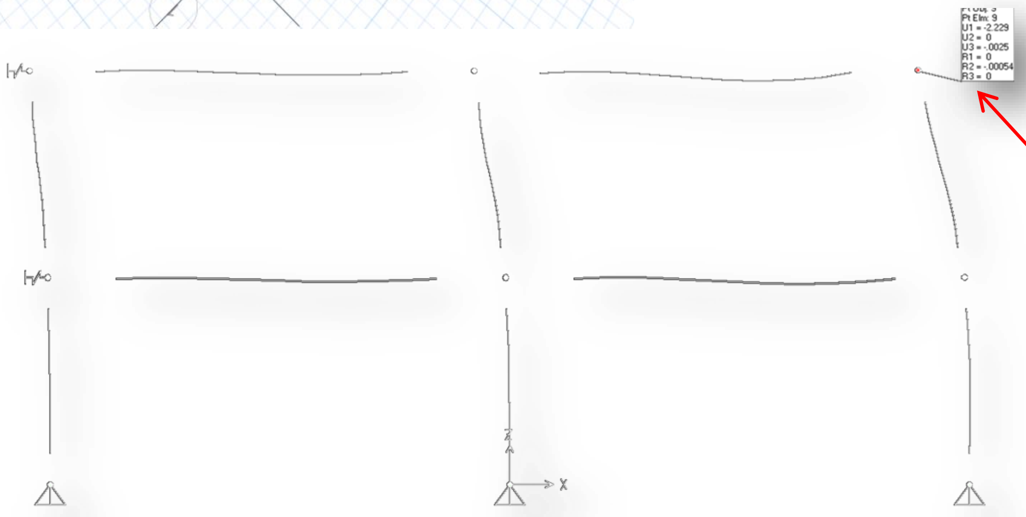
The **Ground Displacement Load** is used to apply specified displacements (translations and rotations) at the grounded end of joint restraints, joint spring, and one-joint Link/Support objects.

Restraints may be considered as **rigid connections** between the joint degrees of freedom and the ground. Springs and one-joint Link/Support objects may be considered as **flexible connections** between the joint degrees of freedom and the ground.

Joint Results



Reaction Forces



```
Pt Elm: 9
U1 = -2.229
U2 =  0
U3 = -.0025
R1 =  0
R2 = -.00054
R3 =  0
```

Joint Displacement

Using Constraints in Structural Model

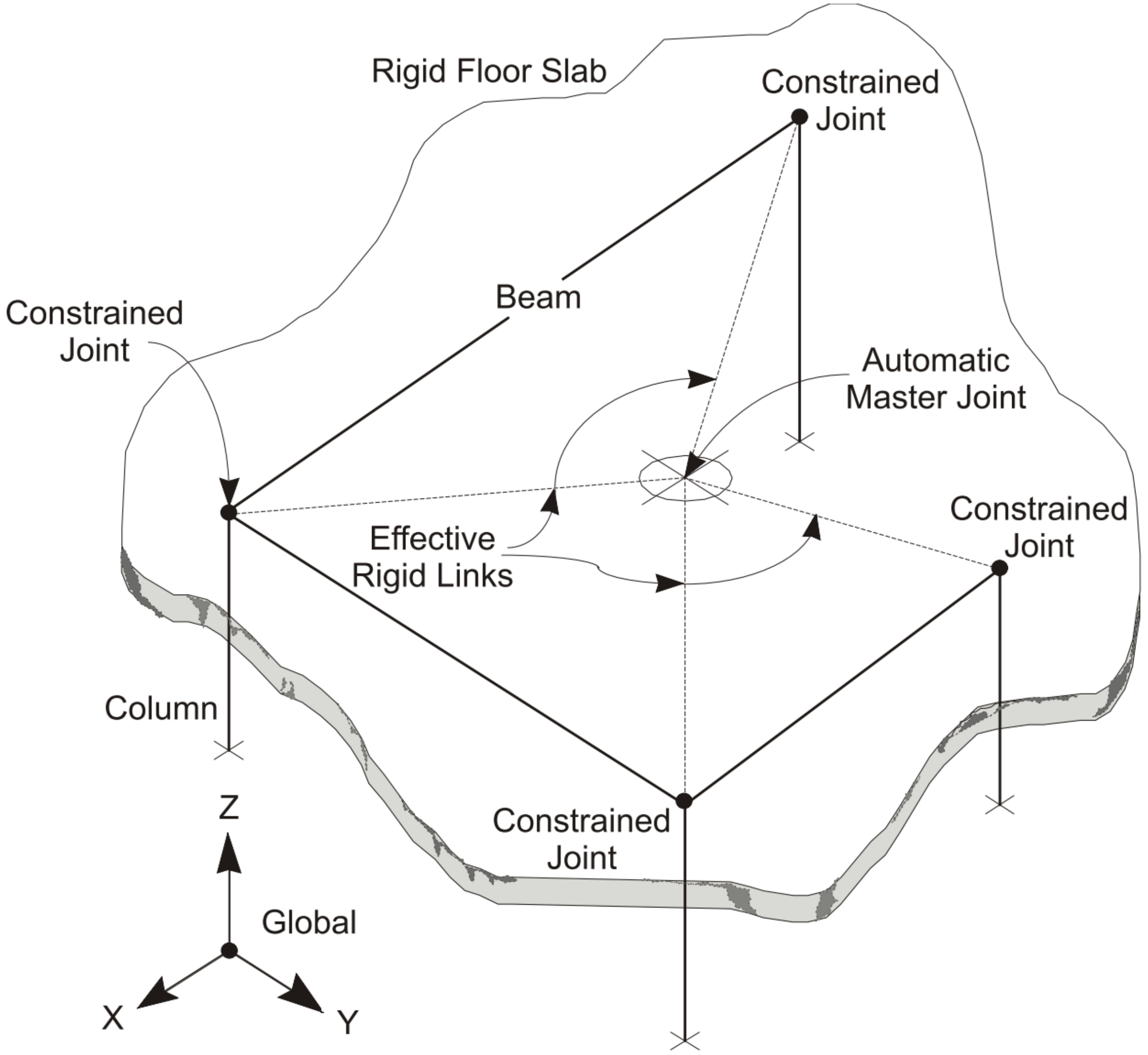
- A constraint consists of a set of two or more constrained joints. The displacements of each pair of joints in the constraint are related by constraint equations. The types of behavior that can be enforced by constraints are:
  - **Rigid-body behavior,** in which the constrained joints translate and rotate together as if connected by rigid links. The types of rigid behavior that can be modeled are:
    - – Rigid Body: fully rigid for all displacements
    - – Rigid Diaphragm: rigid for membrane behavior in a plane
    - – Rigid Plate: rigid for plate bending in a plane
    - – Rigid Rod: rigid for extension along an axis
    - – Rigid Beam: rigid for beam bending on an axis
  - **Equal-displacement behavior,** in which the translations and rotations are equal at the constrained joints
  - **Symmetry and anti-symmetry conditions**

Diaphragm Constraint

- A **Diaphragm Constraint** causes all of its constrained joints to move together as a planar diaphragm that is **rigid against membrane deformation**. Effectively, **all constrained joints are connected to each other by links that are rigid in the plane**, but do not affect out-of-plane (plate) deformation.
- This Constraint can be used to:
  - Model concrete floors (or concrete-filled decks) in building structures, which typically have very high in-plane stiffness
  - Model diaphragms in bridge superstructures
- The use of the Diaphragm Constraint for building structures eliminates the numerical accuracy problems created when the large in-plane stiffness of a floor diaphragm is modeled with membrane elements. It is also very useful in the lateral (horizontal) dynamic analysis of buildings, as it results in a significant reduction in the size of the **eigenvalue problem** to be solved.

Diaphragm Constraint

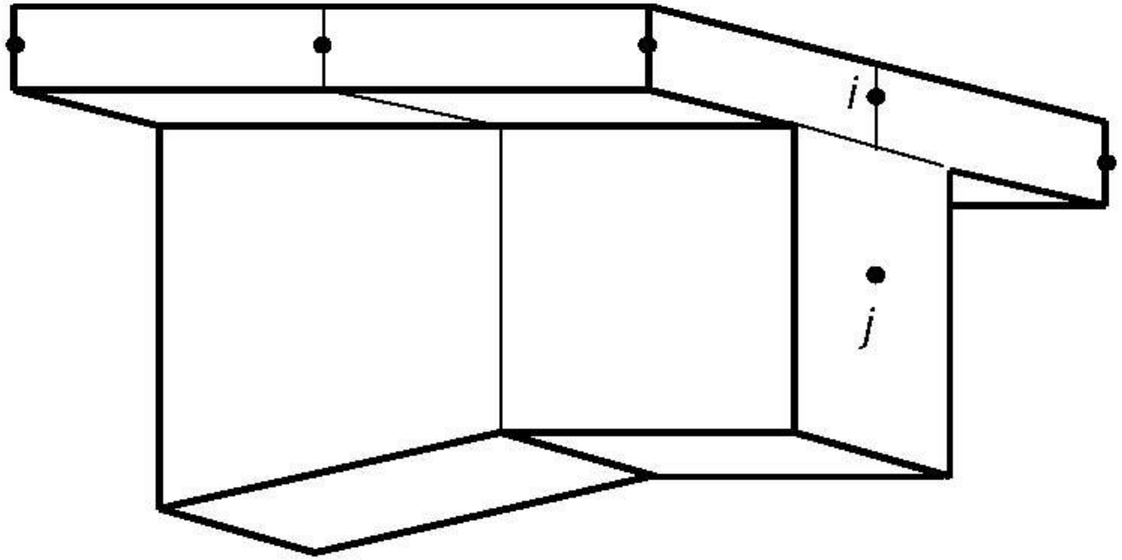
All constrained (slave) nodes are connected with an automatically created master node by rigid links.



Rigid and Semi-Rigid Floor Models

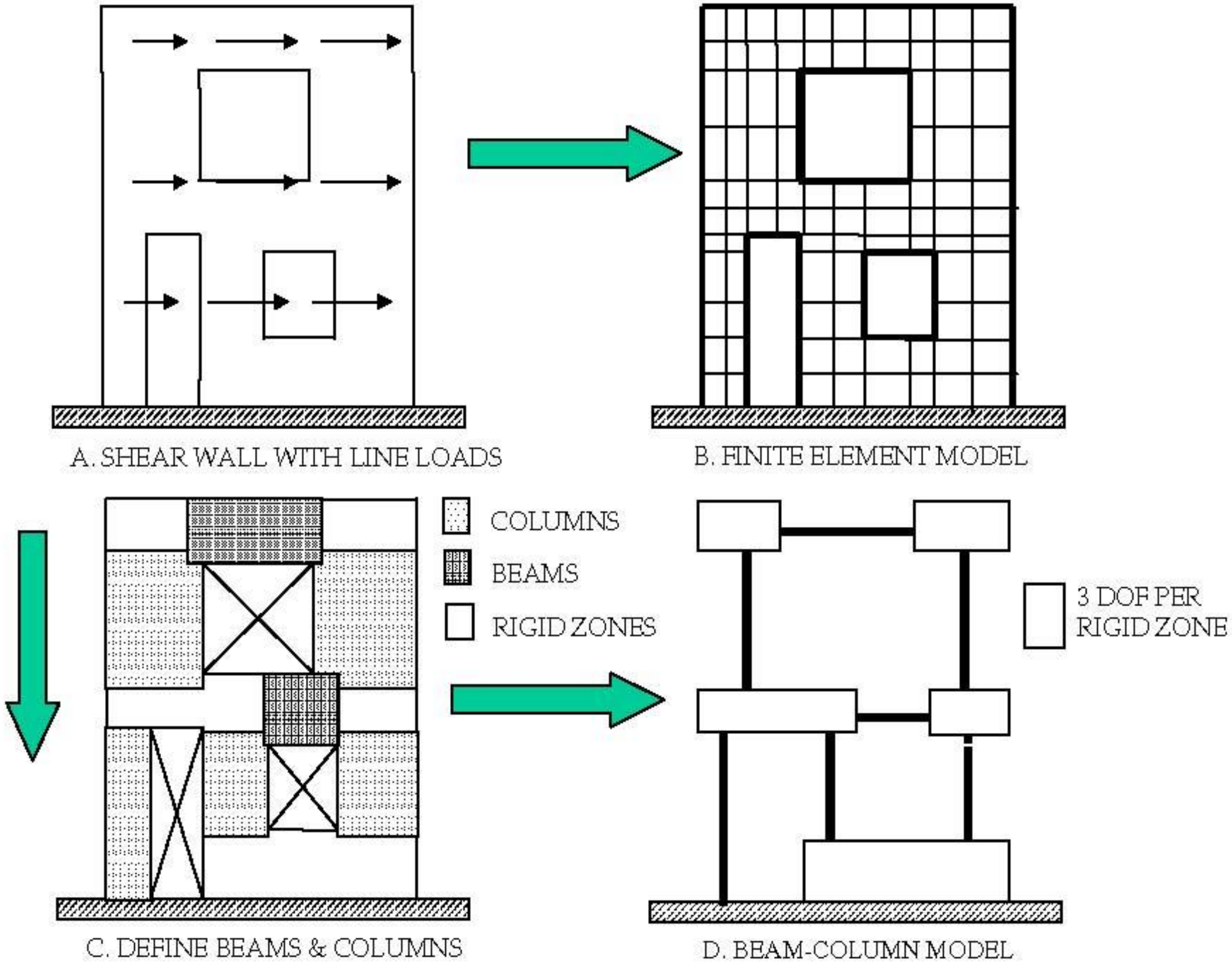
- Rigid diaphragm
  a) Each floor plate is assumed to translate in plan and rotate about a vertical axis as a **rigid body**.
  b) There will be **no in-plane deformations** in the floor plate.
  c) The solution will **not** produce any information on the **diaphragm shear stresses** or recover any axial forces in horizontal members that lie in the plane of the floors.
  d) Automated lateral (seismic and wind) loads will **act at master joint** where the mass of whole diaphragm is **lumped**.
- Semi-rigid (flexible) diaphragm
  a) There will be in-plane deformations in the floor plate.
  b) The solution will produce diaphragm shear stresses or recover any axial forces in horizontal members that lie in the plane of the floors.
  c) Automated lateral (seismic and wind) loads will distribute over at all joints of floor and the mass of diaphragm is also distributed to all nodes.
- Under the influence of lateral loads, significant shear stresses can be generated in the floor systems, and thus it may be sometimes important that the floor plates be modeled as semi-rigid diaphragms.

Use of Constraints in Beam-shell Analysis



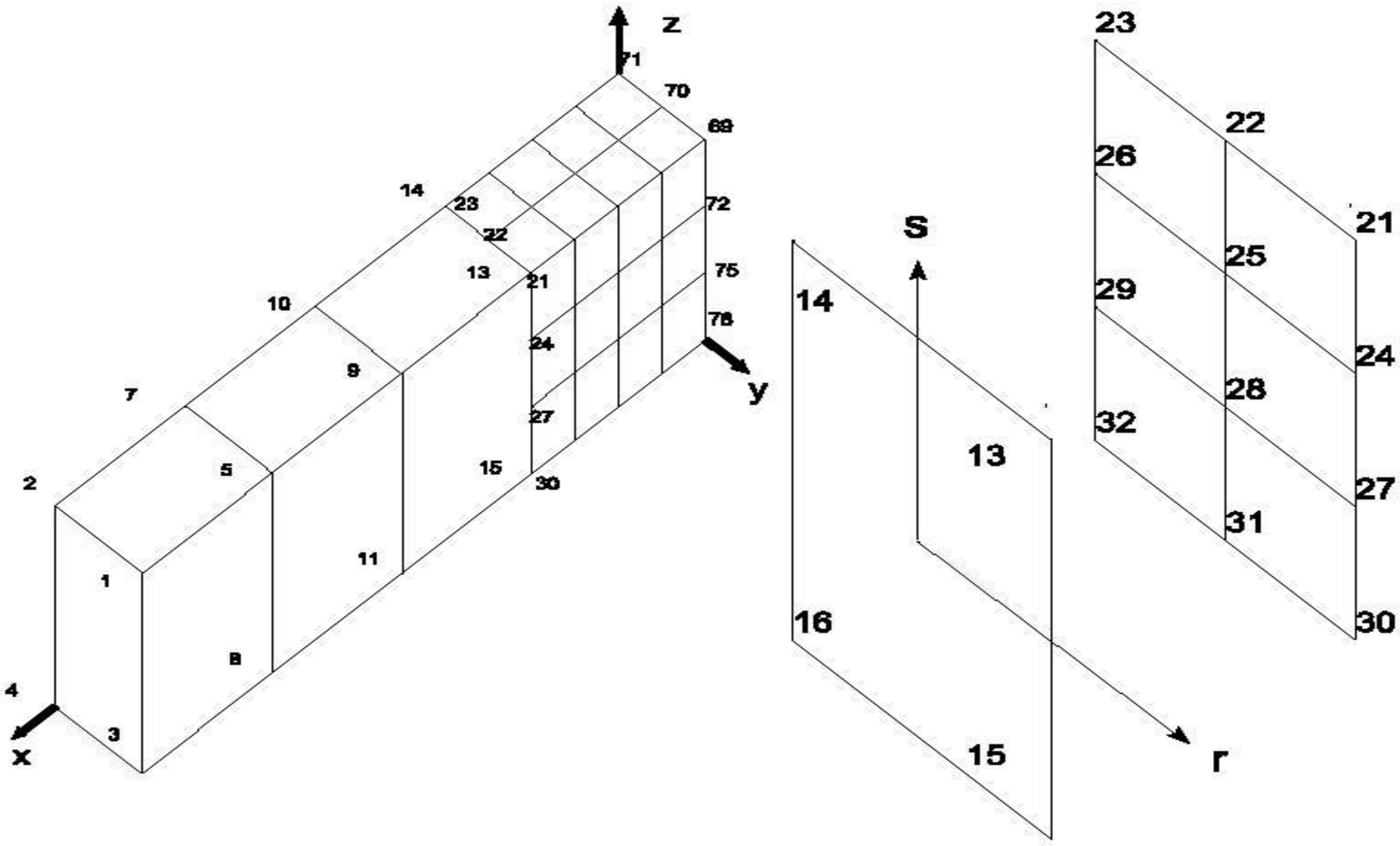
Connection of Beam to Slab by Constraints

Use of Constraints in Shear Wall Analysis



Beam-Column Model of Shear Wall

Use of Constraints for Mesh Transitions

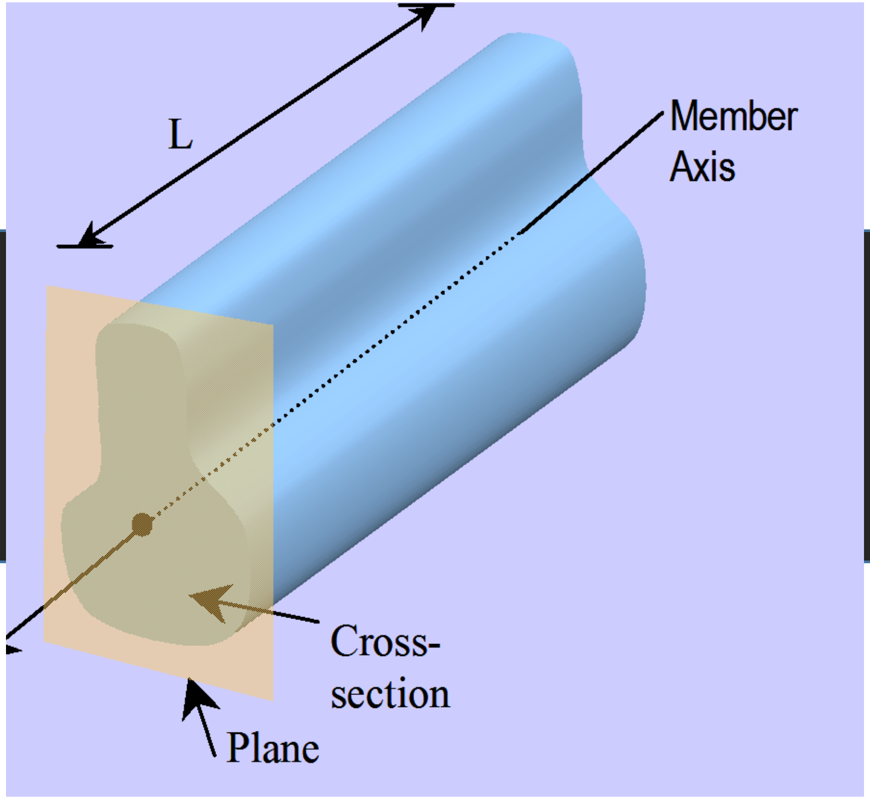


Use of Constraints to Merge Different Finite Element Meshes

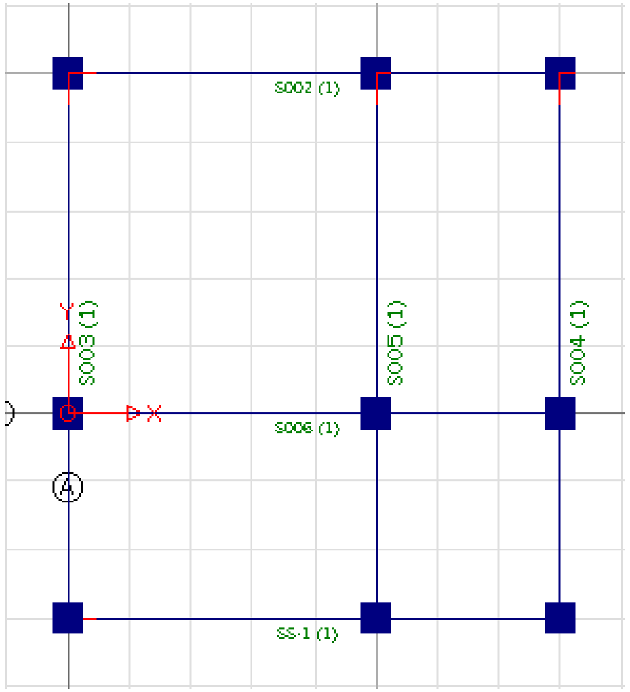
Constraint Outputs

- For each Body, Diaphragm, Plate, Rod, and Beam Constraint having more than two constrained joints, the following information about the Constraint and its master joint is printed in the output file:
  - The translational and rotational local coordinate systems for the master joint
  - The total mass and mass moments of inertia for the Constraint that have been applied to the master joint
  - The center of mass for each of the three translational masses
- The degrees of freedom are indicated as U1, U2, U3, R1, R2, and R3. These are referred to the local coordinate systems of the master joint.

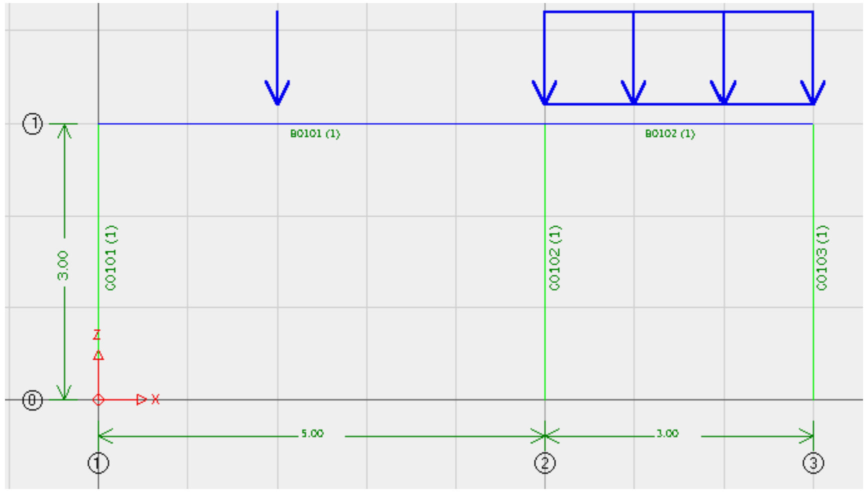
One Dimensional Frame Elements



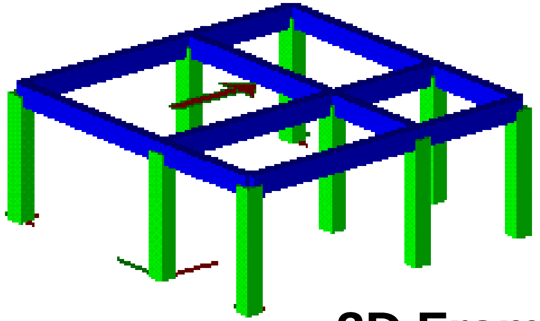
Usage of 1D Frame Elements



2D Grid

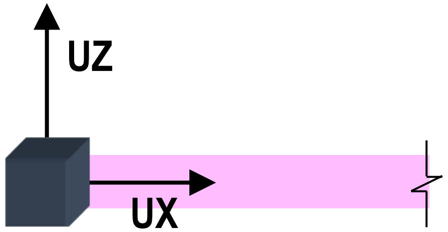


2D Frame

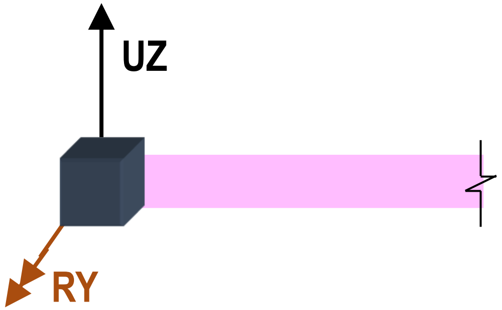


3D Frame

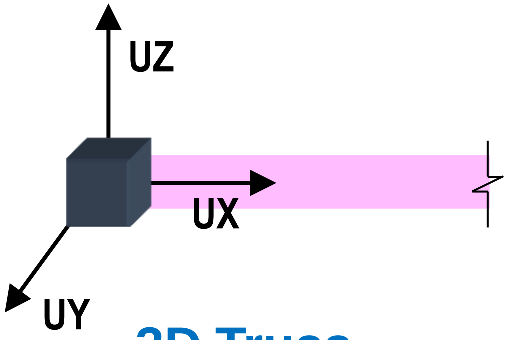
DOF for 1D Elements



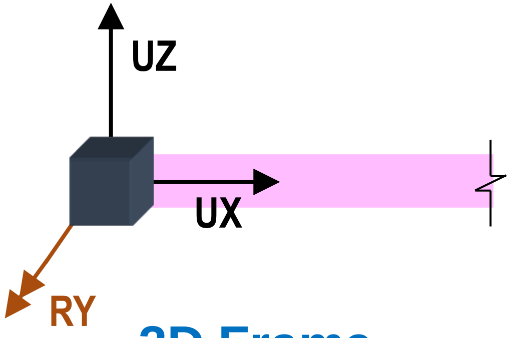
2D Truss



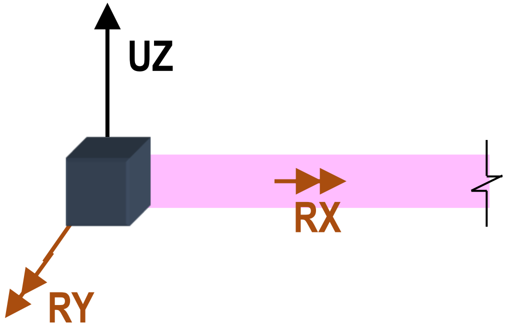
2D Beam



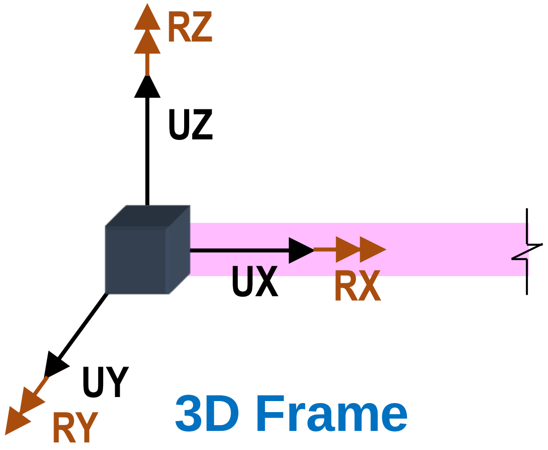
3D Truss



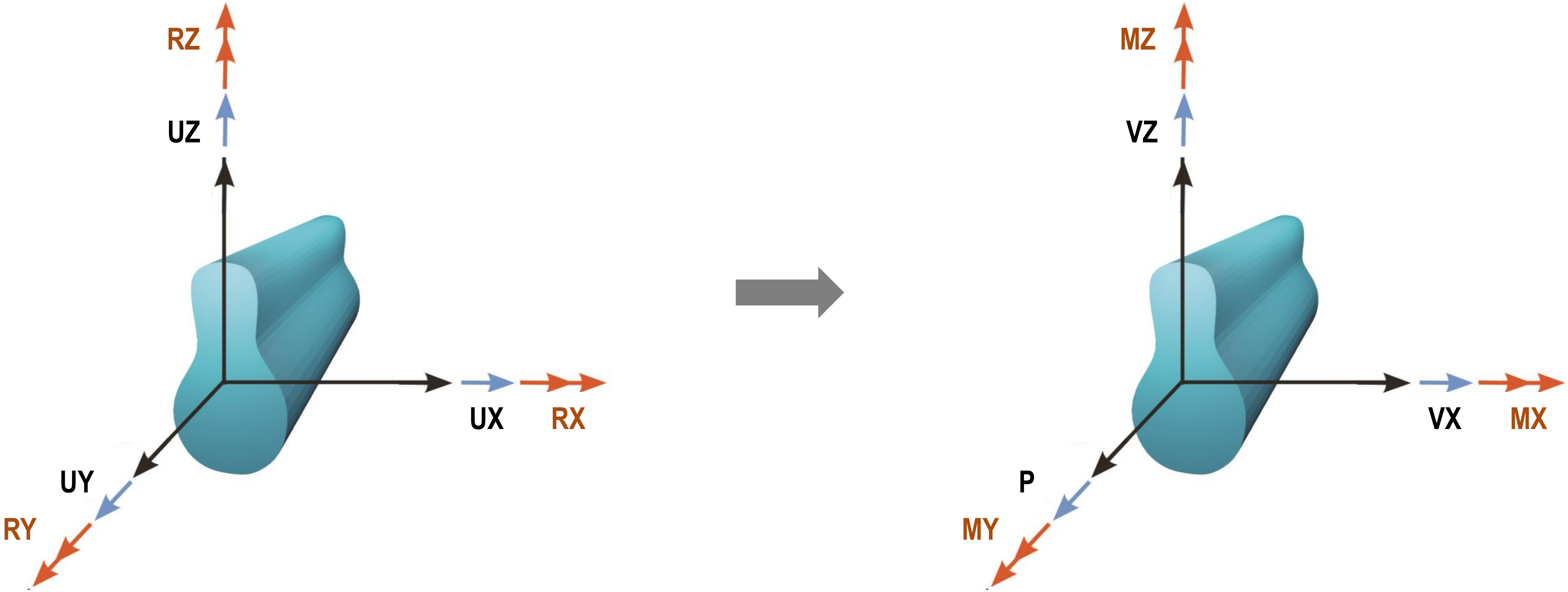
2D Frame



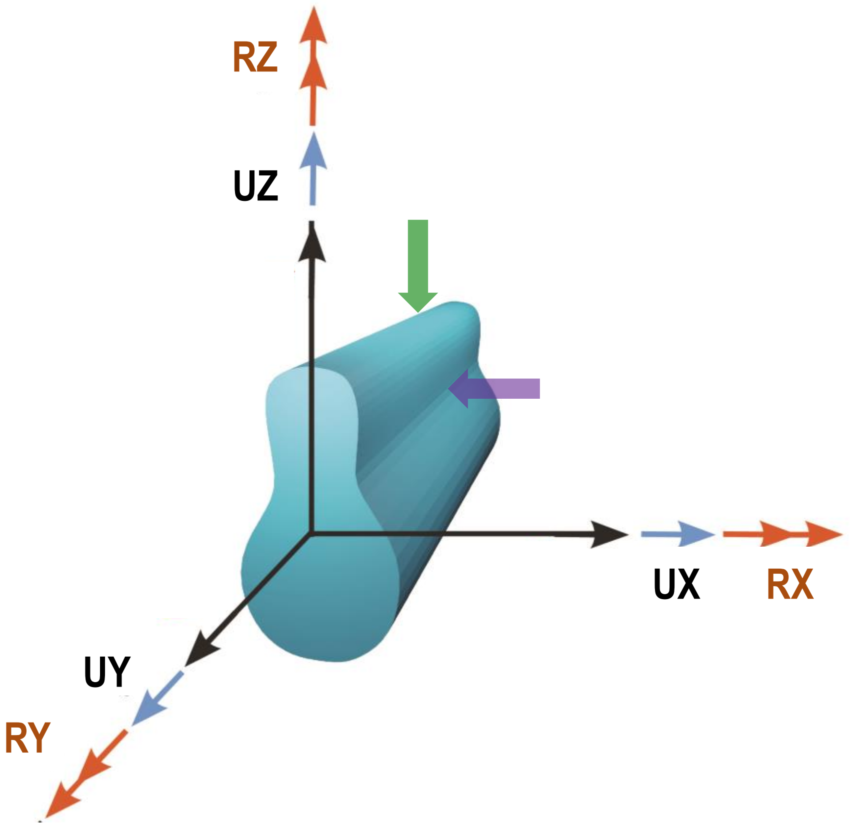
2D Grid



3D Frame


RZ
UZ
UX
RX
UY
RY
MZ
VZ
VX
MX
P
MY

DOFs and Corresponding Deformations (and Actions)



UY → Axial Deformation → Axial Strain → Axial Stress → Axial Force (P)

UZ → Shear Deformation → Shear Strain → Shear Stress → Shear Force (VZ)

UX → Shear Deformation → Shear Strain → Shear Stress → Shear Force (VX)

RY → Torsional Deformation (Twist) → Combined Shear and Axial Strains → Combined Shear and Axial Stress → Shear and Axial Forces caused by Torsion (T)

RX → Bending Deformation (Curvature) → Axial Strains (bending) → Axial Stresses (bending) → Bending Moment about X axis (MX)

May also produce lateral shear stresses and strains

Primary bending moment caused by vertical gravity loading ⬇

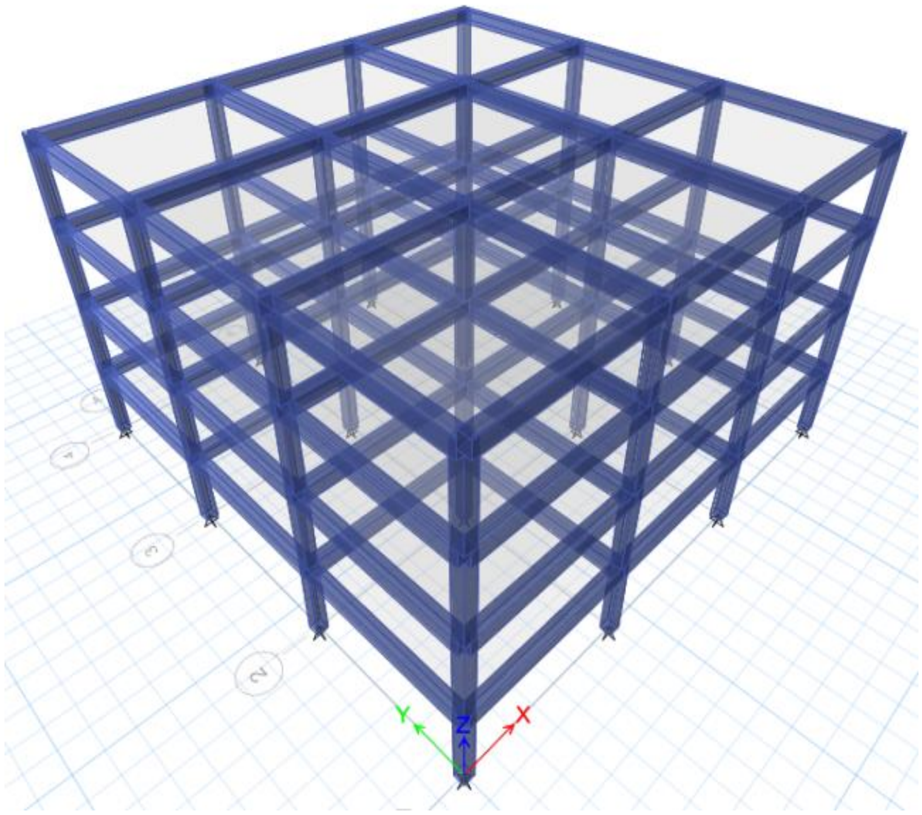
RZ → Bending Deformation (Curvature) → Axial Strains (bending) → Axial Stresses (bending) → Bending Moment about Z axis (MZ)

May also produce lateral shear stresses and strains

Primary bending moment caused by horizontal loading ⬅

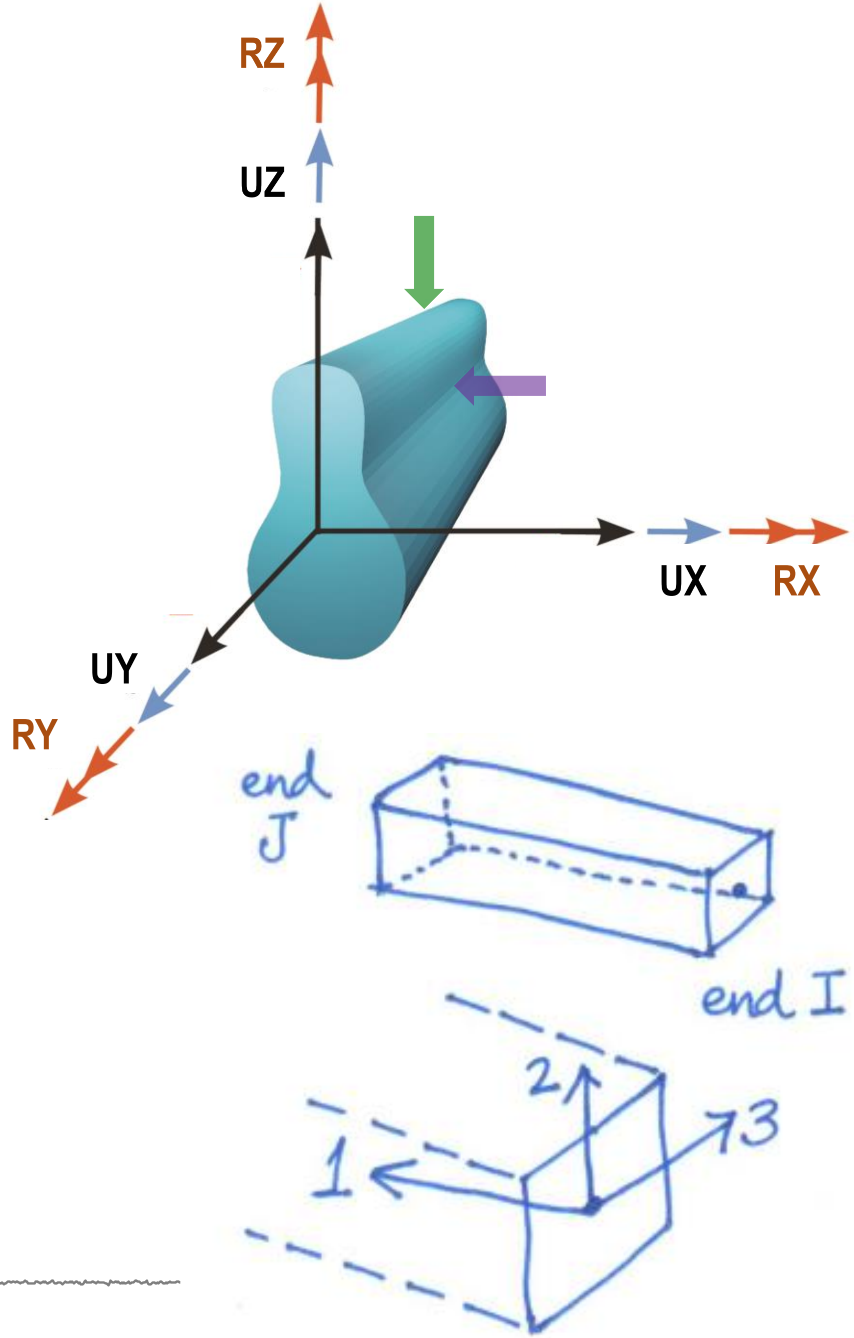
One-dimensional Elements in SAP2000/ETABS

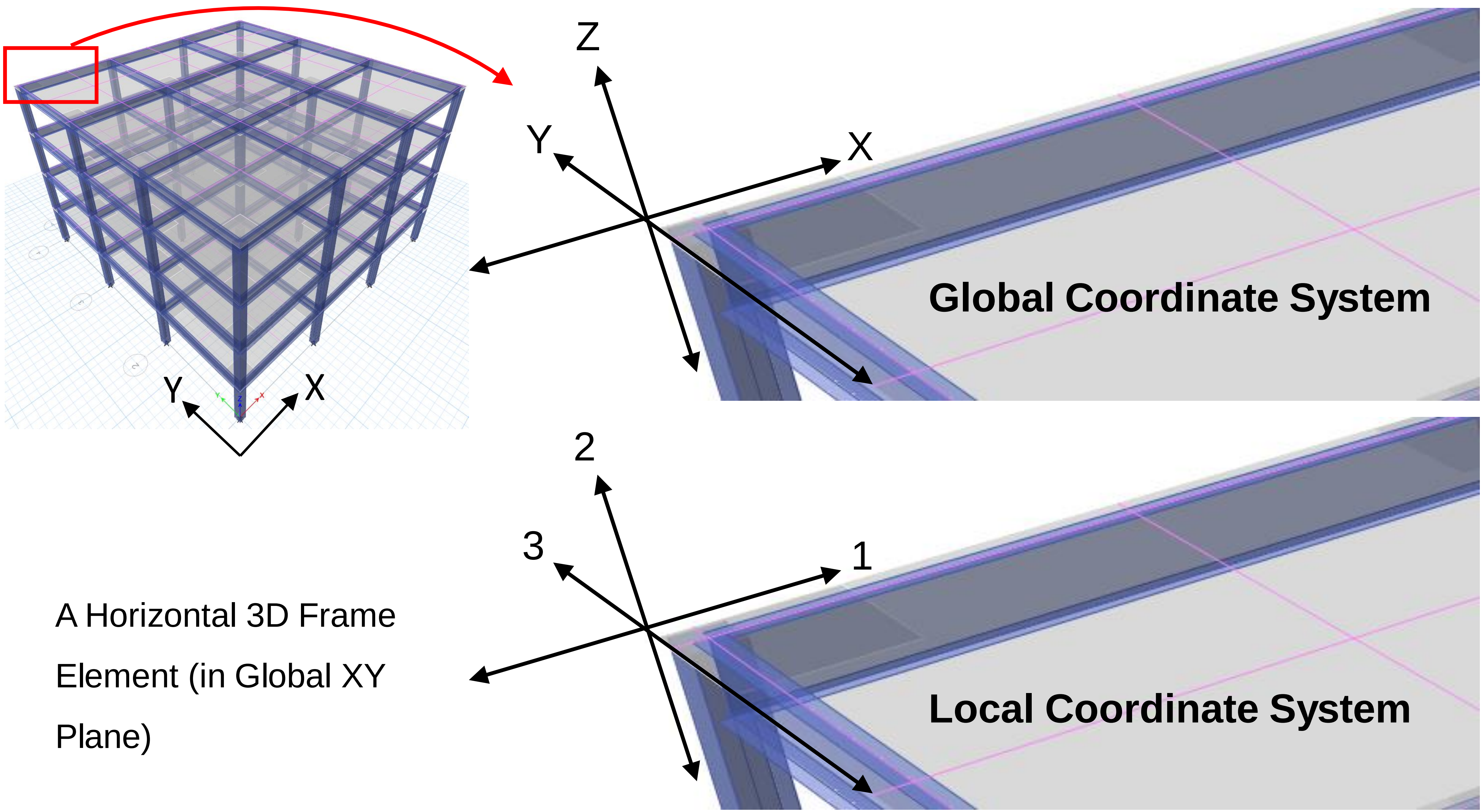
- Simple Frame Elements for
    - Beam, Column
    - Truss, Bracing, etc.
- Non-Linear Link Element for
    - Hook, Gap, Damper
    - Base Isolators
    - Friction
- Plastic Hinge Element



Frame Local Coordinate System in SAP/ETABS

- The two ends of the element are denoted end I and end J, respectively.
- **The local axis 1 is always the longitudinal axis of the element**, the positive direction being directed from End I to End J. This axis is always located at the centroid of the cross section, and connects joint I to joint j.
- The default orientation of the local 2 and 3 axes is determined by the relationship between the local 1 axis and the global Z axis.
- The local 2 axis is taken to have an up ward (+Z) sense unless the element is vertical, in which case the local 2 axis is taken to be horizontal along the global +X direction.
- The local 3 axis is horizontal, i.e., it lies in the X-Y plane. This means that the local 2 axis points vertically up ward for horizontal elements.

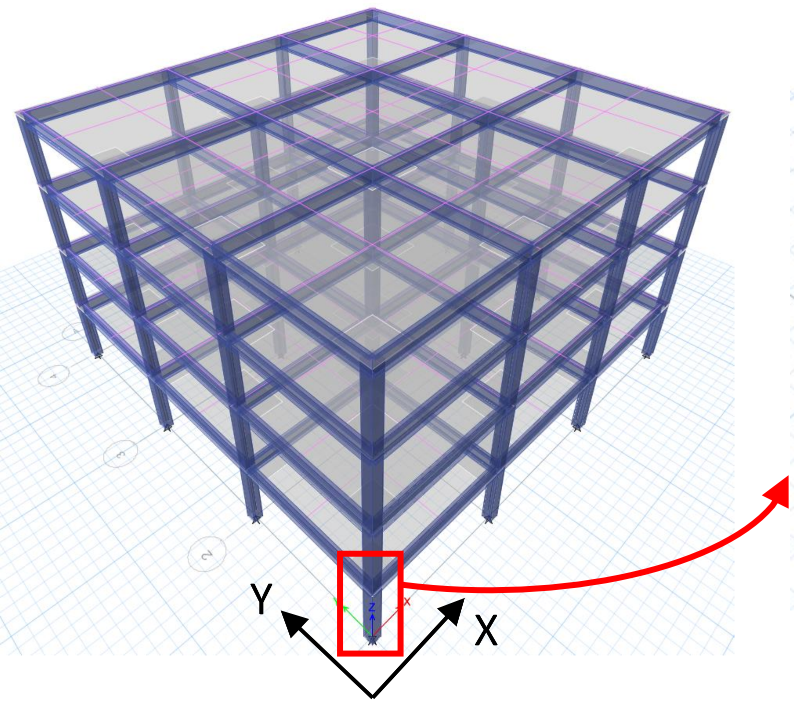


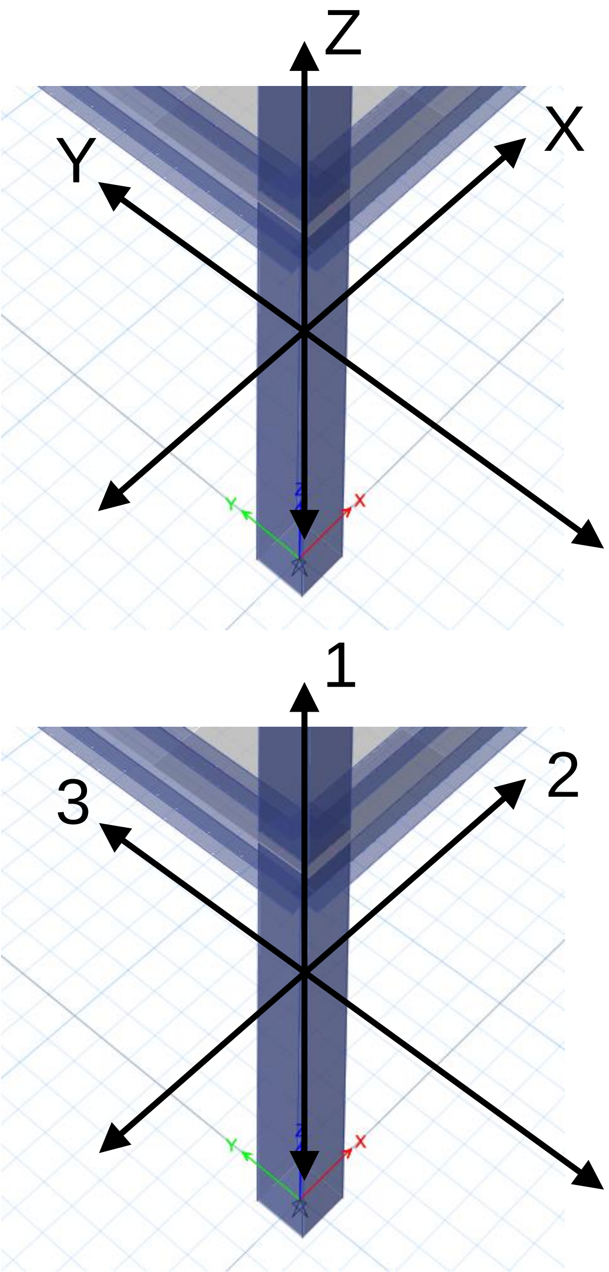


Global Coordinate System

Local Coordinate System

A Horizontal 3D Frame Element (in Global XY Plane)

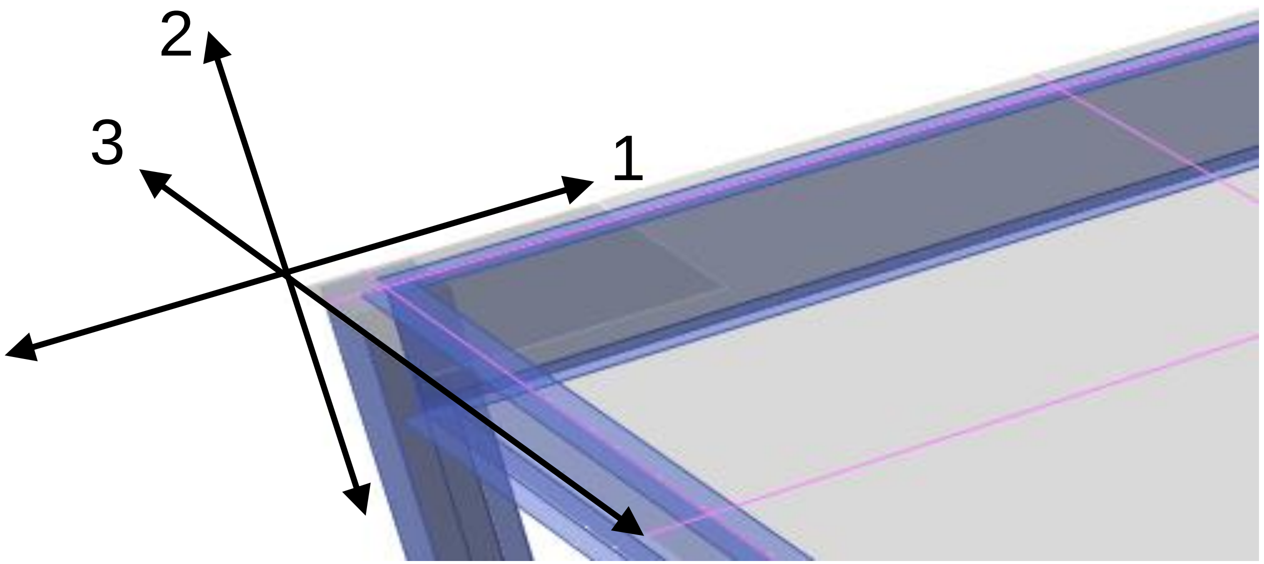




Global Coordinate System

Local Coordinate System

A Vertical 3D Frame Element (in Global XZ Plane)



U1 → P

U2 → V2

U3 → V3

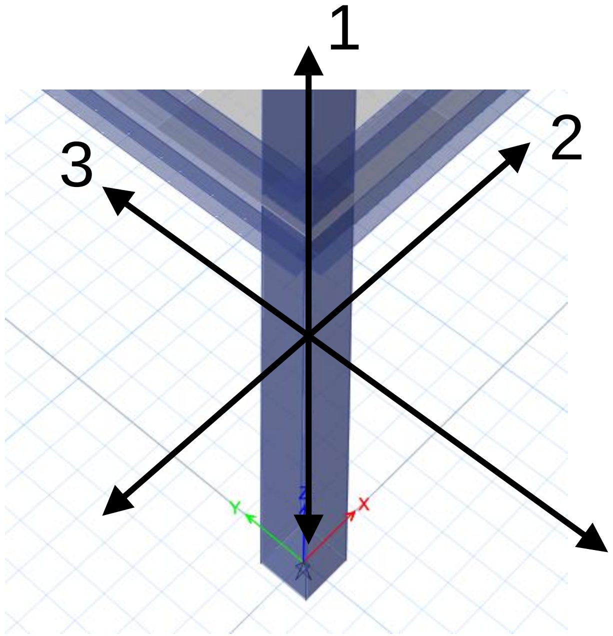
R1 → T

R2 → M2

R3 → M3

Not all combinations are important in all the practical cases !!!

M3	→ A beam subjected to pure bending moment caused by gravity load
P, M3	→ A beam subjected to axial load and bending caused by gravity load
V2	→ A beam subjected to pure shear force caused by gravity load
P, V2, M3	→ A beam subjected to axial load, shear and bending caused by gravity load
T	→ A beam subjected to pure torsion
M3, V2, P, T	→ A beam subjected to bending moment and shear caused by gravity load as well as axial load and torsion



P → A (perfect) concentrically loaded column

P, M3 → A column under uniaxial bending. A column subjected to axial load and bending moment about axis 3. This bending moment can be transferred to this column from beam connecting in axis 2 direction.

P, M2 → A column under uniaxial bending. A column subjected to axial load and bending moment about axis 2. This bending moment can be transferred to this column from beam connecting in axis 3 direction.

P, M2, M3 → A column under biaxial bending. A column subjected to axial load and two bending moments (i.e. both about axis 2 and axis 3). These bending moment can be transferred to this column from beams connecting in axis 2 and axis 3 directions.

U1 → P
U2 → V2
U3 → V3
R1 → T
R2 → M2
R3 → M3

The shear in beam along axis 2 direction (due to gravity load) will be transferred to this column as axial force P. Any axial force in this beam (along axis 2) will be transferred to this column as shear V2. Similarly, the shear in beam along axis 3 direction (due to gravity load) will be transferred to this column as axial load P. Any axial force in this beam (along axis 3) will be transferred to this column as shear V3.

P, M3, V2, M2, V3 → A column subjected to biaxial bending as well as biaxial shear.

P, M3, V2, M2, V3, T → The most unlucky column, subjected to everything, i.e. biaxial bending, biaxial shear as well as torsion.

Cross-section Stiffness and Cross-section Properties

- The action along each degree of freedom is related to the corresponding deformation by the member stiffness, which in turn, depends on the cross-section stiffness.
- So there is a particular cross-section property corresponding to member stiffness for each degree of freedom. Therefore, for the six degrees of freedom defined earlier, the related cross-section properties are:

UY ⇨ Cross-section area, A

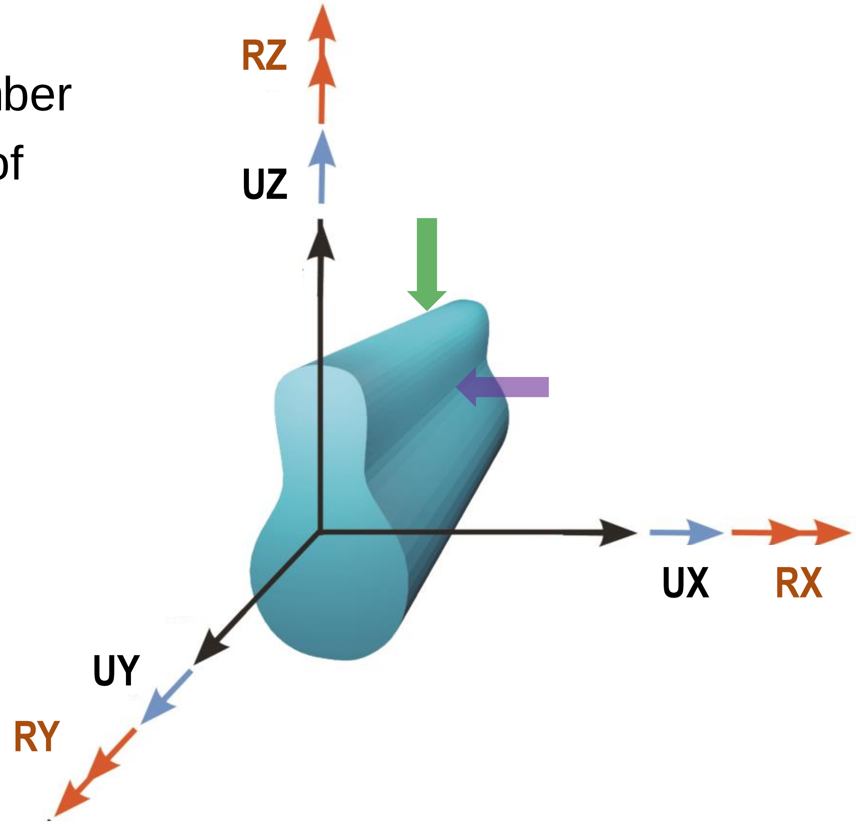
UX ⇨ Shear Area along x, SA_X

UZ ⇨ Shear Area along y, SA_Z

RY ⇨ Torsional Constant, J

RX ⇨ Moment of Inertia, I_X

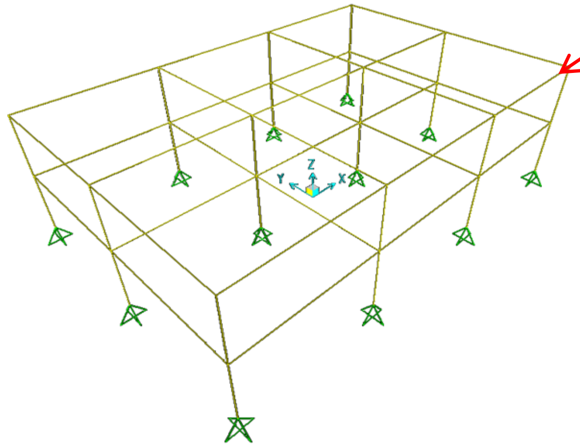
RZ ⇨ Moment of Inertia, I_Z



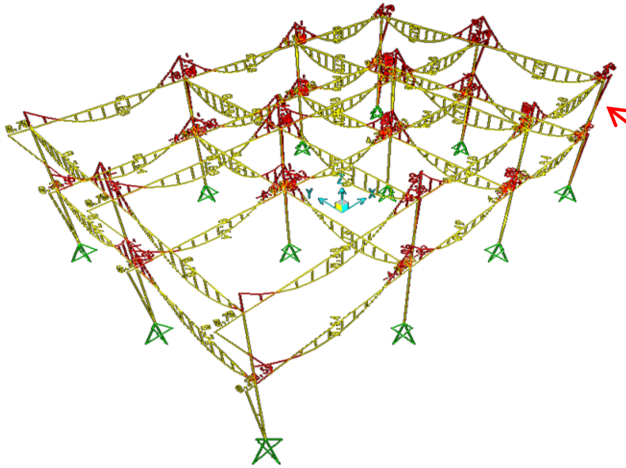
Section Properties

- A **Frame Section** is a set of material and geometric properties that describe the cross-section of one or more Frame elements.
- Sections are defined independently of the Frame elements, and are **assigned** to the elements.
- Section properties are of two basic types:
  - Prismatic — all properties are constant along the full element length
  - Non-prismatic — the properties may vary along the element length

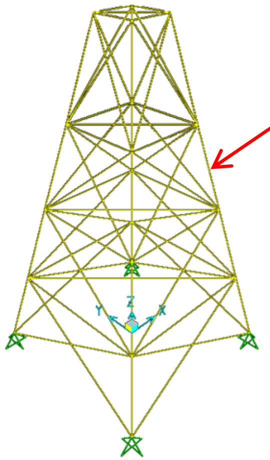
The Frame Elements



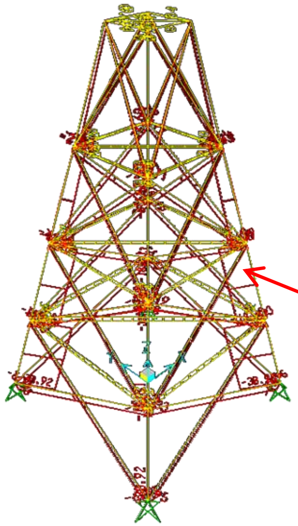
Frame Elements in FE Model



Element Forces

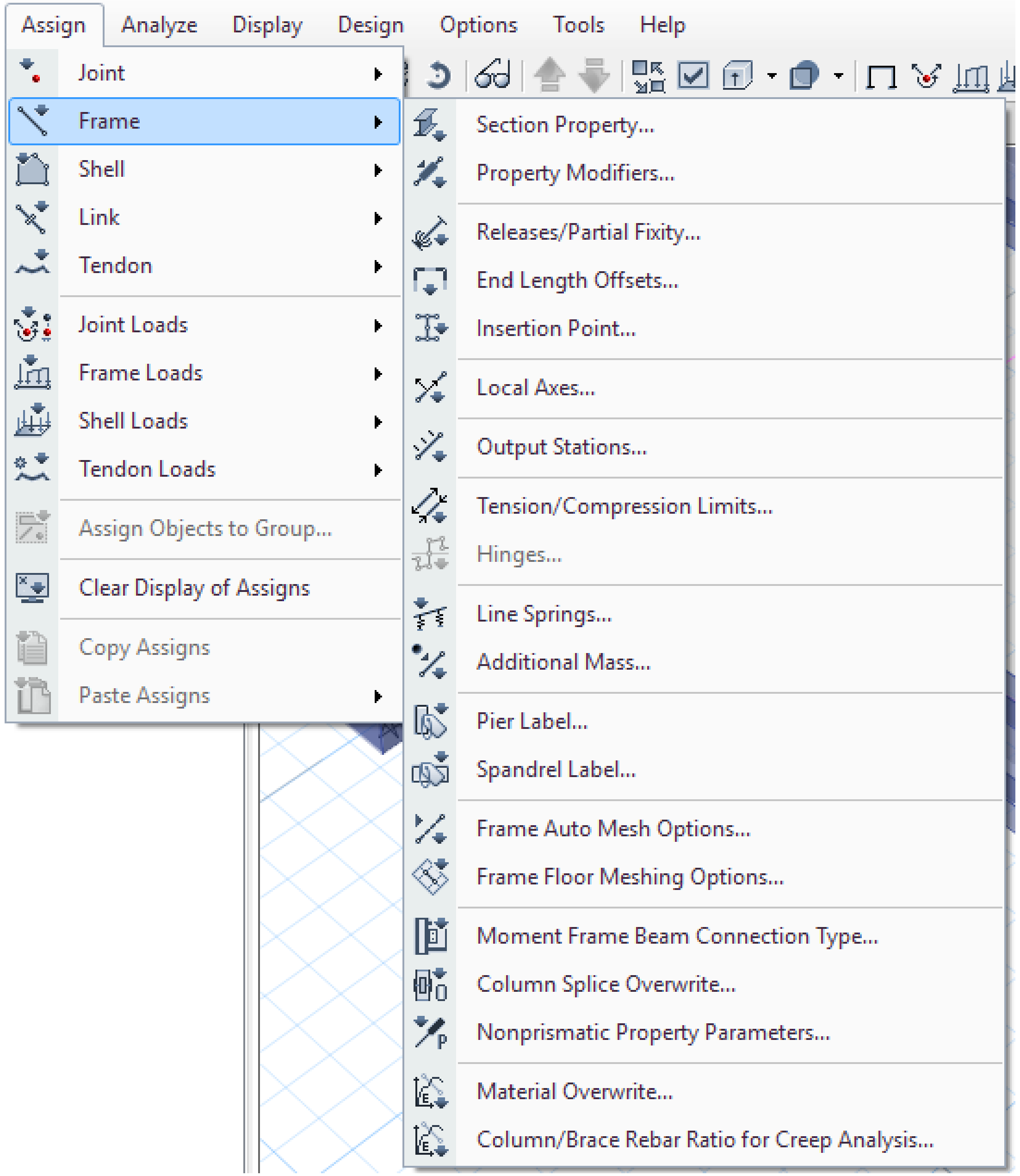


Frame Elements in FE Model



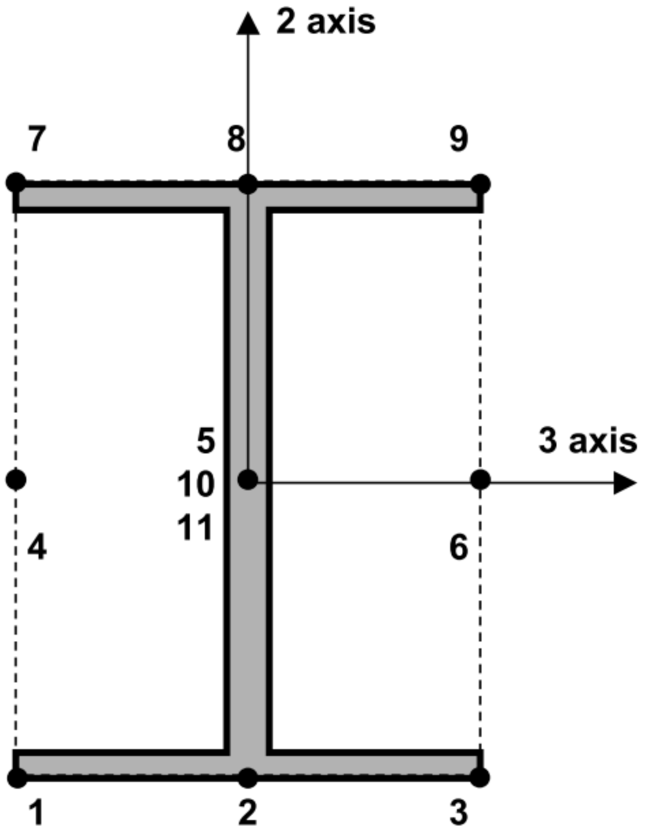
Element Forces

Frame Assignments



Insertion Points

- The local 1 axis of the element runs along the neutral axis of the section, i.e., at the centroid of the section. By default this connects to the joints I and j at the ends of the element.
- However, it is often convenient to specify an other location on the section, such as the top of a beam or an outside corner of a column, to connect to the joints.
- There is a set of **pre-defined locations** within the section, called **cardinal points**, that can be used for this purpose.
- The available choices are shown in Figure. The default location is point 10, the centroid.

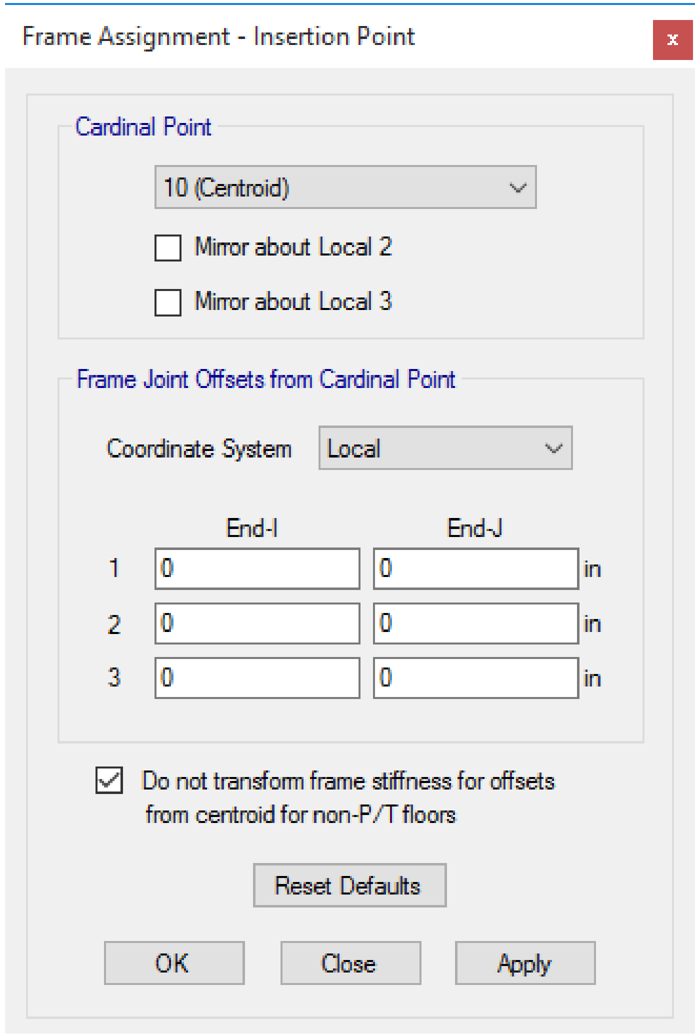


1. Bottom left
2. Bottom center
3. Bottom right
4. Middle left
5. Middle center
6. Middle right
7. Top left
8. Top center
9. Top right
10. Centroid
11. Shear center

Note: For doubly symmetric members such as this one, cardinal points 5, 10, and 11 are the same.

Insertion Points

- You can further offset the cardinal point from the joint by specifying **joint offsets**.
- The joint offsets together with the cardinal point make up the insertion point assignment. The total offset from the joint to the centroid is given as the sum of the joint offset plus the distance from the cardinal point to the centroid.
- This feature is useful, as an example, for modeling beams and columns when the beams do not frame into the center of the column.



Frame Assignment - Insertion Point

Cardinal Point

10 (Centroid)

☐ Mirror about Local 2

☐ Mirror about Local 3

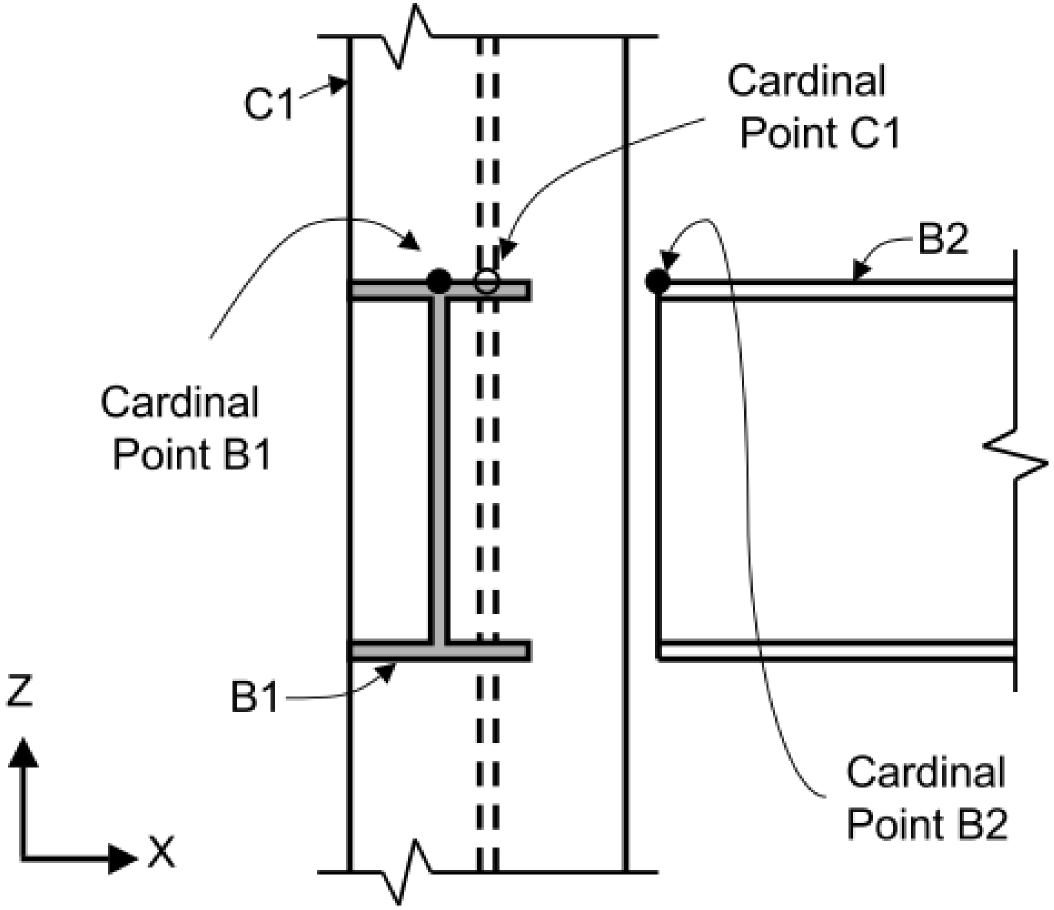
Frame Joint Offsets from Cardinal Point

Coordinate System: Local

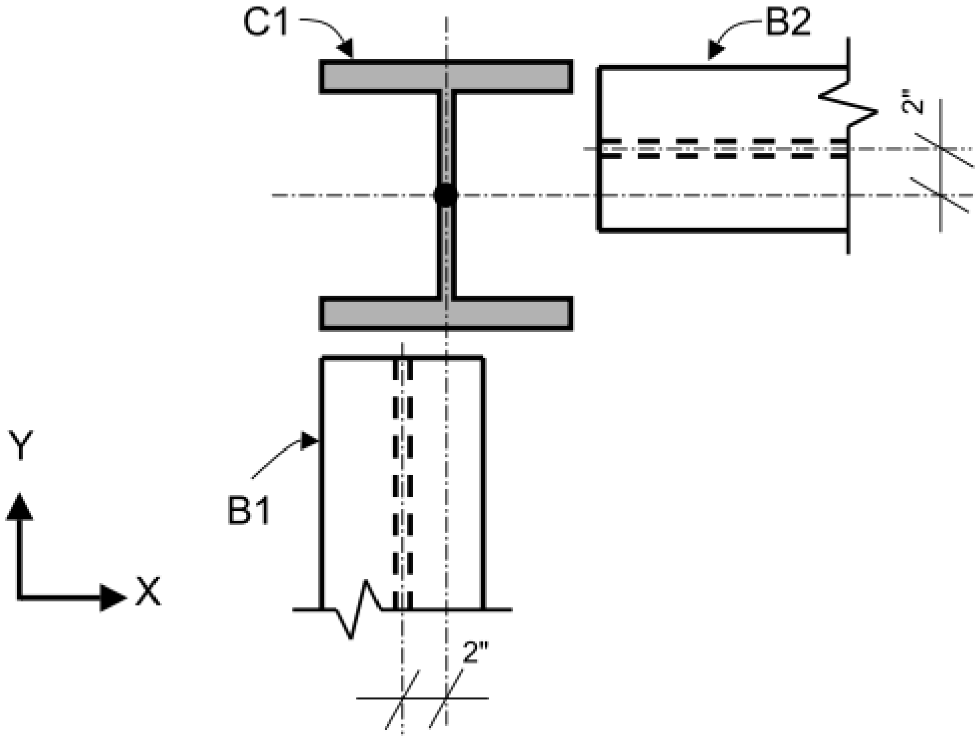
	End-I	End-J	
1	0	0	in
2	0	0	in
3	0	0	in

☑ Do not transform frame stiffness for offsets from centroid for non-P/T floors

Insertion Points and Joint Offsets



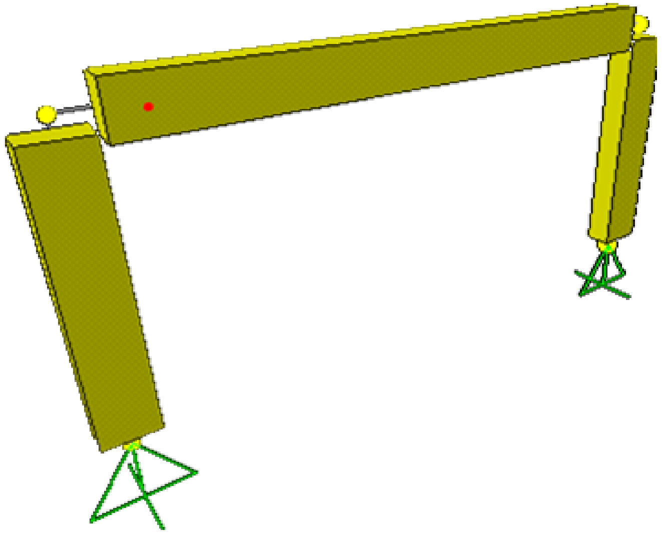
Elevation

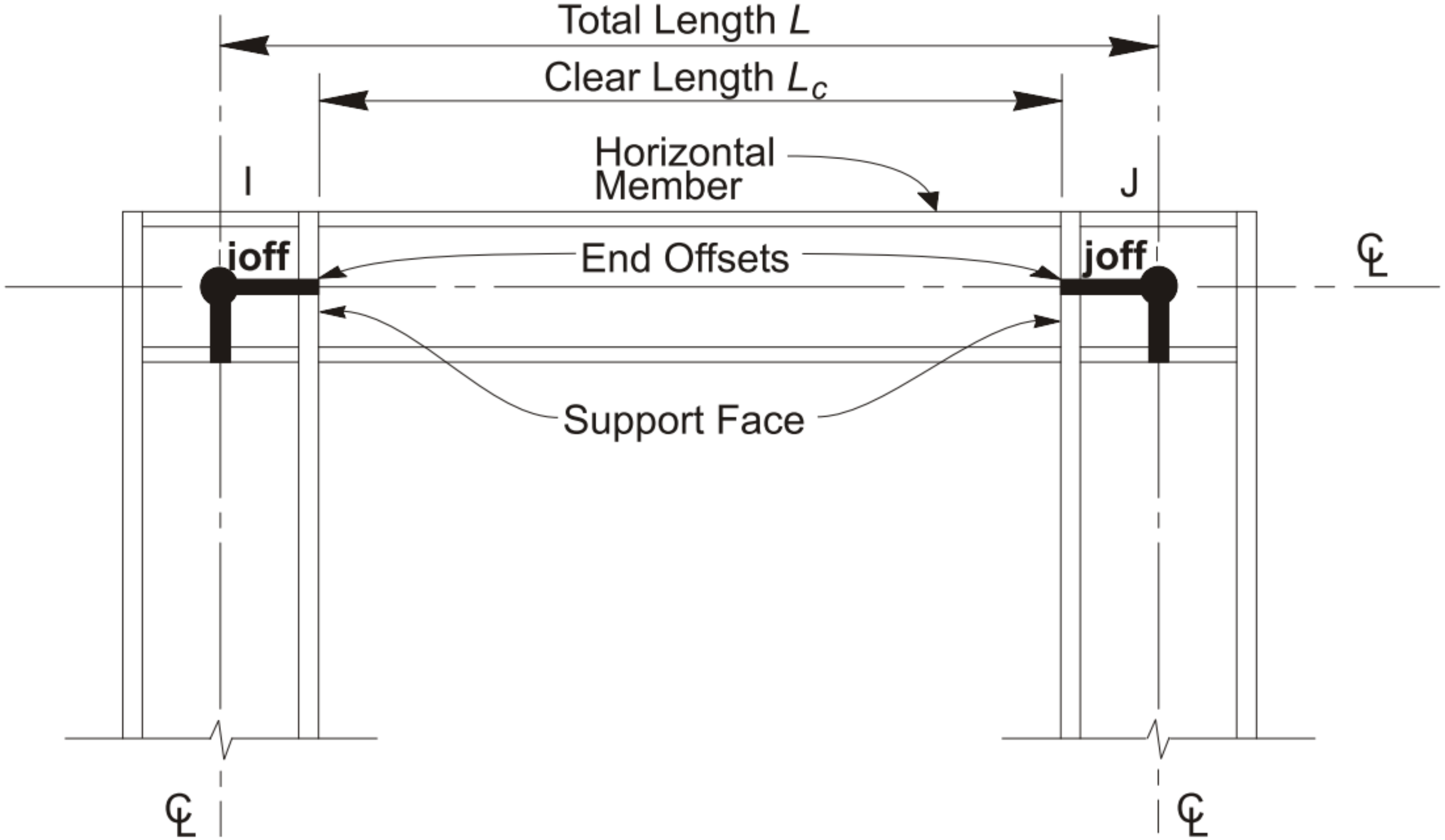


Plan

Rigid End Offsets

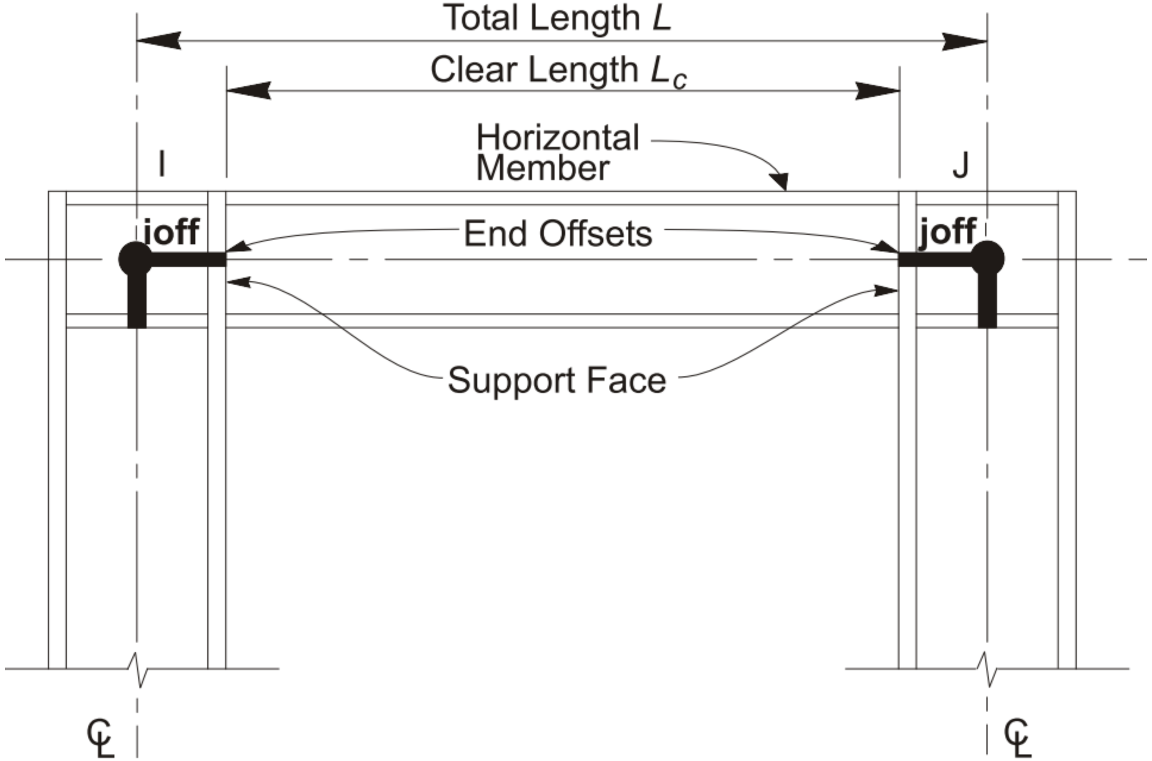
- Rigid End connections to model large joints
- Automated end offset evaluation and assignment





Rigid-End Factor

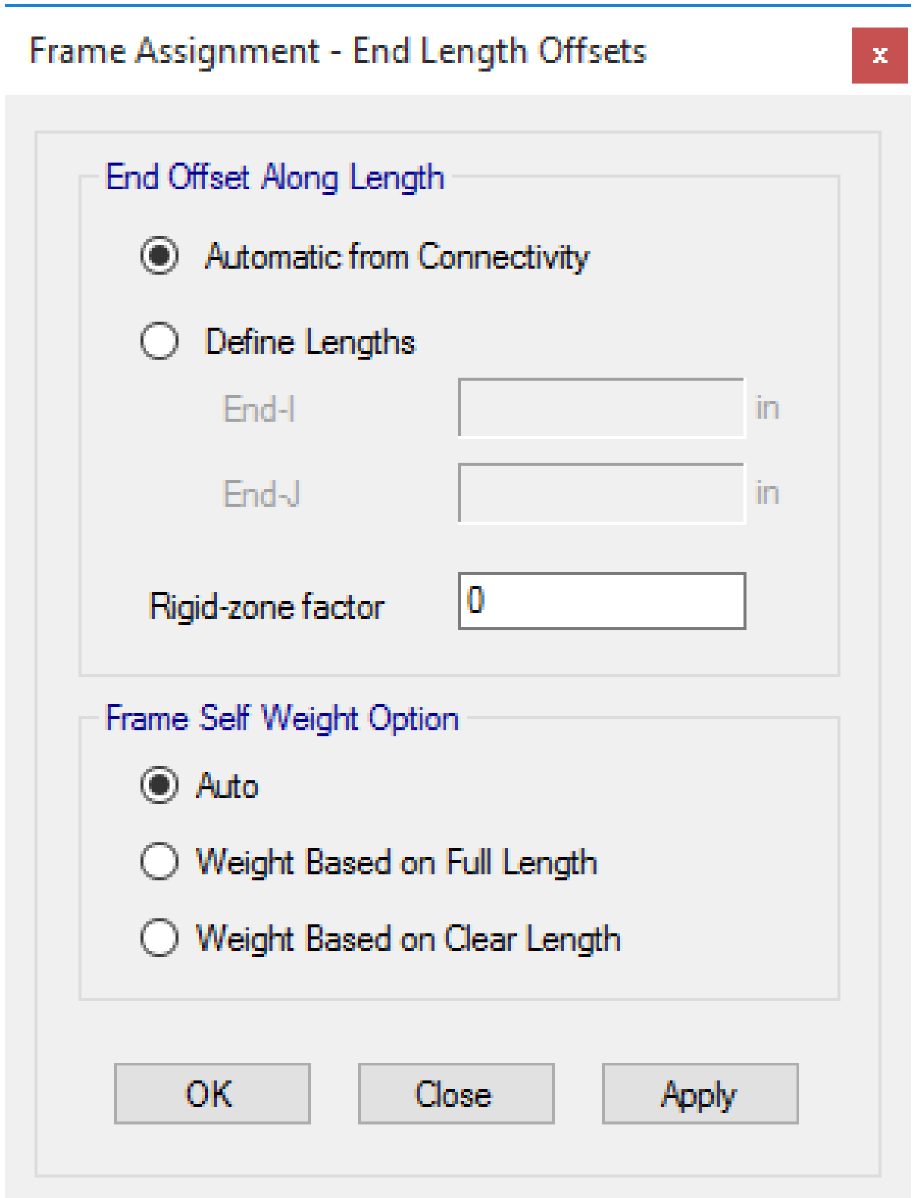
- An analysis based upon the centerline-to-centerline (joint-to-joint) geometry of Frame elements may overestimate deflections in some structures. This is due to the stiffening effect caused by overlapping cross sections at a connection. It is more likely to be significant in concrete than in steel structures.
- You may specify a **rigid-end factor** for each element using parameter rigid, which gives the fraction of each end off set that is assumed to be rigid for bending and shear deformation.
- The length rigid zone factor × ioff, starting from joint I, is assumed to be rigid. Similarly, the length rigid zone factor × joff is rigid at joint j.



The flexible length of the element is = **Total Length – rigid zone factor (ioff + joff)**

Rigid End Factor

- The default value for rigid is zero. The maximum value of unity would indicate that the end offsets are fully rigid.
- You must use engineering judgment to select the appropriate value for this parameter. It will depend upon the geometry of the connection, and may be different for the different elements that frame into the connection.
- Typically the value for rigid would not exceed about 0.5.
- **For concrete frames, 0.5 is recommended.**
- The rigid-zone offsets never affect axial and torsional deformation. The full element length is assumed to be flexible for these deformations.



Frame Assignment - End Length Offsets

End Offset Along Length

● Automatic from Connectivity

○ Define Lengths

End-I in

End-J in

Rigid-zone factor 0

Frame Self Weight Option

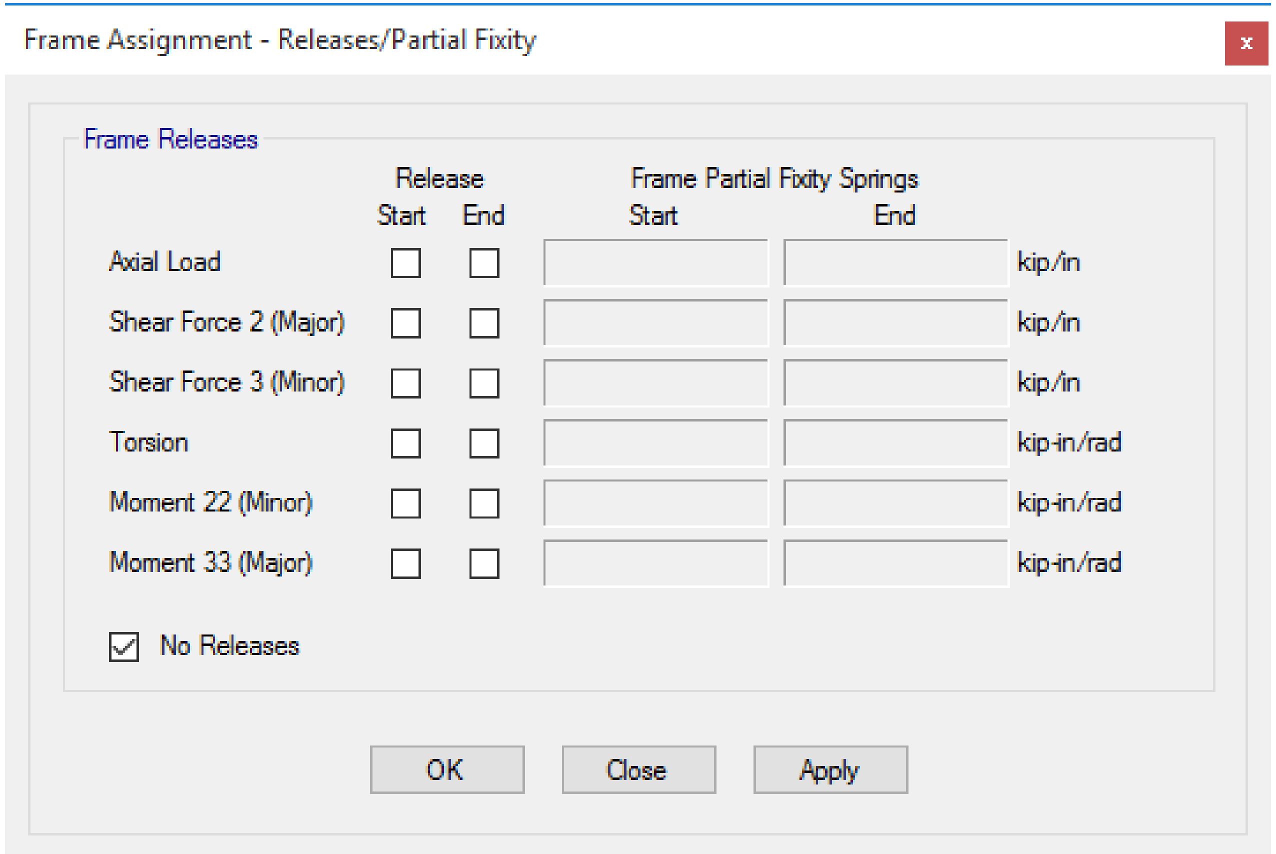
● Auto

○ Weight Based on Full Length

○ Weight Based on Clear Length

End Releases

- **Release:** Removing the capacity of a frame end to resist any of P, M3, M2, V3, V2 and T.
- Any or all of the six actions can be released or partially fixed.



Frame Assignment - Releases/Partial Fixity

Frame Releases

	Release		Frame Partial Fixity Springs		
	Start	End	Start	End	
Axial Load	☐	☐			kip/in
Shear Force 2 (Major)	☐	☐			kip/in
Shear Force 3 (Minor)	☐	☐			kip/in
Torsion	☐	☐			kip-in/rad
Moment 22 (Minor)	☐	☐			kip-in/rad
Moment 33 (Major)	☐	☐			kip-in/rad

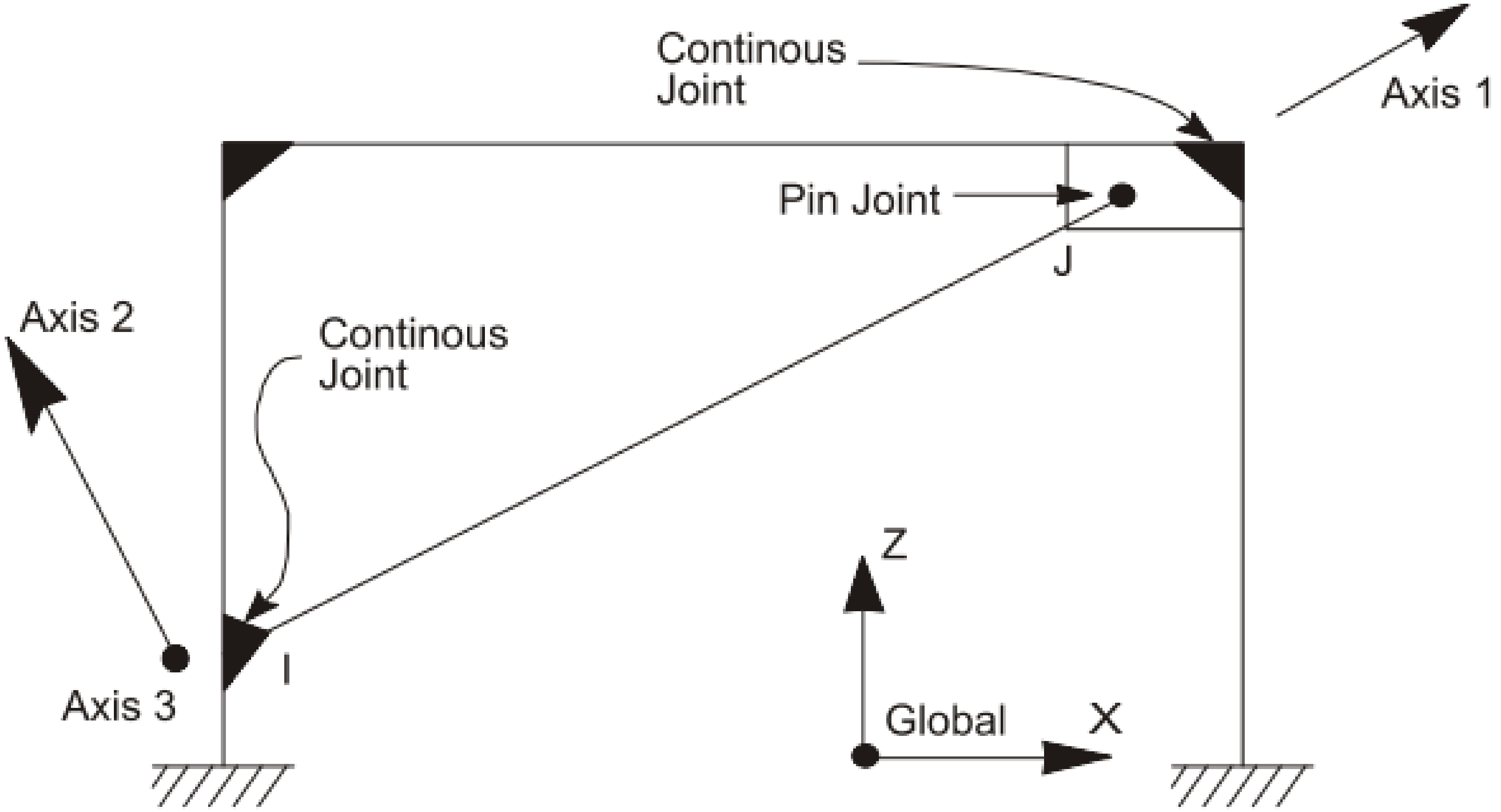
☑ No Releases

For Example, a frame element with M3, M2 and T releases at both ends

=

A 3D Truss Element

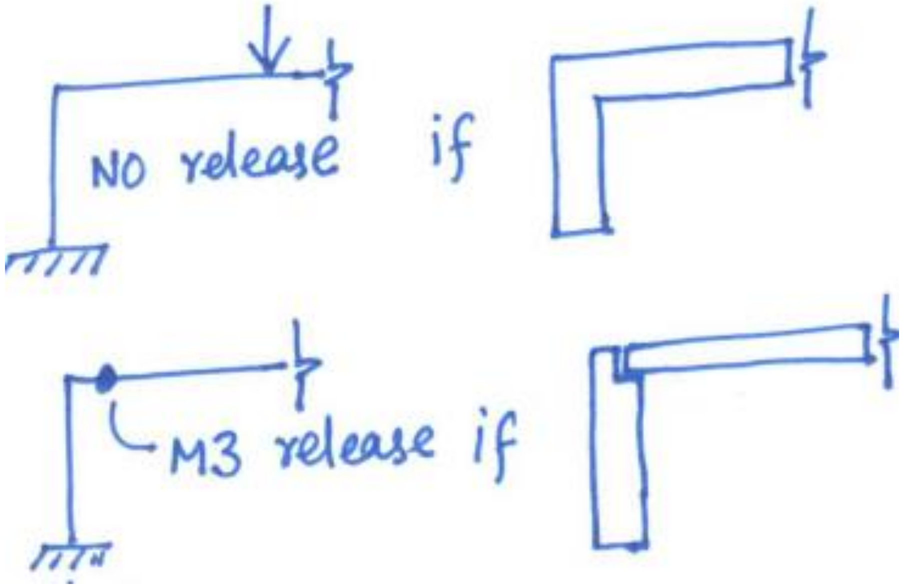
End Releases to model pin joints in diagonal elements (struts)



For diagonal element: R3 is released at end J

End Releases in frames used to model RC Beams

- For example, an M3 release in a beam will act like an internal hinge. The end will not transfer moments to connecting element.



- In RC beams, apart from geometry, reinforcement also determines whether the end condition should be released or partially released.

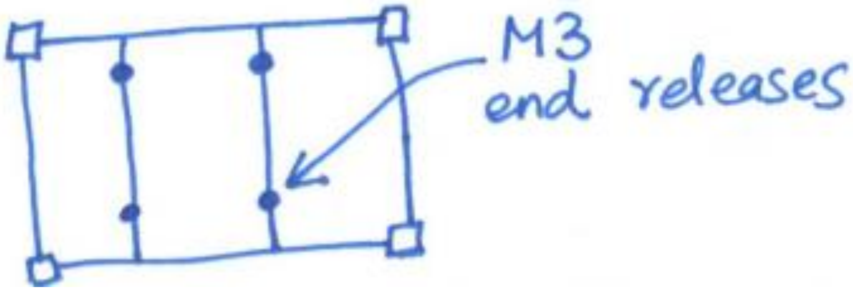
Cases where we can use M3 release

a) Intentionally not transferring the end moment. For example, to reduce reinforcement → Model like no resistance to moment.

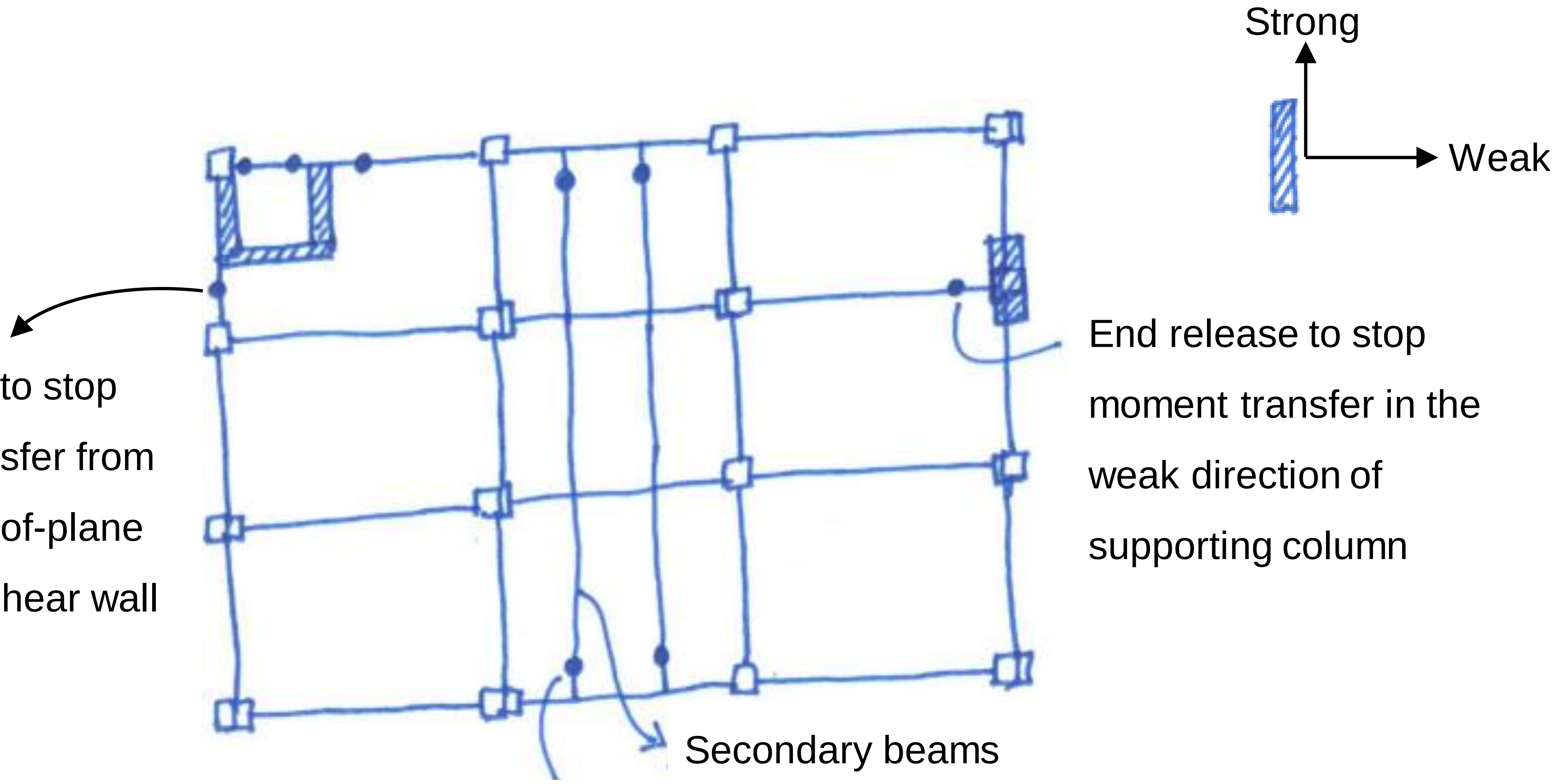
b) Not enough development length available to rebars to behave like a rigid connection.

c) Column is so thin and unable to carry moments coming from beam.

d) On secondary beams supported on main beams (girders). This will avoid the introduction of torsion → Just simply supported. If there is a series of supported beams, then one release at start and one at the end is enough.



Cases where we can use M3 release



End release to stop moment transfer from beam to out-of-plane direction of shear wall leg

End release to stop moment transfer in the weak direction of supporting column

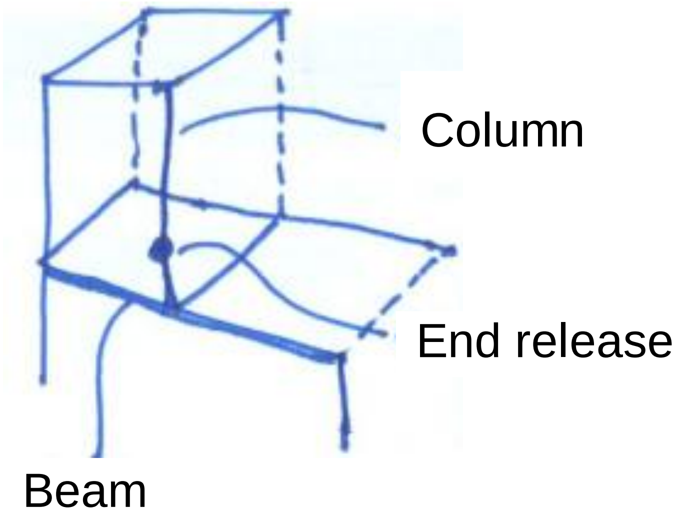
Secondary beams

Releases at only the start and end in whole series

Cases where we can use M3 release

e) On the nides of beams connecting to walls and columns on their weak axis (i.e. perpendicular to the longer dimension of leg or cross-section)

f) Columns supported on beams (to avid torsion in supporting beams)



End Releases in Frames

- Select Frame → Assign → Frame → Releases/Partial Fixity

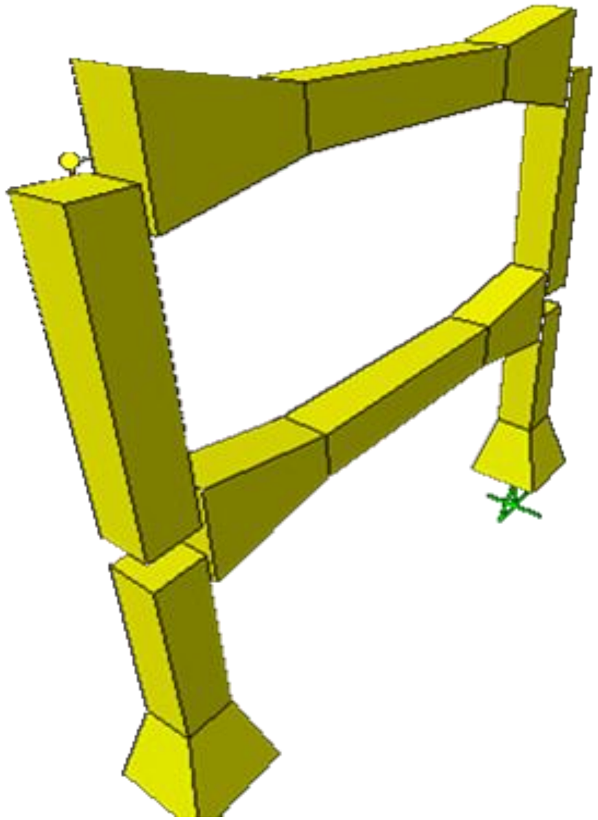
Completely remove the capacity of a node to resist a particular action

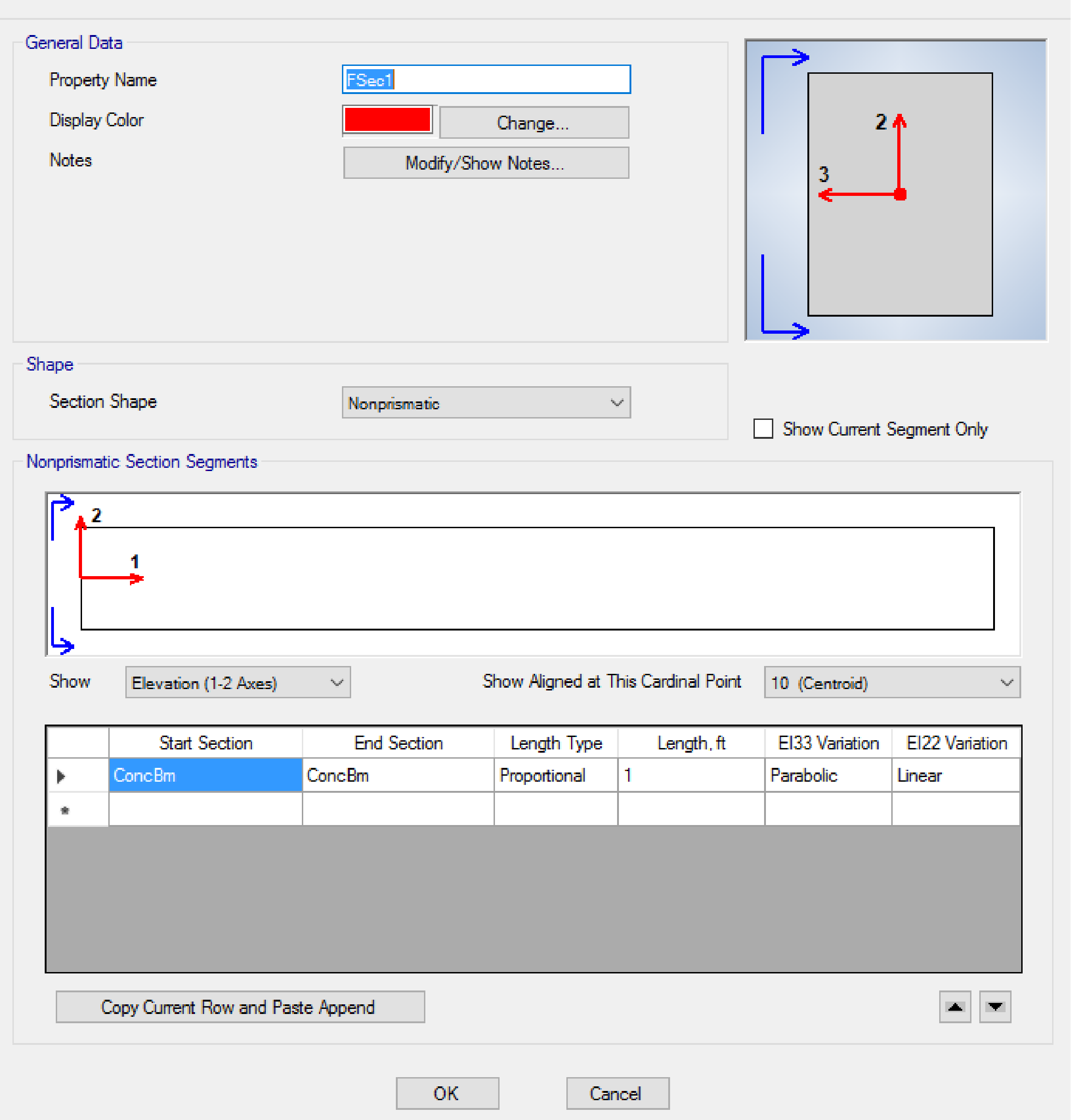
Provide your own value of stiffness coefficients to resist a particular action (i.e. corresponding to any degree of freedom)

- Wrongly applied releases → **Structural Stability**
- The following sets of releases are unstable, either alone or in combination, and are not permitted.
  - Releasing U1 at both ends;
  - Releasing U2 at both ends;
  - Releasing U3 at both ends;
  - Releasing R1 at both ends;
  - Releasing R2 at both ends and U3 at either end;
  - Releasing R3 at both ends and U2 at either end.

Non-prismatic Frame Elements

- Multiple non-prismatic segments over element length to model beams of variable sections





General Data

Property Name: FSec1

Display Color: 🟥

Shape

Section Shape: Nonprismatic

☐ Show Current Segment Only

Nonprismatic Section Segments

Show: Elevation (1-2 Axes)

Show Aligned at This Cardinal Point: 10 (Centroid)

	Start Section	End Section	Length Type	Length, ft	EI33 Variation	EI22 Variation
▸	ConcBm	ConcBm	Proportional	1	Parabolic	Linear
*						

Nonlinear Properties of Frame Elements

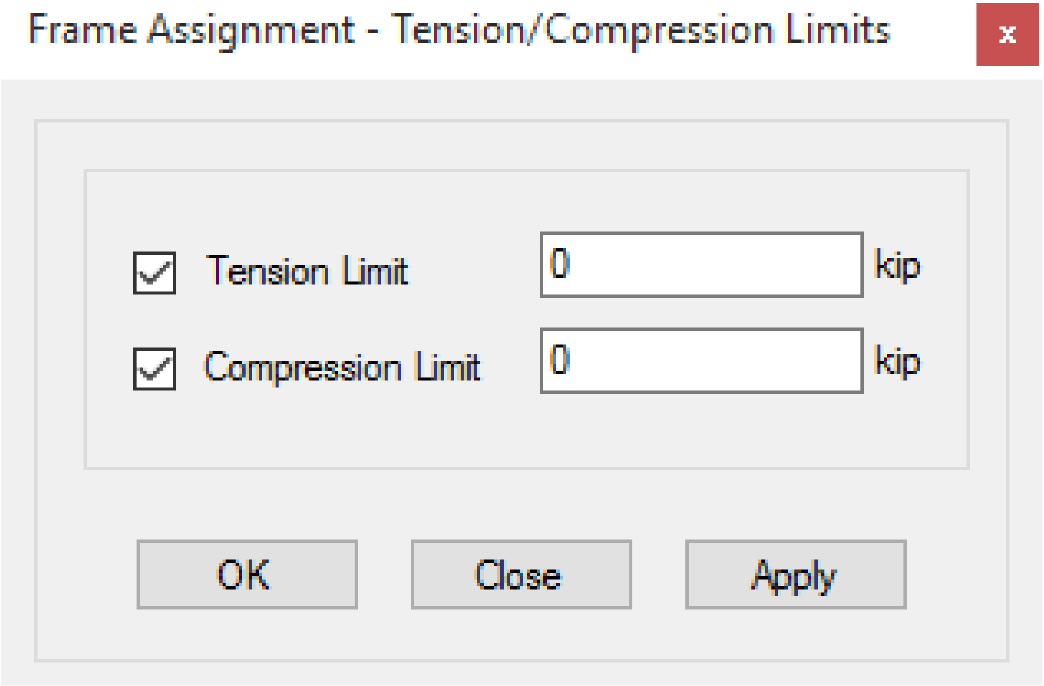
- Two types of nonlinear properties are available for the Frame/Cable element:
  1) **Tension/compression limits and**
  2) **Plastic hinges**

- When nonlinear properties are present in the element, they only affect nonlinear analyses. Linear analyses starting from zero conditions (the un-stressed state) behave as if the nonlinear properties were not present.

Tension/Compression Limits

- You may specify a **maximum tension and/or a maximum compression** that a frame/cable element may take.
- In the most common case, you can define a no-compression cable or brace by specifying the compression limit to be zero.
- If you specify a tension limit, it must be zero or a positive value. If you specify a compression limit, it must be zero or a negative value. If you specify a tension and compression limit of zero, the element will carry no axial force.
- The tension/compression limit behavior is elastic. Any axial extension beyond the tension limit and axial shortening beyond the compression limit will occur with zero axial stiffness. These deformations are recovered elastically at zero stiffness.
- Bending, shear, and torsional behavior are not affected by the **axial nonlinearity**.

Tension/Compression Limits


Frame Assignment - Tension/Compression Limits

☑ Tension Limit 0 kip

☑ Compression Limit 0 kip

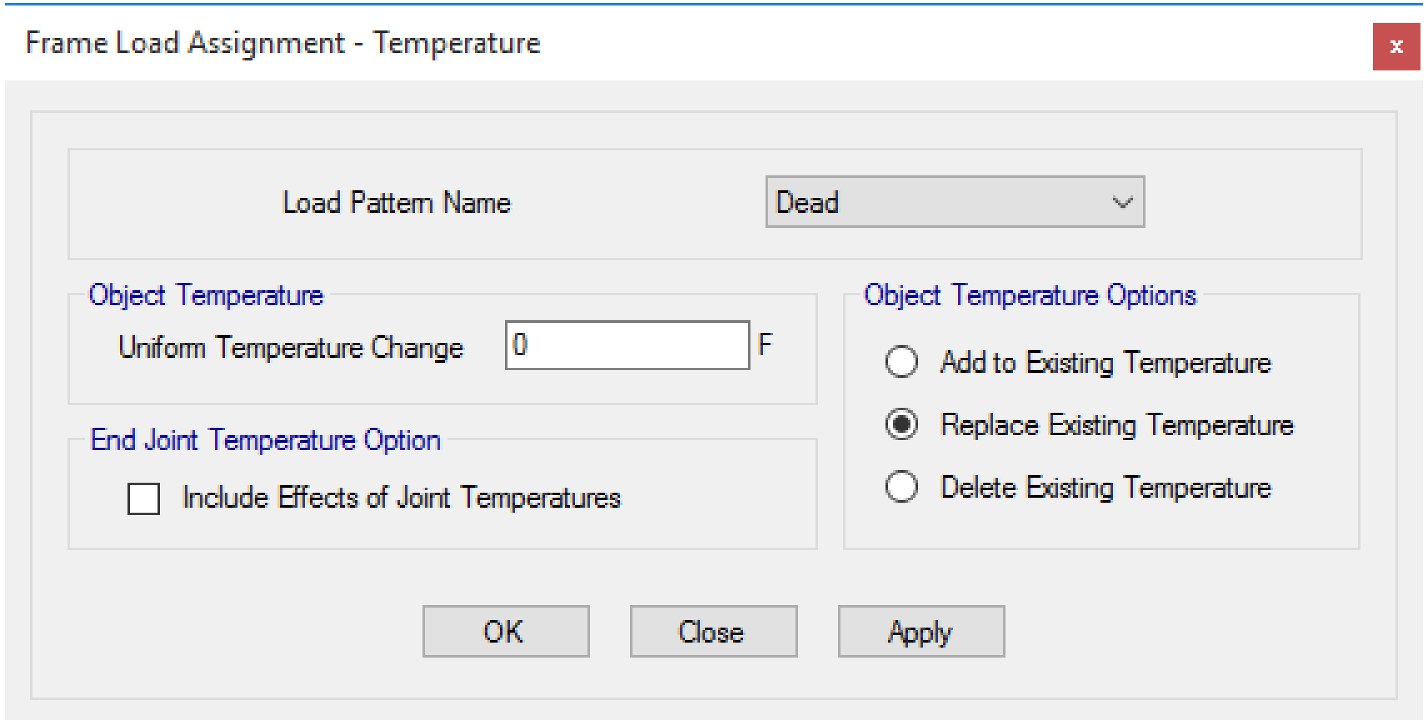
- Diagonal struts (representing e.g. the masonry infills) can be modeled with Tension/Compression Limits

Plastic Hinges

- You may insert plastic hinges at any number of locations along the clear length of the element.
- **This is a topic of Lecture 6 (b); Nonlinear Modeling of Structures**

Temperature Loads

- Temperature Load creates **thermal strain** in the Frame element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element.
- All specified Temperature Loads represent a change in temperature from the unstressed state for a linear analysis, or from the previous temperature in a nonlinear analysis.



Frame Load Assignment - Temperature

Load Pattern Name: Dead

Object Temperature

Uniform Temperature Change: 0 F

End Joint Temperature Option

☐ Include Effects of Joint Temperatures

Object Temperature Options

○ Add to Existing Temperature

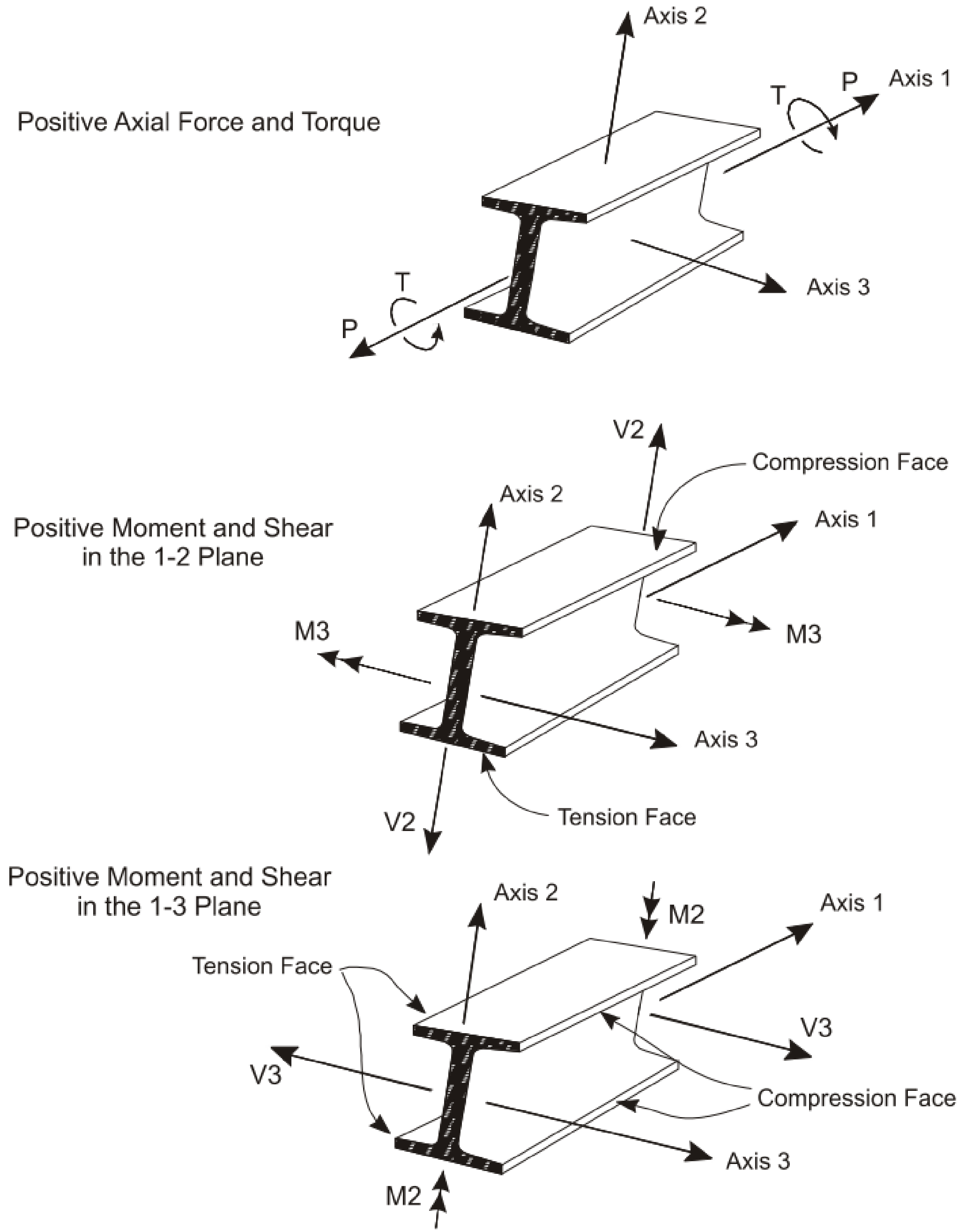
● Replace Existing Temperature

○ Delete Existing Temperature

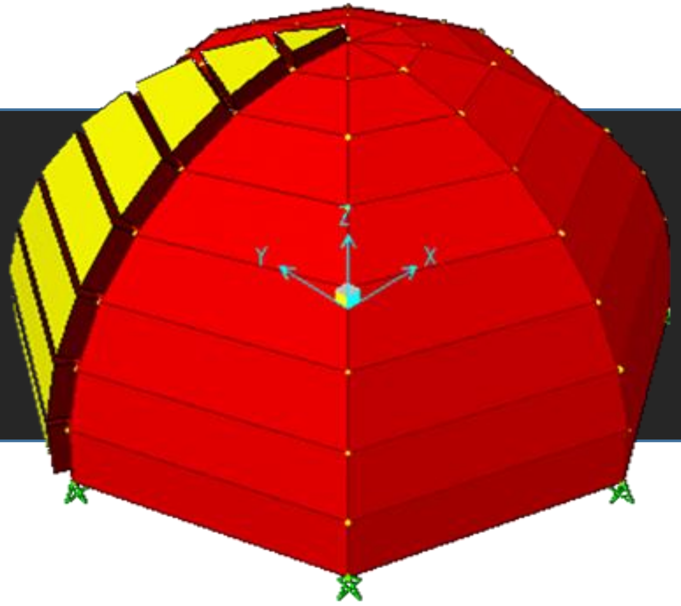
Internal Force Output

The Frame element internal forces are the forces and moments that result from integrating the stresses over an element cross-section. These internal forces are:

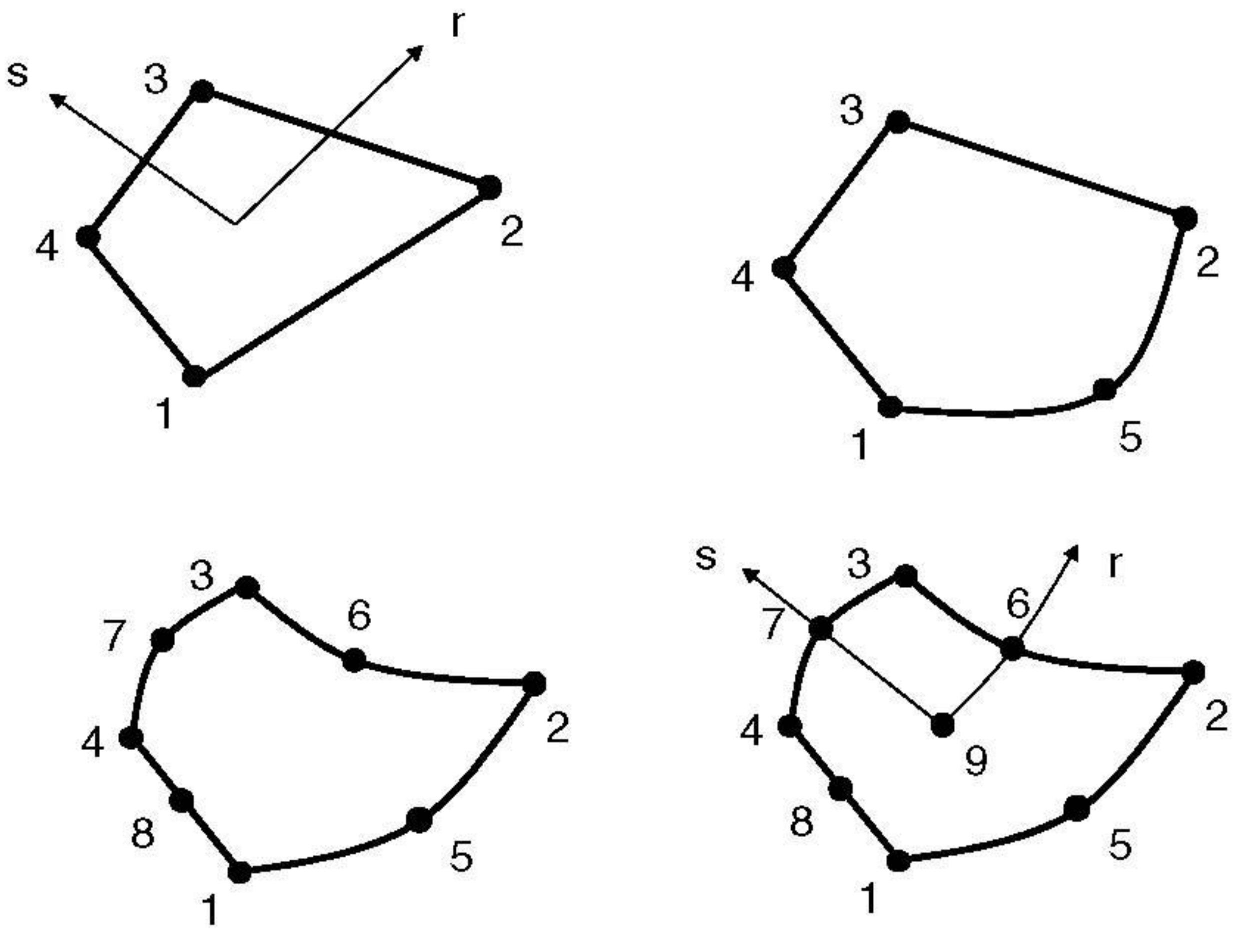
- P, the axial force
- V2, the shear force in the 1-2 plane
- V3, the shear force in the 1-3 plane
- T, the axial torque
- M2, the bending moment in the 1-3 plane (about the 2 axis)
- M3, the bending moment in the 1-2 plane (about the 3 axis)



Two Dimensional Elements

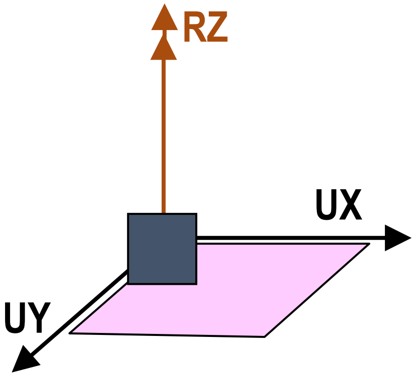


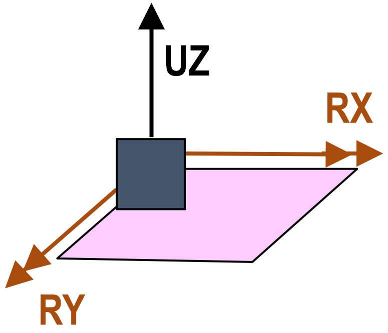
Two-dimensional Elements

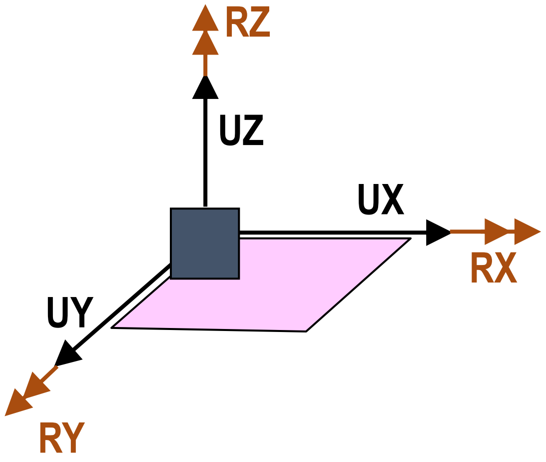


Four-to Nine-Node Two-Dimensional Isoparametric Elements

DOF for 2D Elements

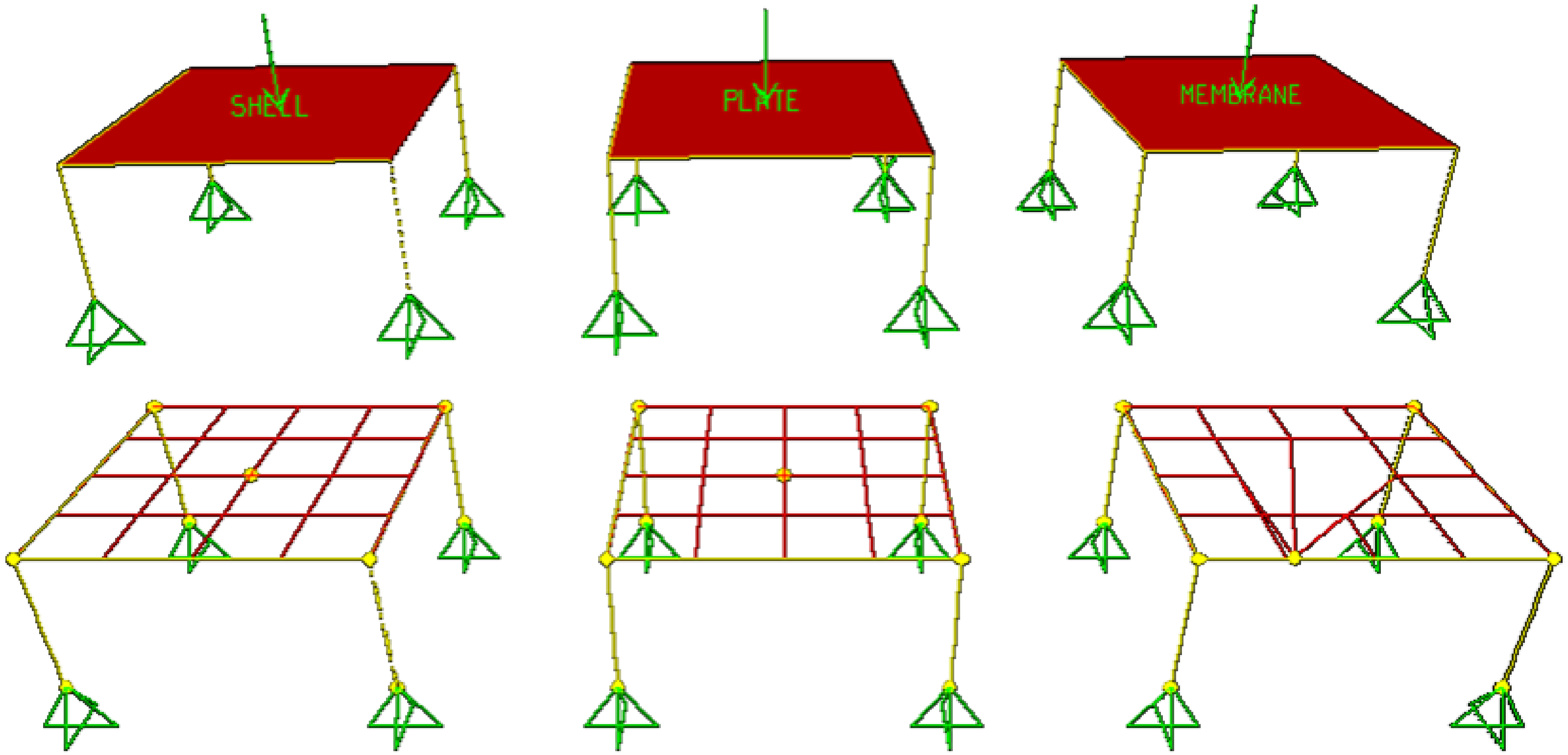




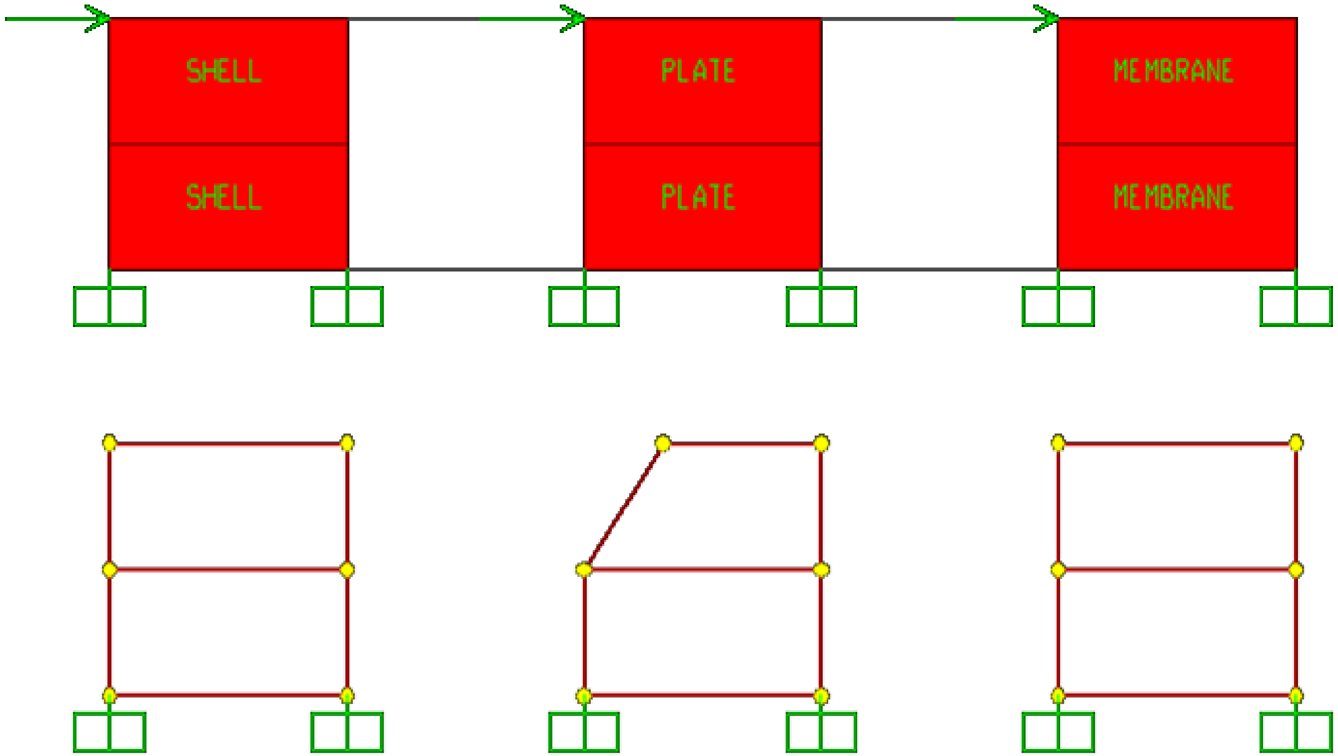


Membrane + **Plate** = **Shell**

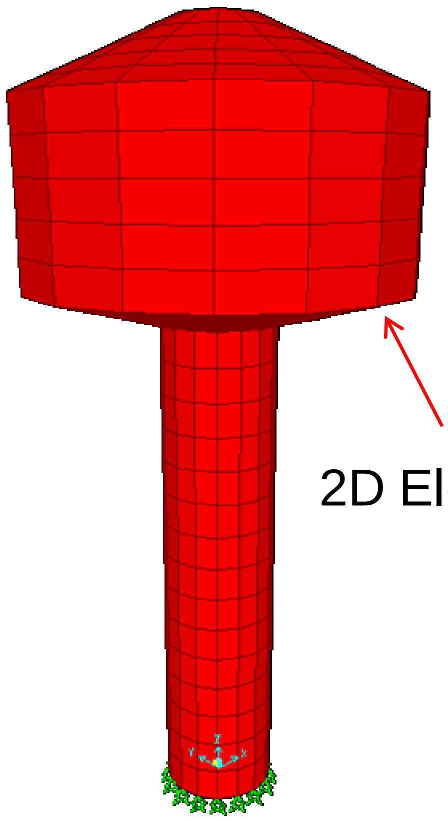
DOF for 2D Elements



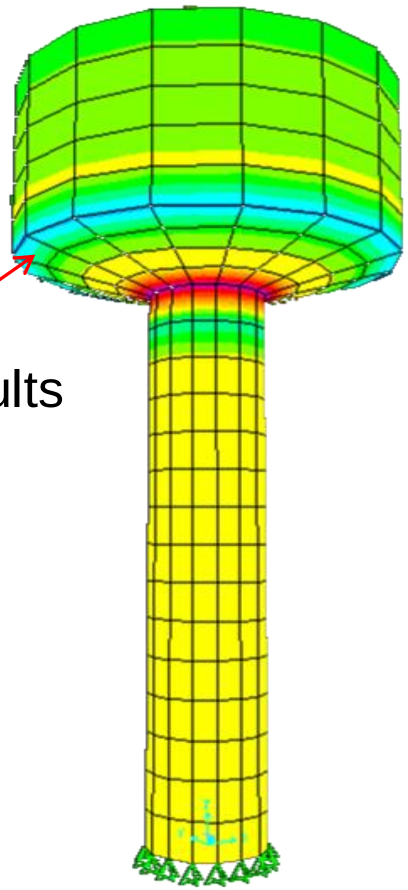
DOF for 2D Elements



2D Elements in FE Model



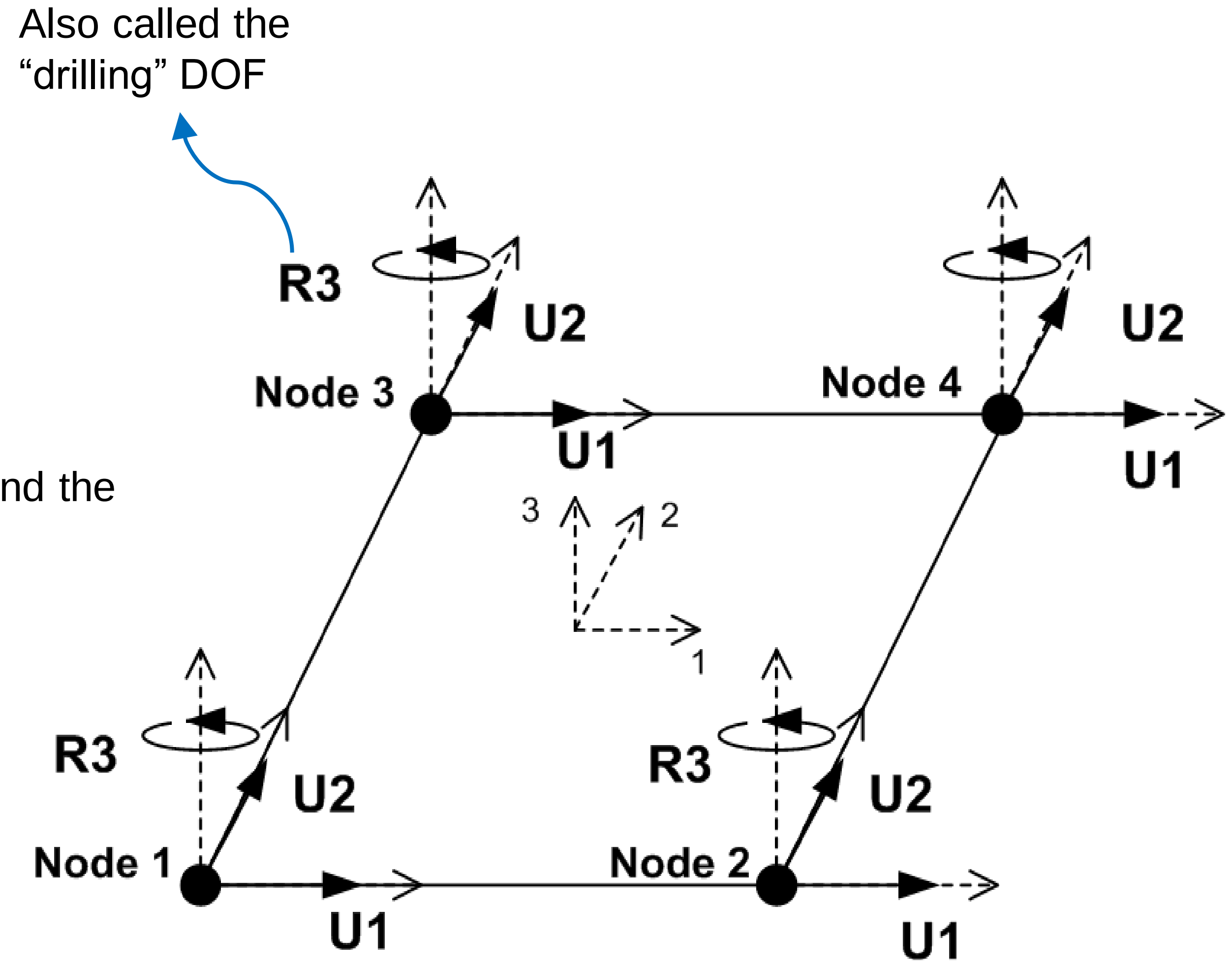
2D Elements in FE Model



Stress Results

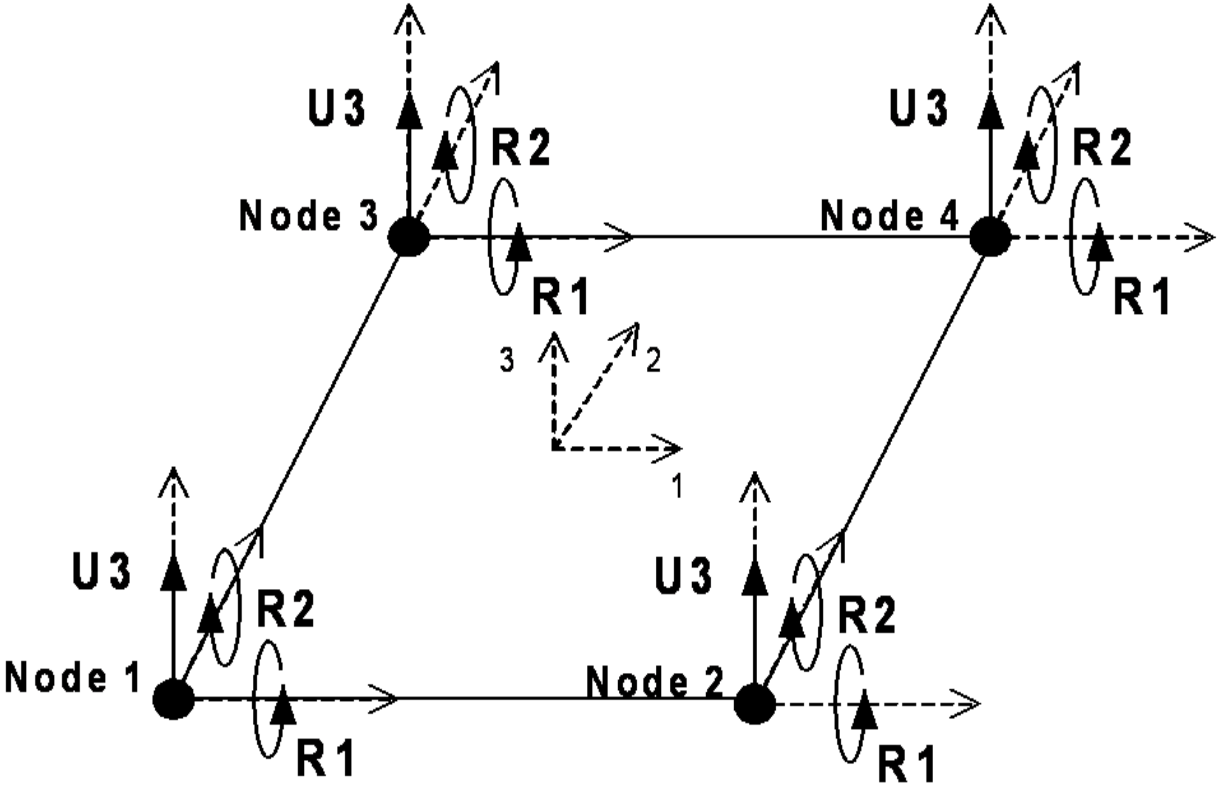
The Membrane Element

- 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
  - Pure membrane behavior; only the in-plane forces and the normal (drilling) moment can be supported
- Application
  - For Modeling surface elements carrying in-plane loads
  - Walls, Deep Beams, Domes, Thin Shells
- Building Specific Application
  - For representing floor slabs for Lateral Load Analysis.
  - Model Shear walls, Floor Diaphragm etc.


Also called the “drilling” DOF

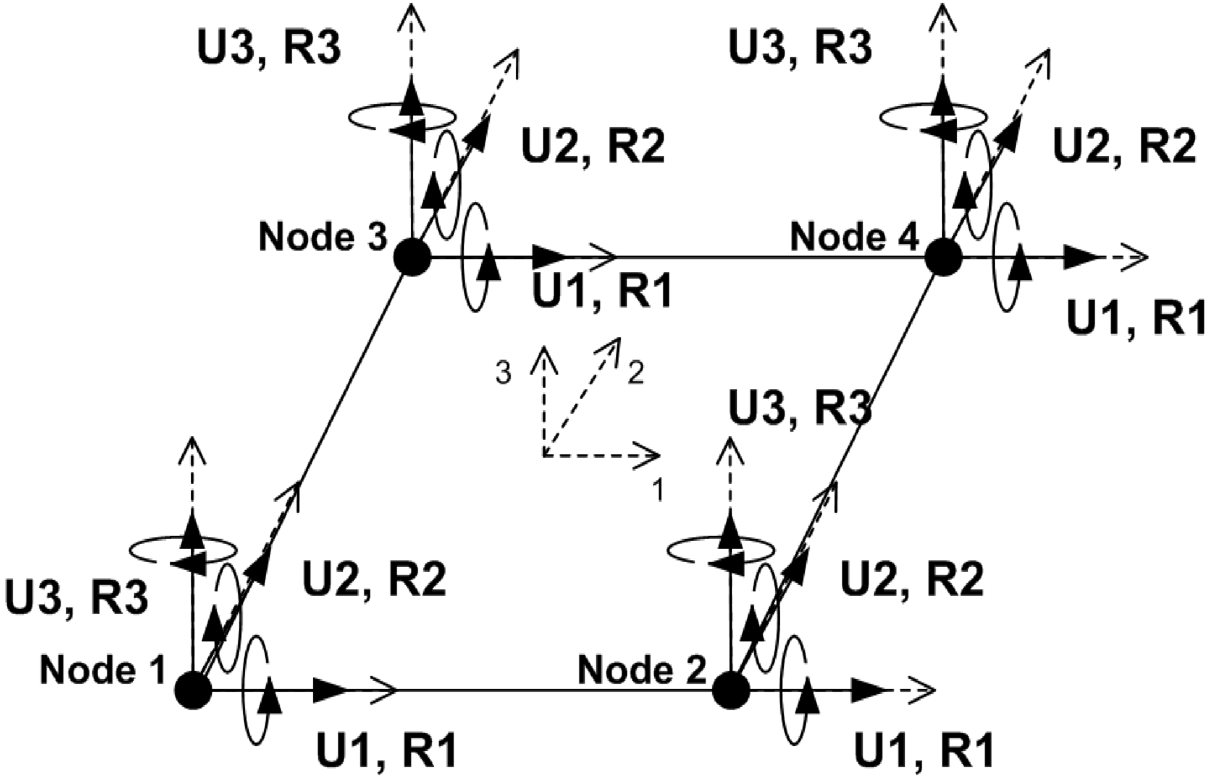
The Plate Element

- 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, Footings, Mats



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
  - Every surface type member
- Building Specific Application
  - May be used for modeling of general slabs and shear wall systems.



Area Objects in SAP2000/ETABS/CSI Bridge

- Finite Element Classification
  - Shell
  - Plate
    - Thick Plate
    - Thin Plate

    Based on shear deformation
  - Membrane
- Building Specific Classification
  - Plank – One way
  - Slab – One way or Two way
  - Deck – One way

Shell Elements in SAP/ETABS

- The Shell element is a three- or four- node formulation that combines membrane and plate-bending behavior.
- A four-point numerical integration formulation is used for the Shell stiffness. Stresses and internal forces and moments, in the element local coordinate system, are evaluated at the 2-by-2 Gauss integration points and extrapolated to the joints of the element.
- Structures that can be modeled with this element include:
  - Floor systems
  - Wall systems
  - Bridge decks
  - Three-dimensional curved shells, such as tanks and domes
  - Detailed models of beams, columns, pipes, and other structural members
- Two distinct formulations are available: **homogenous** and **layered**.

Homogeneous Shell Element in SAP/ETABS

- The homogeneous shell combines independent membrane and plate behavior. These behaviors become coupled if the element is warped (non-planar.)
- The membrane behavior uses an iso-parametric formulation that includes translational in-plane stiffness components and a “drilling” rotational stiffness component in the direction normal to the plane of the element [Taylor and Simo (1985) and Ibrahimbegovic and Wilson (1991)].
- Plate-bending behavior includes two-way, out-of-plane, plate rotational stiffness components and a translational stiffness component in the direction normal to the plane of the element. You may choose a **thin-plate (Kirchhoff) formulation** that neglects transverse shearing deformation, or a **thick-plate (Mindlin/Reissner) formulation** which includes the effects of transverse shearing deformation.
- For each homogeneous Shell element in the structure, you can choose to model pure-membrane, pure-plate, or full-shell behavior. It is generally recommended that you use the full shell behavior unless the entire structure is planar and is adequately restrained.

Thickness Formulation

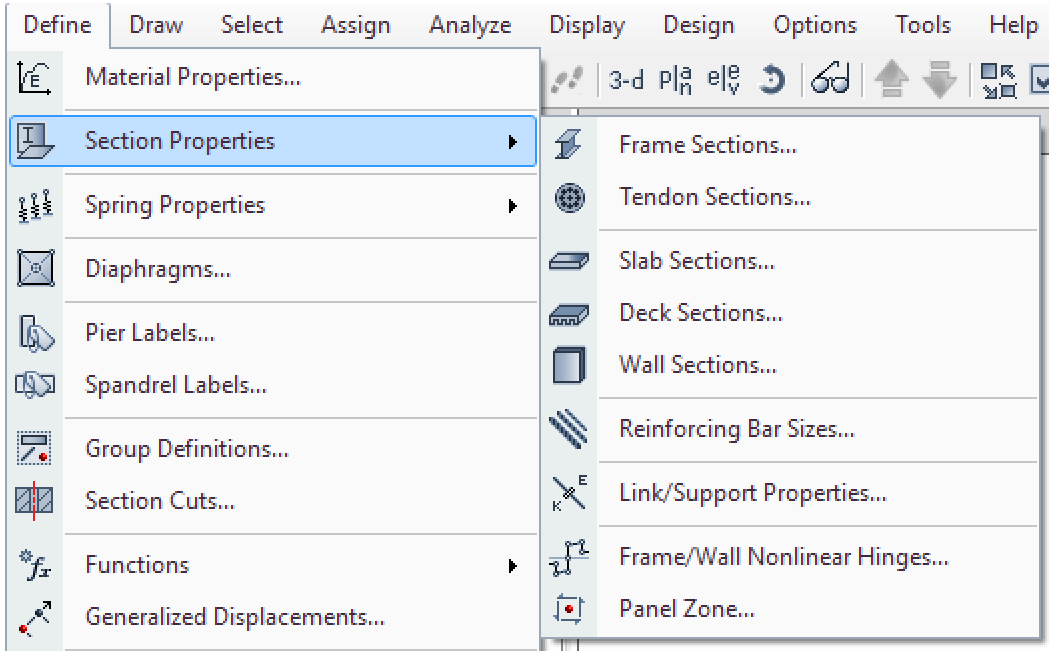
- **THICK**: A thick-plate (Mindlin/Reissner) formulation is used which includes the effects of transverse shear deformation
- **THIN**: A thin-plate (Kirchhoff) formulation is used that neglects transverse shearing deformation

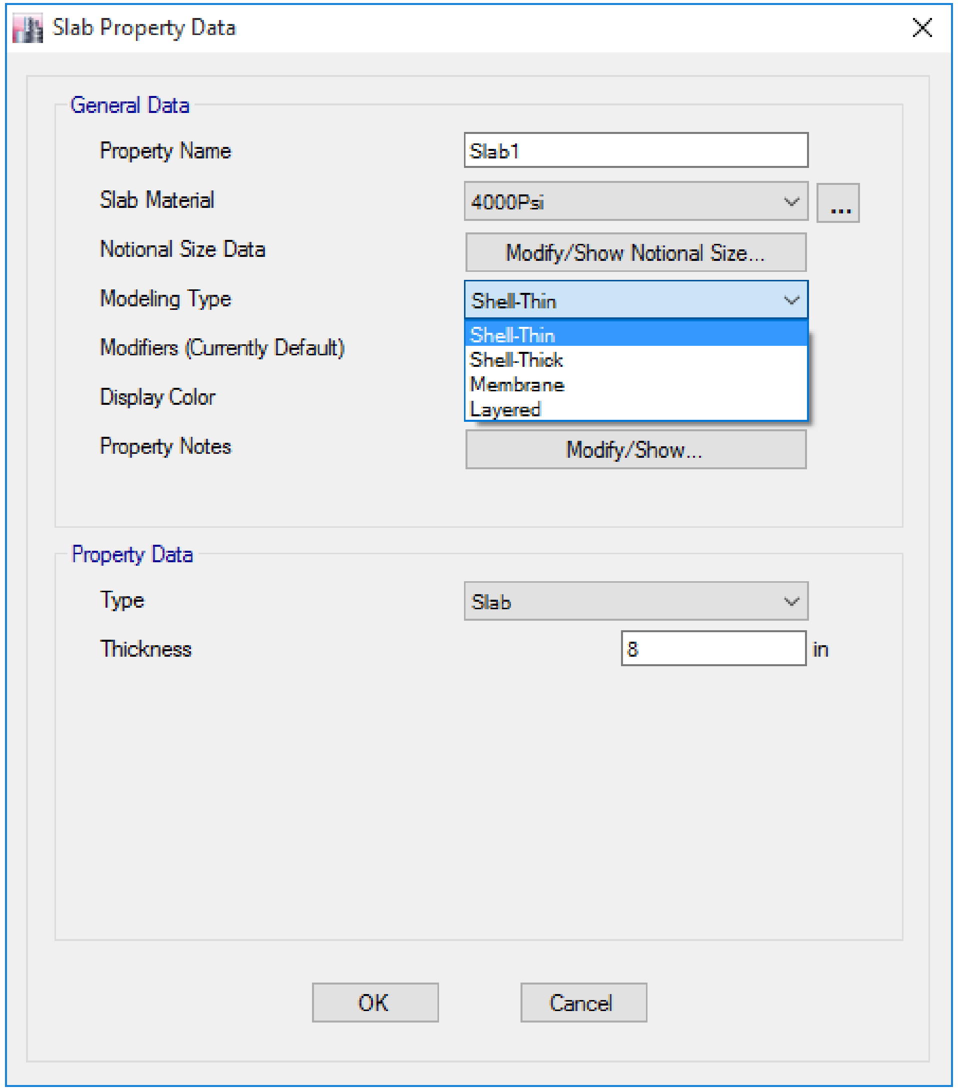
- Shearing deformations tend to be important when the thickness is greater than about **one-tenth to one-fifth of the span**. They can also be quite significant in the vicinity of bending-stress concentrations, such as near sudden changes in thickness or support conditions, and near holes or re-entrant corners.

- Even for thin-plate bending problems where shearing deformations are truly negligible, the thick-plate formulation tends to be more accurate, although somewhat stiffer, than the thin-plate formulation.

- The thickness formulation has no effect upon membrane behavior, only upon plate-bending behavior.

Homogeneous Shell Element in SAP/ETABS




Slab Property Data

General Data

Property Name: Slab1

Slab Material: 4000Psi

Notional Size Data: Modify/Show Notional Size...

Modeling Type: Shell-Thin

- Shell-Thin
- Shell-Thick
- Membrane
- Layered

Modifiers (Currently Default)

Display Color

Property Notes: Modify/Show...

Property Data

Type: Slab

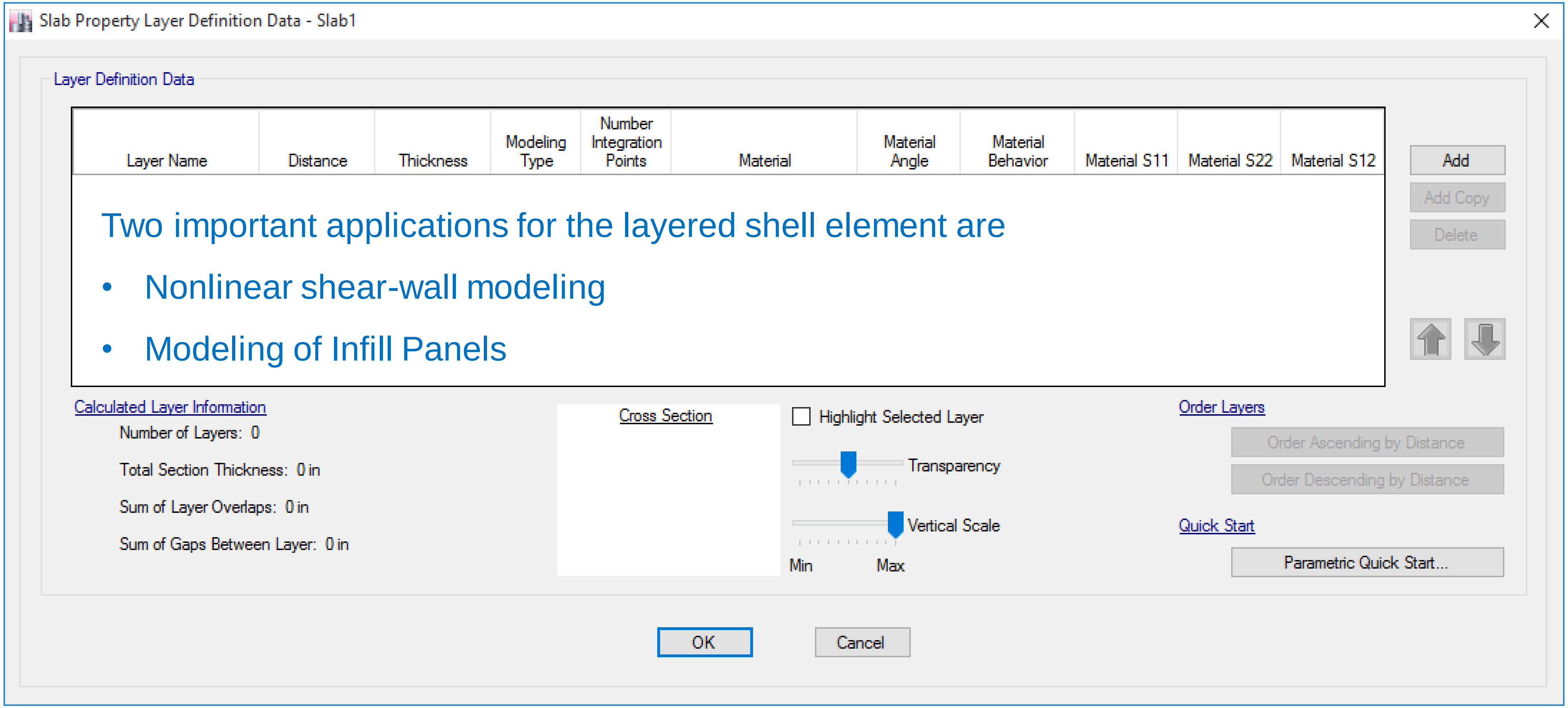
Thickness: 8 in

The mass contributed by the Shell element is lumped at the element joints. No inertial effects are considered within the element itself.

Layered Shell Element in SAP/ETABS

- The layered shell allows any number of layers to be defined in the thickness direction, each with an independent location, thickness, behavior, and material.
- Material behavior may be nonlinear.
- Membrane deformation within each layer uses a strain-projection method (Hughes, 2000.)
- Unlike for the homogeneous shell, the “drilling” degrees of freedom are not used, and they should not be loaded.
- For bending, a Mindlin/Reissner formulation is used which always includes transverse shear deformations.
- The layered Shell usually represents full-shell behavior, although you can control this on a layer-by-layer basis. Unless the layering is fully symmetrical in the thickness direction, membrane and plate behavior will be coupled.

Layered Shell Element in SAP/ETABS



Number of Layers: 0

Total Section Thickness: 0 in

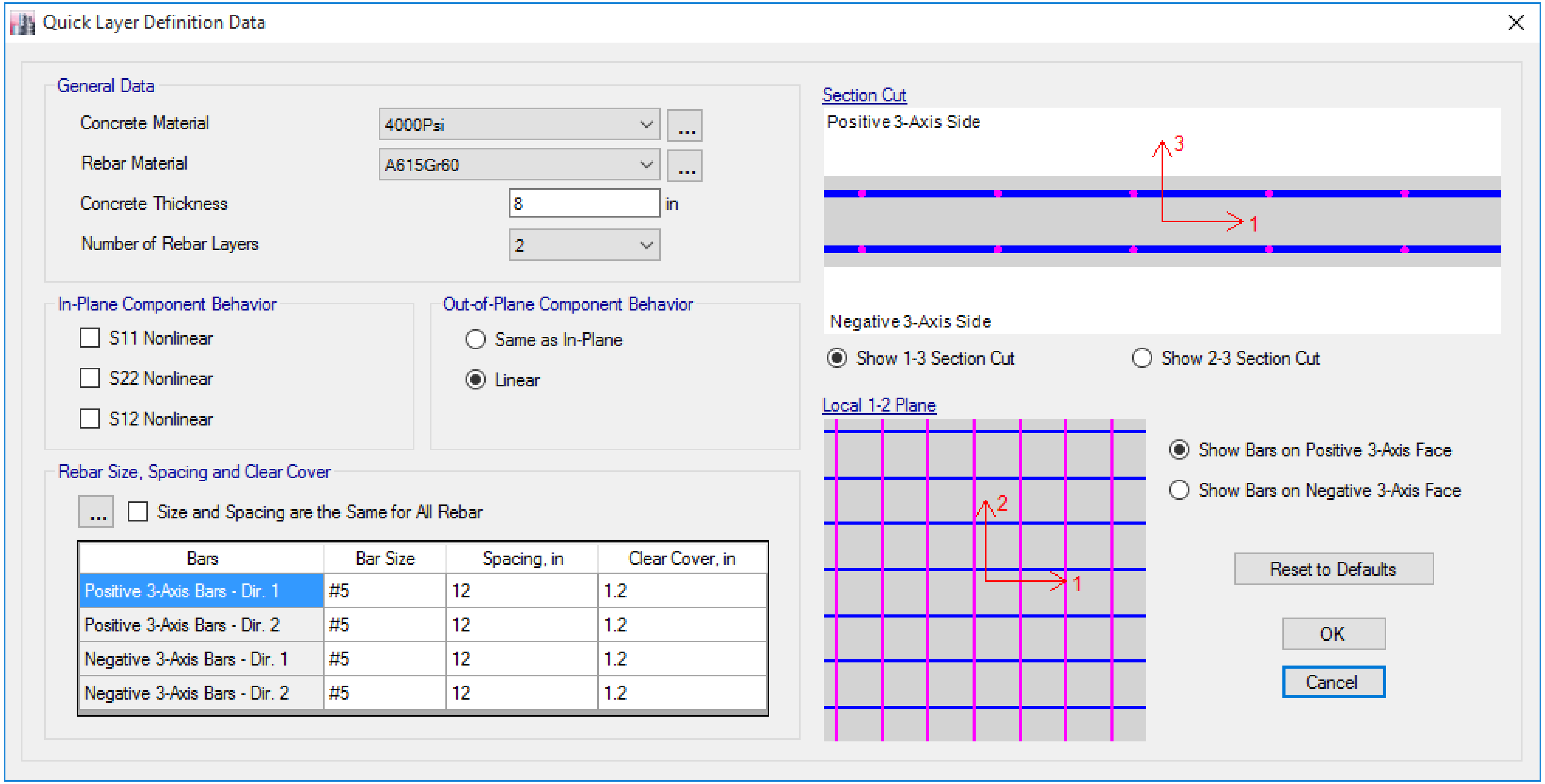
Sum of Layer Overlaps: 0 in

Sum of Gaps Between Layer: 0 in

Two important applications for the layered shell element are

- Nonlinear shear-wall modeling
- Modeling of Infill Panels

Layered Shell Element in SAP/ETABS



General Data

Concrete Material: 4000Psi

Rebar Material: A615Gr60

Concrete Thickness: 8 in

Number of Rebar Layers: 2

In-Plane Component Behavior

☐ S11 Nonlinear

☐ S22 Nonlinear

☐ S12 Nonlinear

Out-of-Plane Component Behavior

○ Same as In-Plane

● Linear

Rebar Size, Spacing and Clear Cover

☐ Size and Spacing are the Same for All Rebar

Bars	Bar Size	Spacing, in	Clear Cover, in
Positive 3-Axis Bars - Dir. 1	#5	12	1.2
Positive 3-Axis Bars - Dir. 2	#5	12	1.2
Negative 3-Axis Bars - Dir. 1	#5	12	1.2
Negative 3-Axis Bars - Dir. 2	#5	12	1.2

● Show 1-3 Section Cut ○ Show 2-3 Section Cut

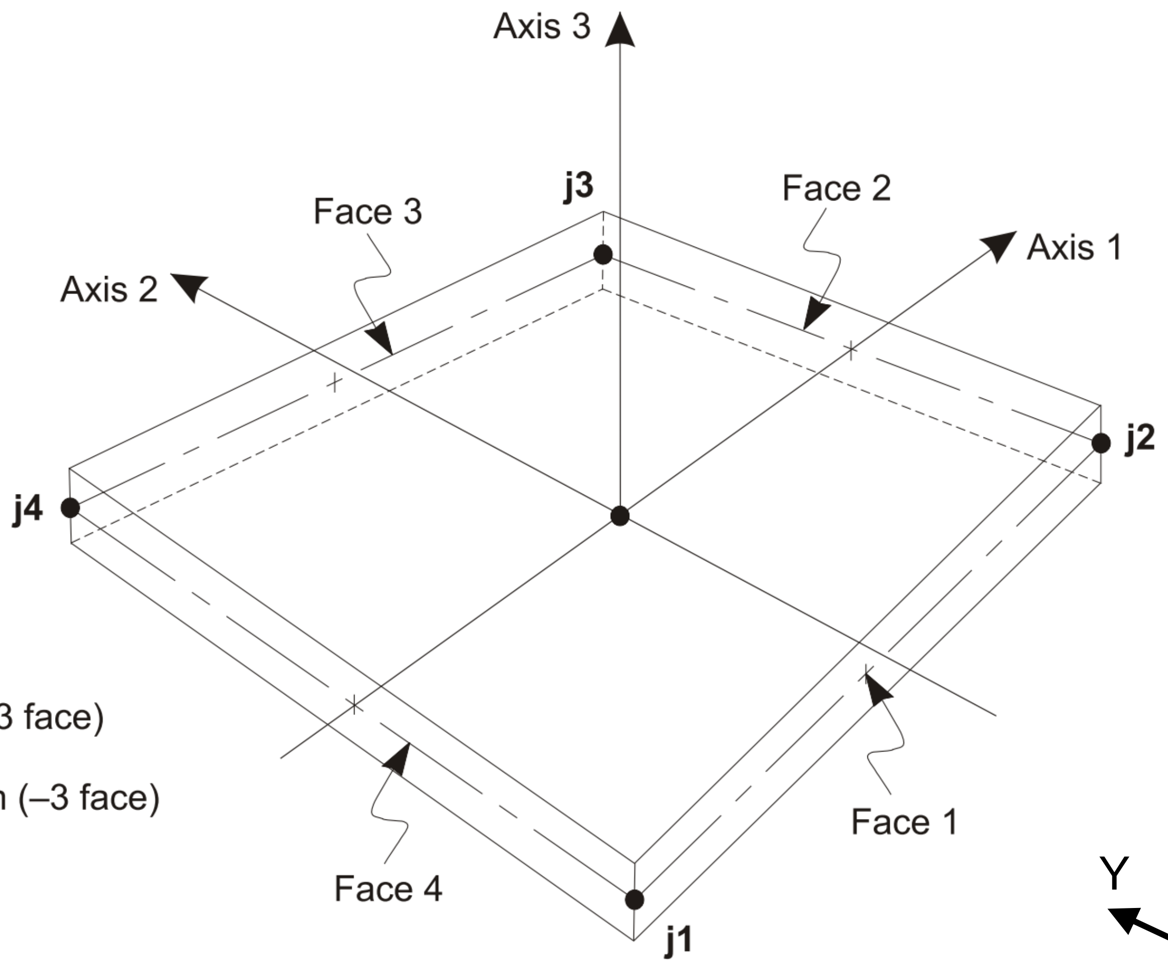
● Show Bars on Positive 3-Axis Face

○ Show Bars on Negative 3-Axis Face

Local Coordinates for Shell Element

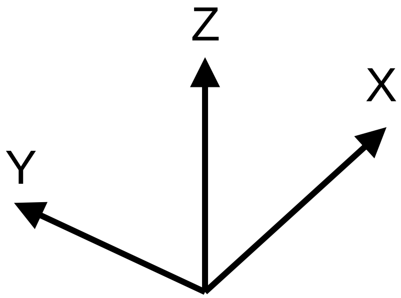
- Each Shell element has its own local coordinate system used to define Material properties, loads and output.
- The axes of this local system are denoted 1, 2 and 3. **The first two axes lie in the plane of the element the third axis is normal to the plane of the Shell element.**
- The default orientation of the local 1 and 2 axes is determined by the relationship between the local 3 axis and the global Z axis:
  - The local 3-2 plane is taken to be vertical, i.e., parallel to the Z axis
  - The local 2 axis is taken to have an upward (+Z) sense unless the element is horizontal, in which case the local 2 axis is taken along the global +Y direction
  - The local 1 axis is horizontal, i.e., it lies in the X-Y plane

Shell Elements in SAP2000 (Four-node Quadrilateral Shell Element)

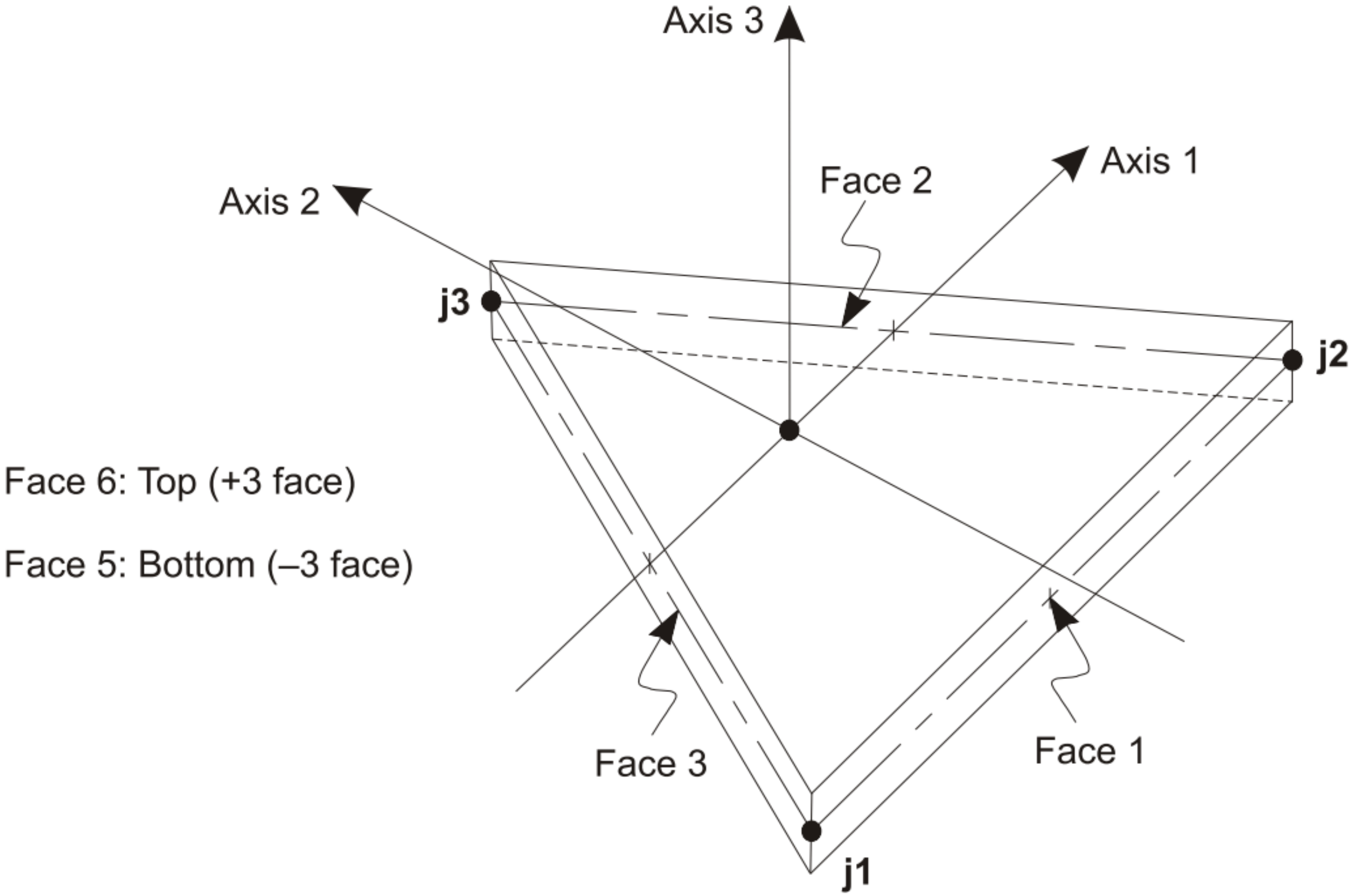


Face 6: Top (+3 face)

Face 5: Bottom (–3 face)



Shell Elements in SAP2000 (Three-node Triangular Shell Element)

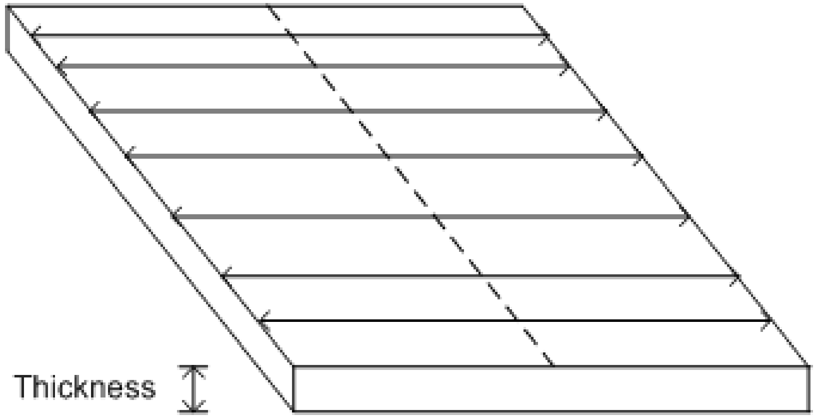


Section Properties

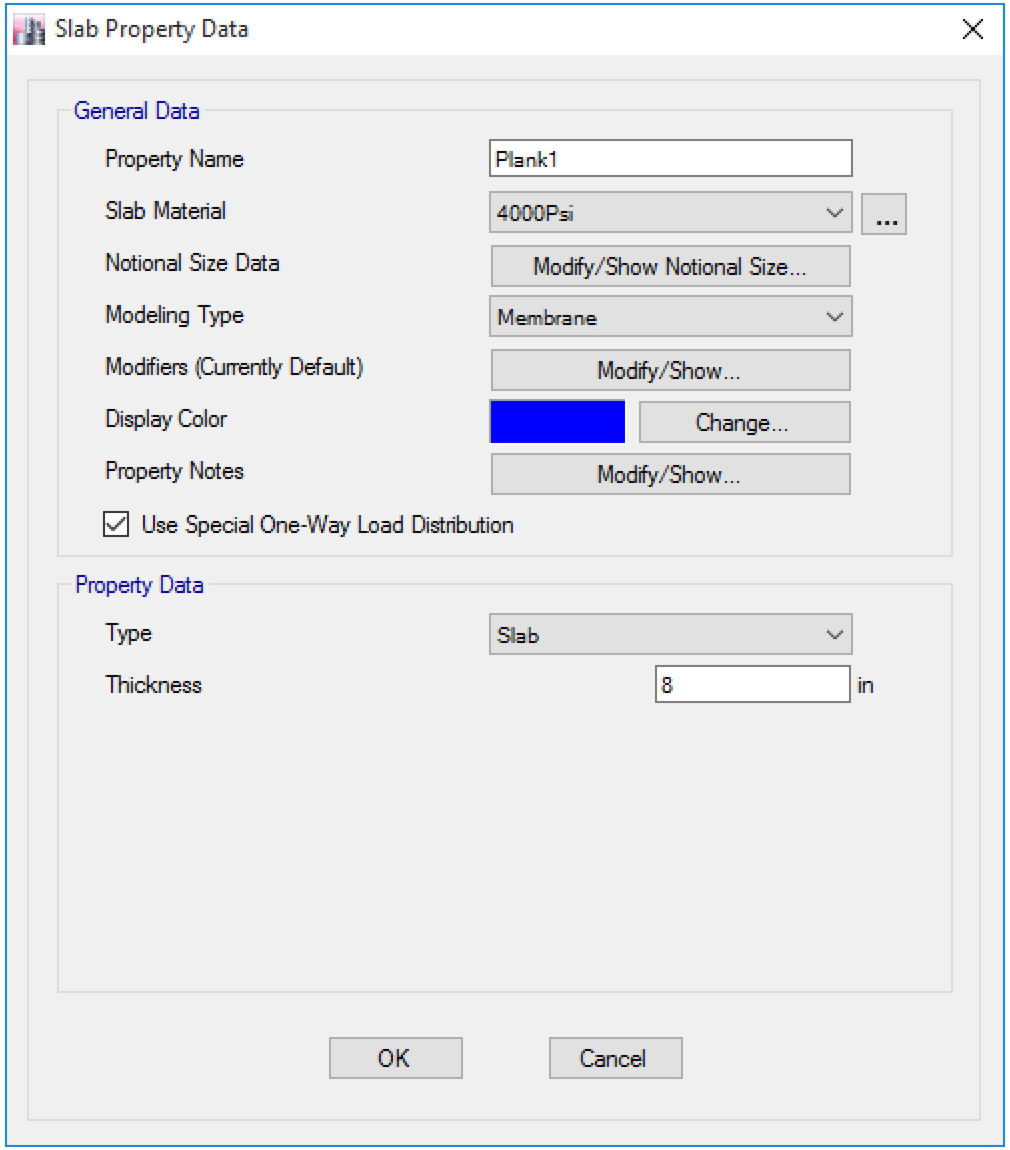
- A **Shell Section** is a set of material and geometric properties describing the cross-section of one or more Shell elements.
- Sections are defined independently of the Shell elements, and are referenced during the definition of the elements.
- Section Type
  - **Membrane** (Supports only the in-plane forces and the normal (drilling) moment, Linear homogeneous material).
  - **Plate** (Supports only the bending moments and the transverse force, Thick- or thin-plate formulation, Linear homogeneous material).
  - **Shell** (Supports all moments and forces, Thick- or thin-plate formulation, Linear homogeneous material).
  - **Layered** (Multiple layers, each with a different material, thickness, behavior, and location, Provides full-shell behavior un less all layers have only membrane or only plate behavior, With full-shell behavior, it supports all forces and moments except the “drilling” moment, Thick-plate formulation; may be non linear).

Area Object: Plank

- By default use **one-way load transfer mechanism**
- Generally used to model pre-cast slabs
- Can also be simple RC solid slab



Plank



General Data

Property Name: Plank1

Slab Material: 4000Psi

Notional Size Data:

Modeling Type: Membrane

Modifiers (Currently Default):

Display Color: 🟦

Property Notes:

☑ Use Special One-Way Load Distribution

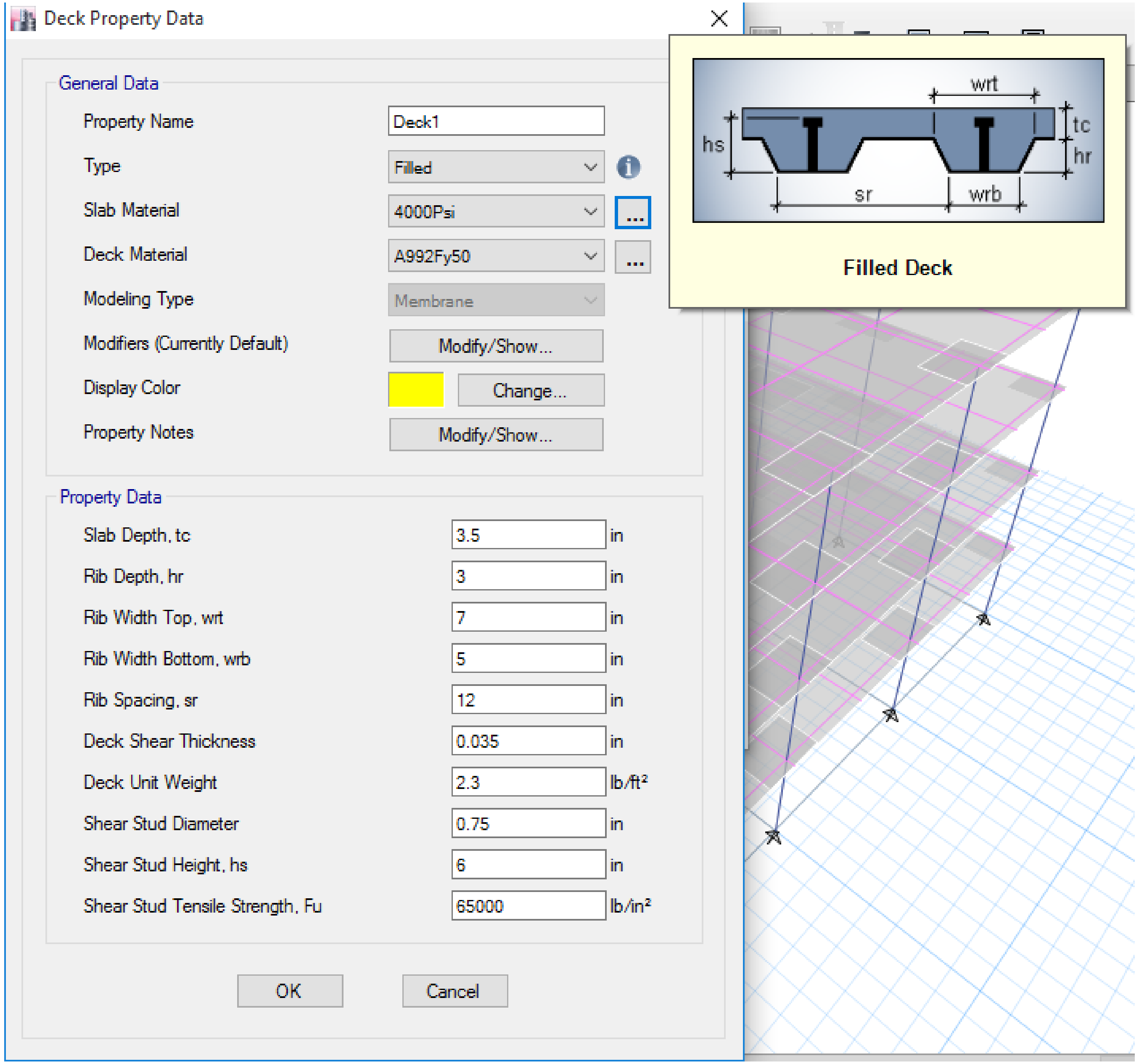
Property Data

Type: Slab

Thickness: 8 in

Area Object: Deck

- Use **one-way load transfer mechanism**
- Metallic Composite Slabs
- Includes shear studs
- Generally used in association with composite beams
- Deck slabs may be
    - Filled Deck
    - Unfilled Deck
    - Solid Slab Deck



Deck Property Data

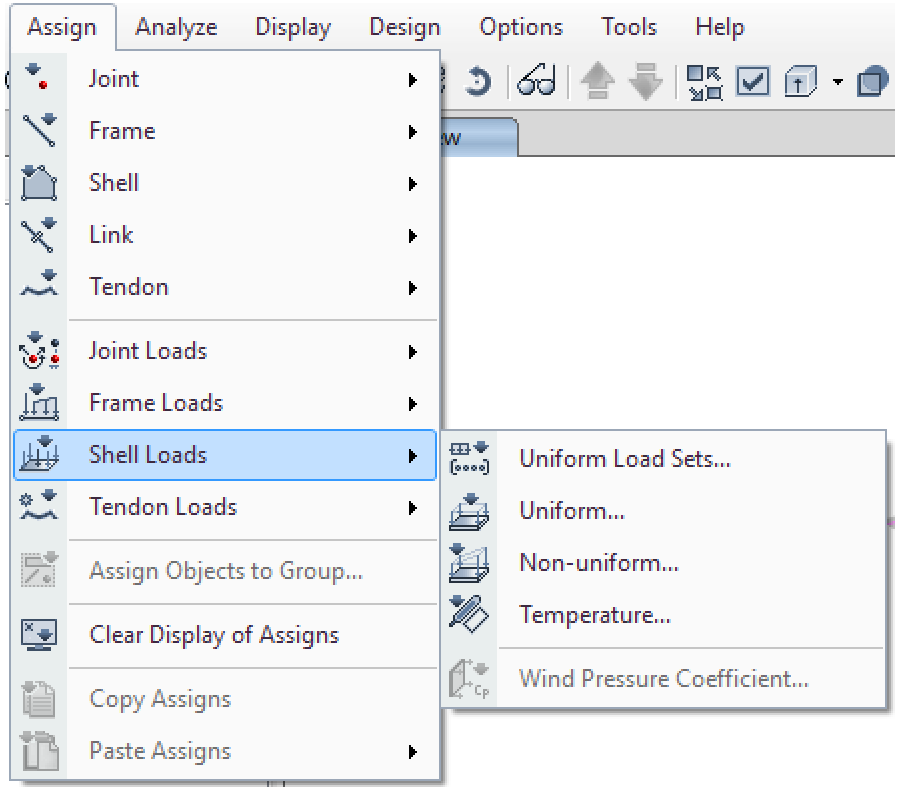
General Data

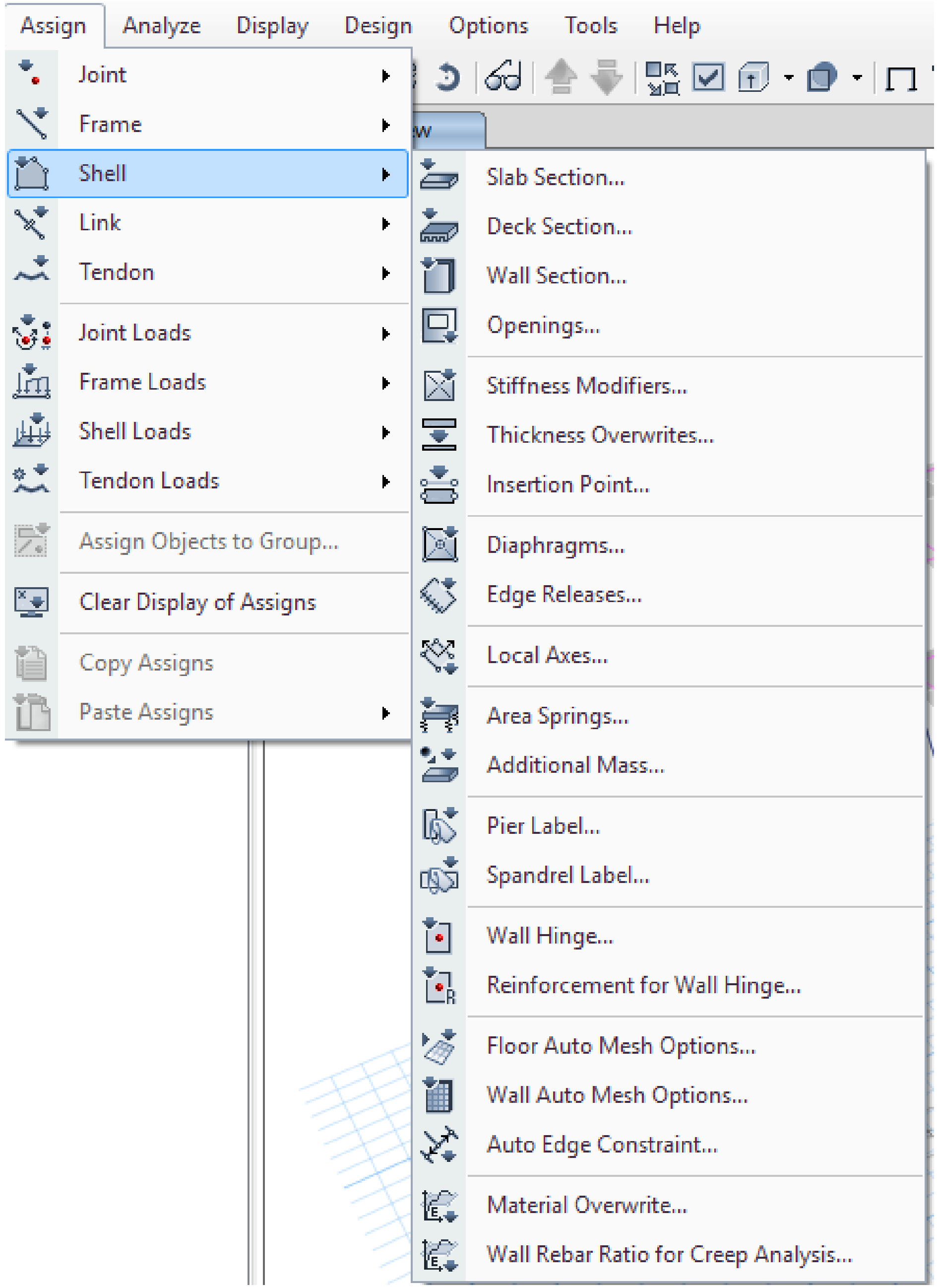
Property Name	Deck1
Type	Filled
Slab Material	4000Psi
Deck Material	A992Fy50
Modeling Type	Membrane
Modifiers (Currently Default)	Modify/Show...
Display Color	🟨 Change...
Property Notes	Modify/Show...

Property Data

Slab Depth, tc	3.5	in
Rib Depth, hr	3	in
Rib Width Top, wrt	7	in
Rib Width Bottom, wrb	5	in
Rib Spacing, sr	12	in
Deck Shear Thickness	0.035	in
Deck Unit Weight	2.3	lb/ft²
Shear Stud Diameter	0.75	in
Shear Stud Height, hs	6	in
Shear Stud Tensile Strength, Fu	65000	lb/in²

Shell Assignments

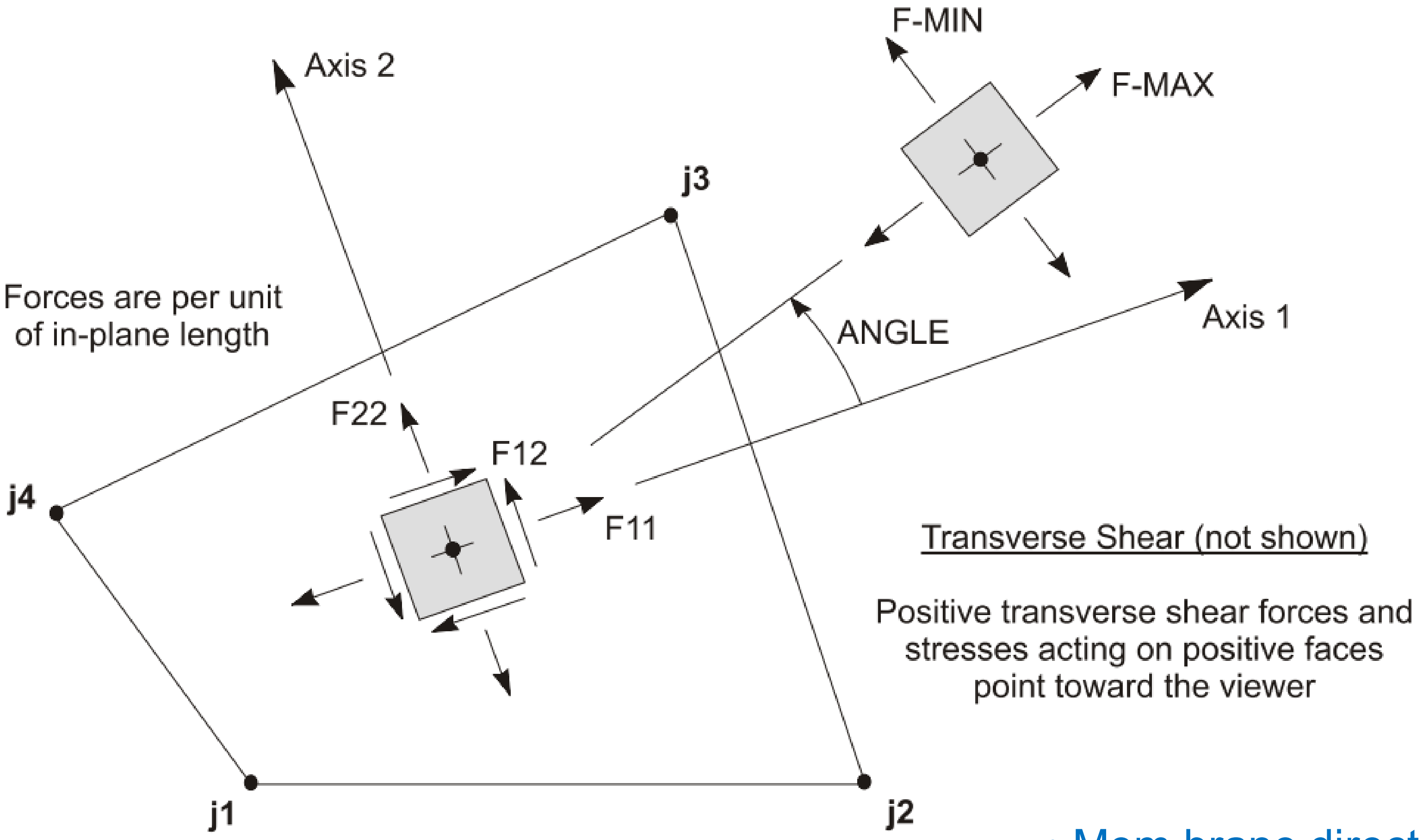




Internal Force and Stress Output

- The Shell element internal forces (also called stress resultants) are the forces and moments that result from integrating the stresses over the element thickness. For a homogeneous shell, these internal forces are:
  - Membrane direct forces: F11, F22
  - Membrane shear force: F12
  - Plate bending moments: M11, M22
  - Plate twisting moments: M12
  - Plate transverse shear forces: V13, V23
- It is very important to note that these stress resultants are forces and moments per unit of in-plane length. They are present at every point on the mid-surface of the element.

Internal Force and Stress Output



Forces are per unit of in-plane length

Transverse Shear (not shown)

Positive transverse shear forces and stresses acting on positive faces point toward the viewer

STRESSES AND MEMBRANE FORCES

Stress Sij Has Same Definition as Force Fij

- Membrane direct forces: F11, F22
- Membrane shear force: F12

Internal Force and Stress Output

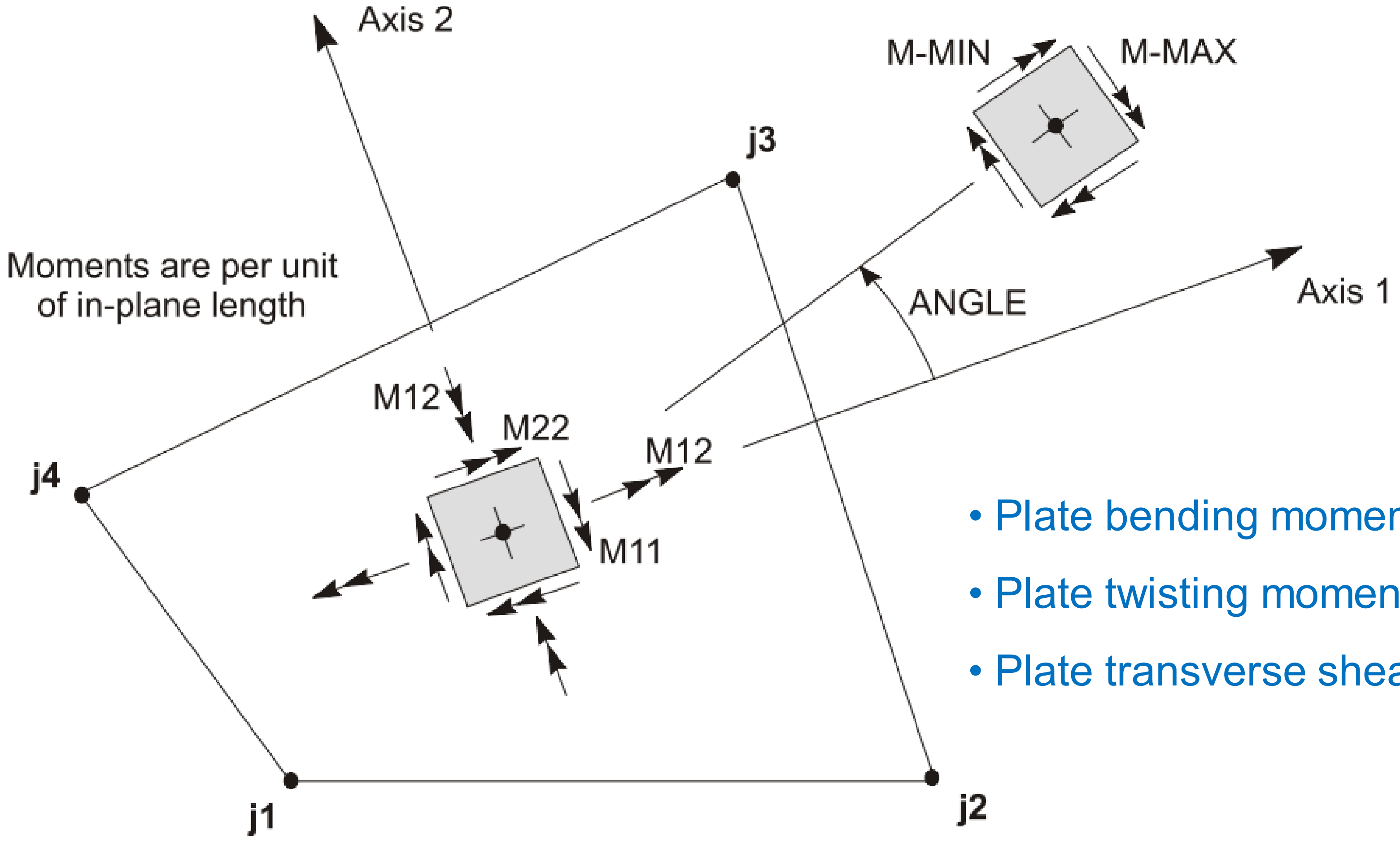
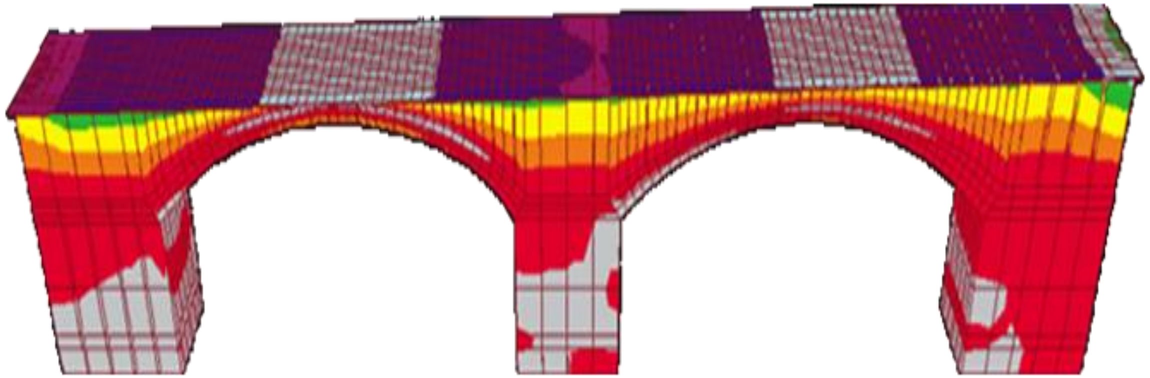


PLATE BENDING AND TWISTING MOMENTS

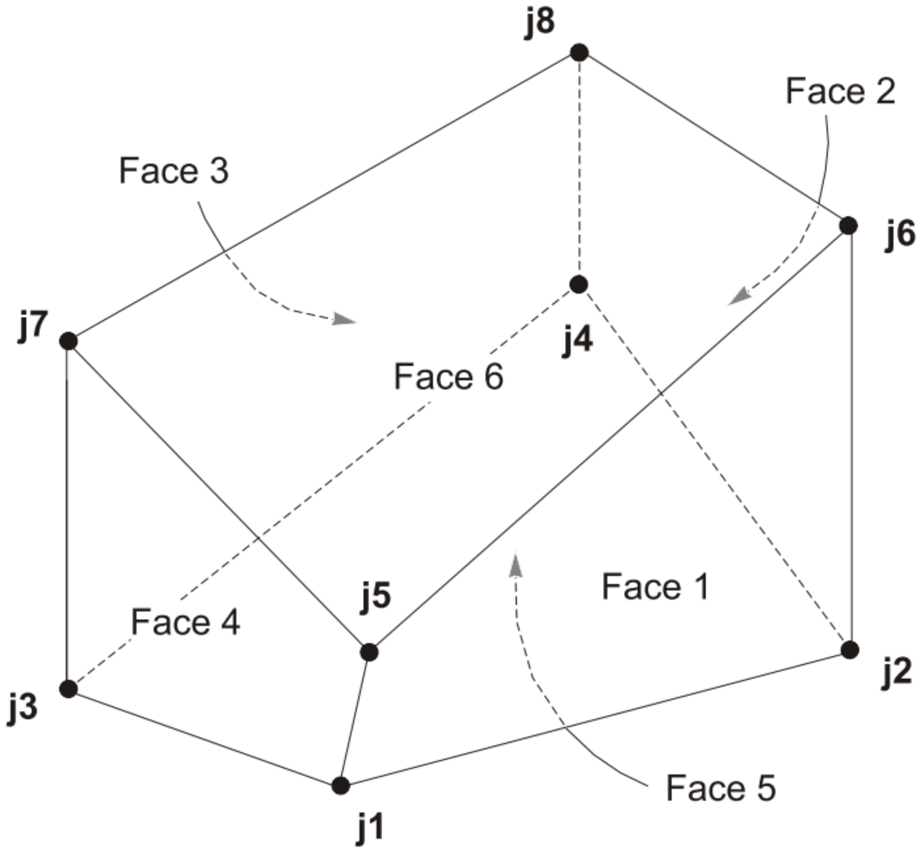
- Plate bending moments: M11, M22
- Plate twisting moments: M12
- Plate transverse shear forces: V13, V23

Three Dimensional Elements



The Solid Element

- The Solid element is an **eight-node element** used to model three-dimensional solid structures.
- Each solid has six quadrilateral faces with a joint at each corner. Nodes may be collapsed to form **wedges, tetrahedra, and other irregular volumes**. This is done by specifying the same joint number for two or more of the eight corner nodes, so long as the ordering of the nodes remains the same. Examples include:
  - Wedge (triangular bottom, triangular top): j1, j2, j3 = j4, j5, j6, j7 = j8
  - Tetrahedron (triangular bottom, point top): j1, j2, j3 = j4, j5 = j6 = j7 = j8
  - 7-node (rectangular bottom, triangular top): j1, j2, j3, j4, j5, j6, j7 = j8
  - Pyramid (rectangular bottom, point top): j1, j2, j3, j4, j5 = j6 = j7 = j8
- An 2 x 2 x 2 numerical integration scheme is used for the Solid. Stresses in the element local coordinate system are evaluated at the integration points and extrapolated to the joints of the element.



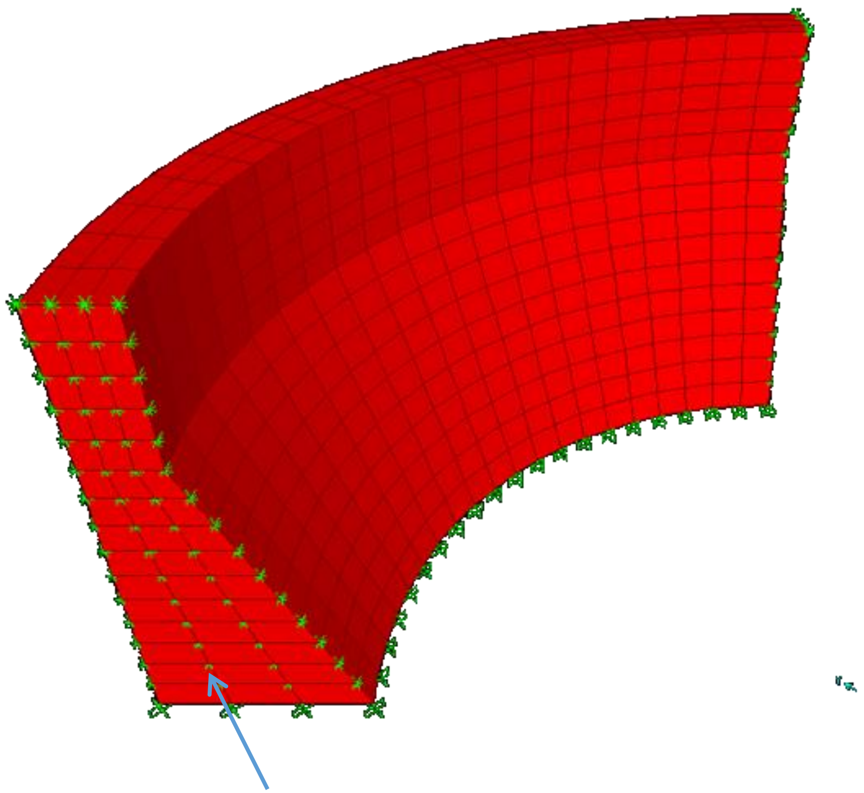
Degrees of Freedom and Local Coordinate System

- The Solid element activates the **three translational degrees of freedom at each of its connected joints**.
- Rotational degrees of freedom are **not** activated.
- This element contributes stiffness to all of these translational degrees of freedom.
- Each Solid element has its own element local coordinate system used to define Material properties, loads and output.
- The axes of this local system are denoted 1, 2 and 3.
- These axes always correspond with the global coordinate axes X, Y and Z, respectively, regardless of the orientation of the element.

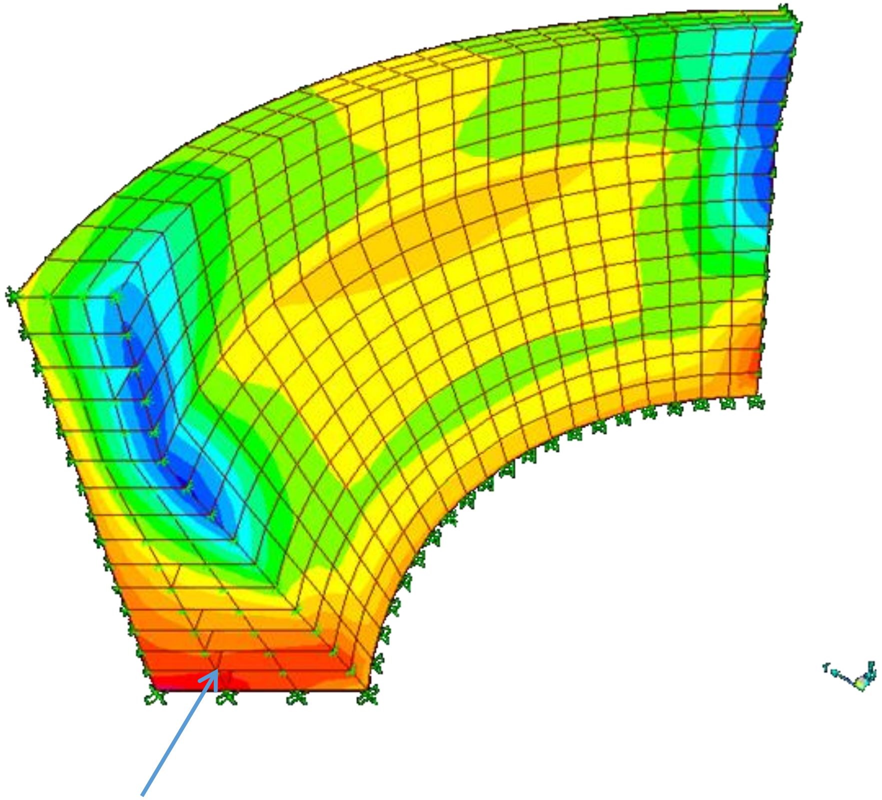
Material Properties

- The material properties for each Solid element are specified by reference to a previously-defined Material.
- Fully anisotropic material properties are used.
- The material properties used by the Solid element are:
    - The moduli of elasticity, e1, e2, and e3
    - The shear moduli, g12, g13, and g23
    - All of the Poisson’s ratios, u12, u13, u23, ..., u56
    - The coefficients of thermal expansion, a1, a2, a3, a12, a13, and a23
    - The mass density, m, used for computing element mass
    - The weight density, w, used for computing Self-Weight and Gravity Loads

Solid Elements in FE Model

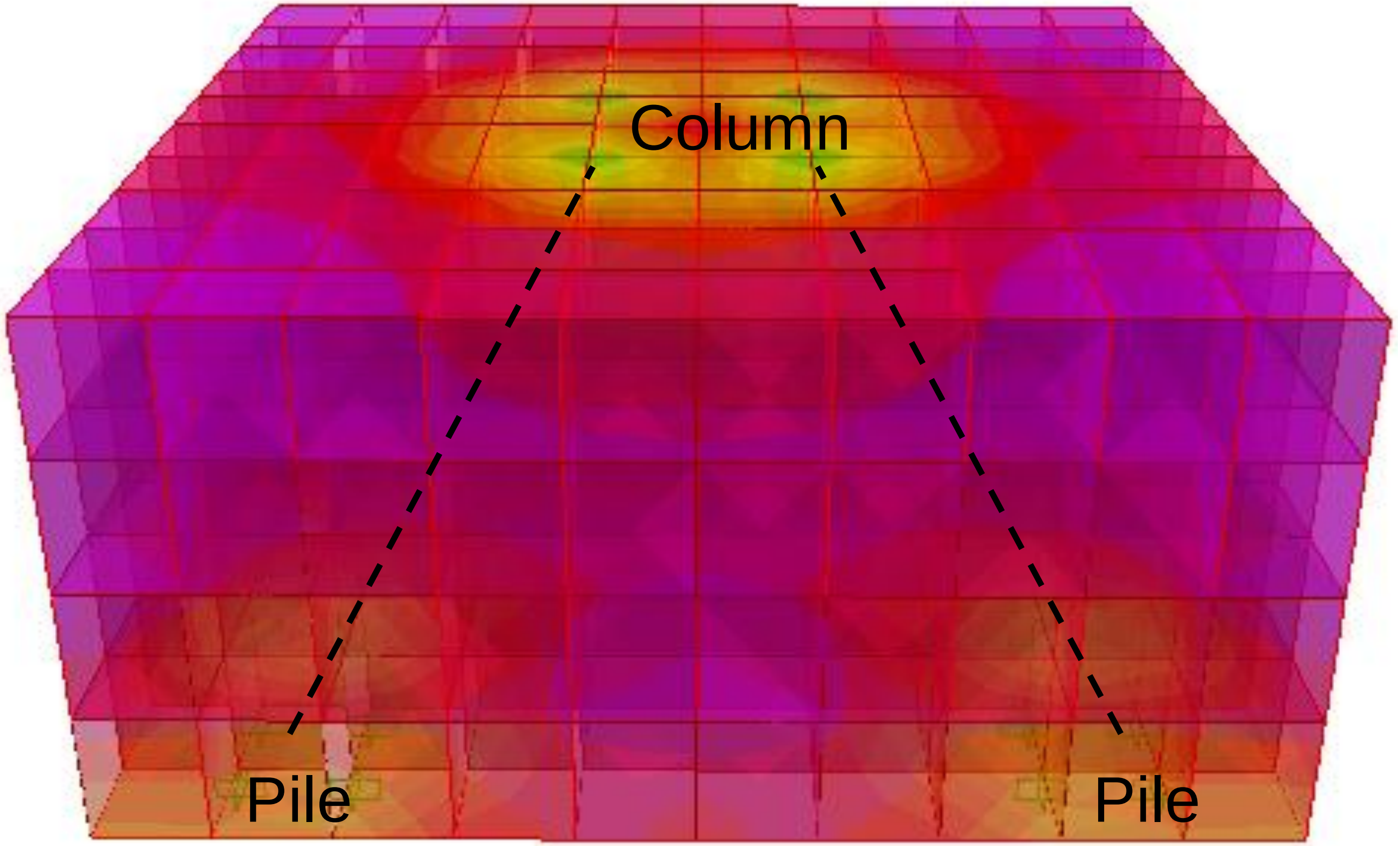


Solid Elements in FE Model

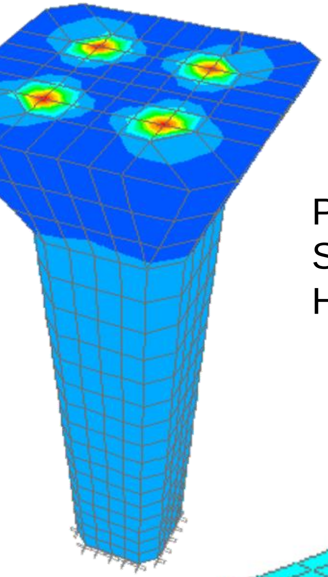


Stress in Solid Elements

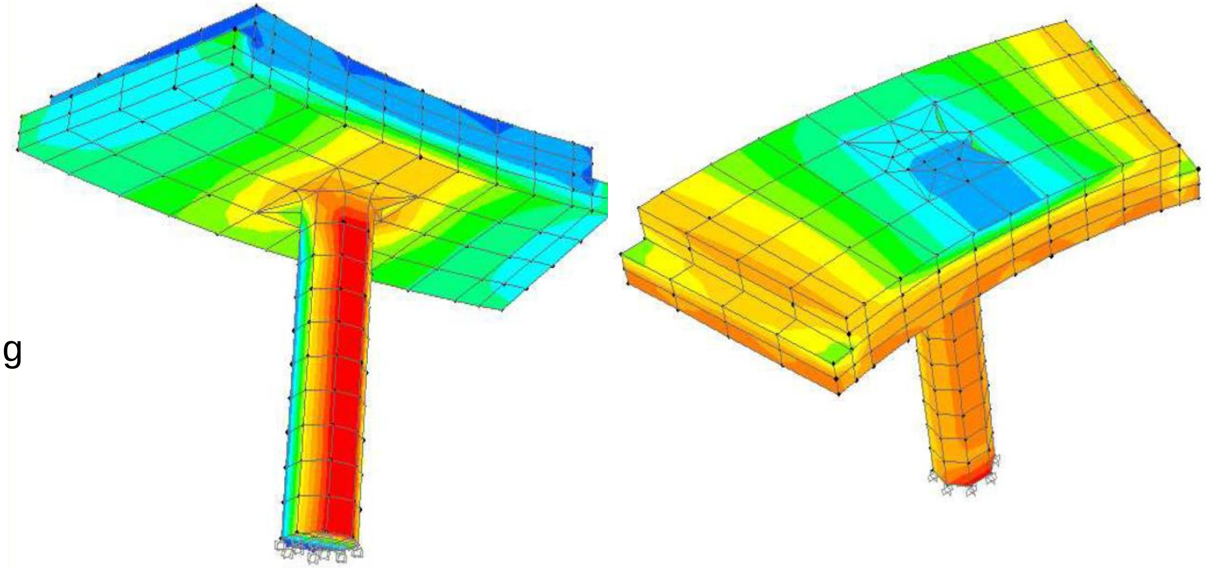
Solid Elements in FE Model



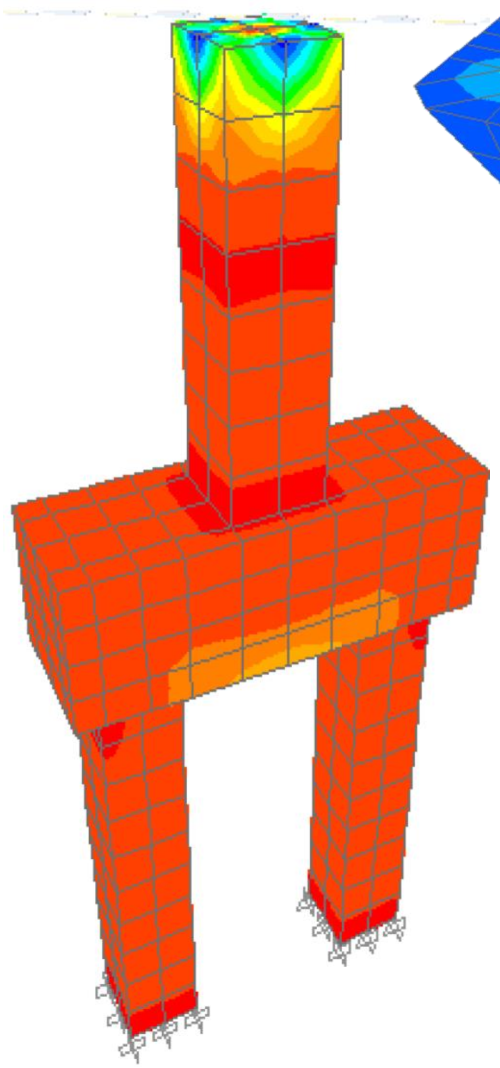
Stress in Pile-Cap (SVMax)



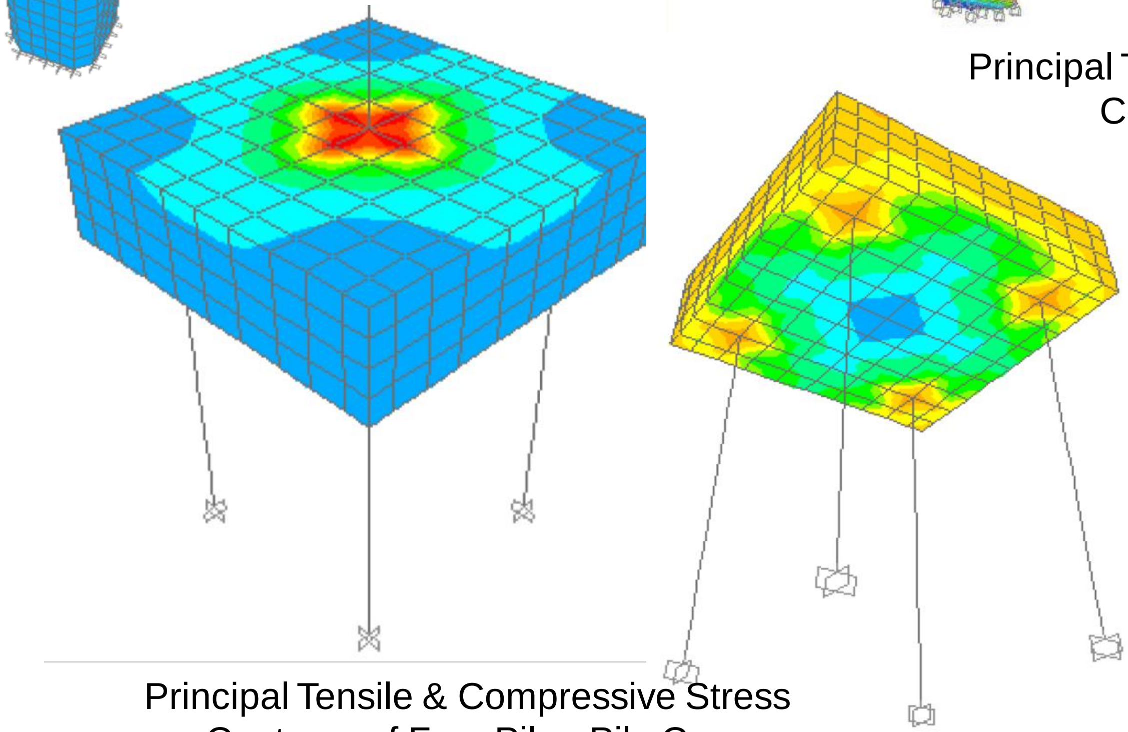
Principal Compressive Stress Contours of Pier Head under Point Loading



Principal Tensile Stress & Compressive Stress Contours of Curved Pier Head

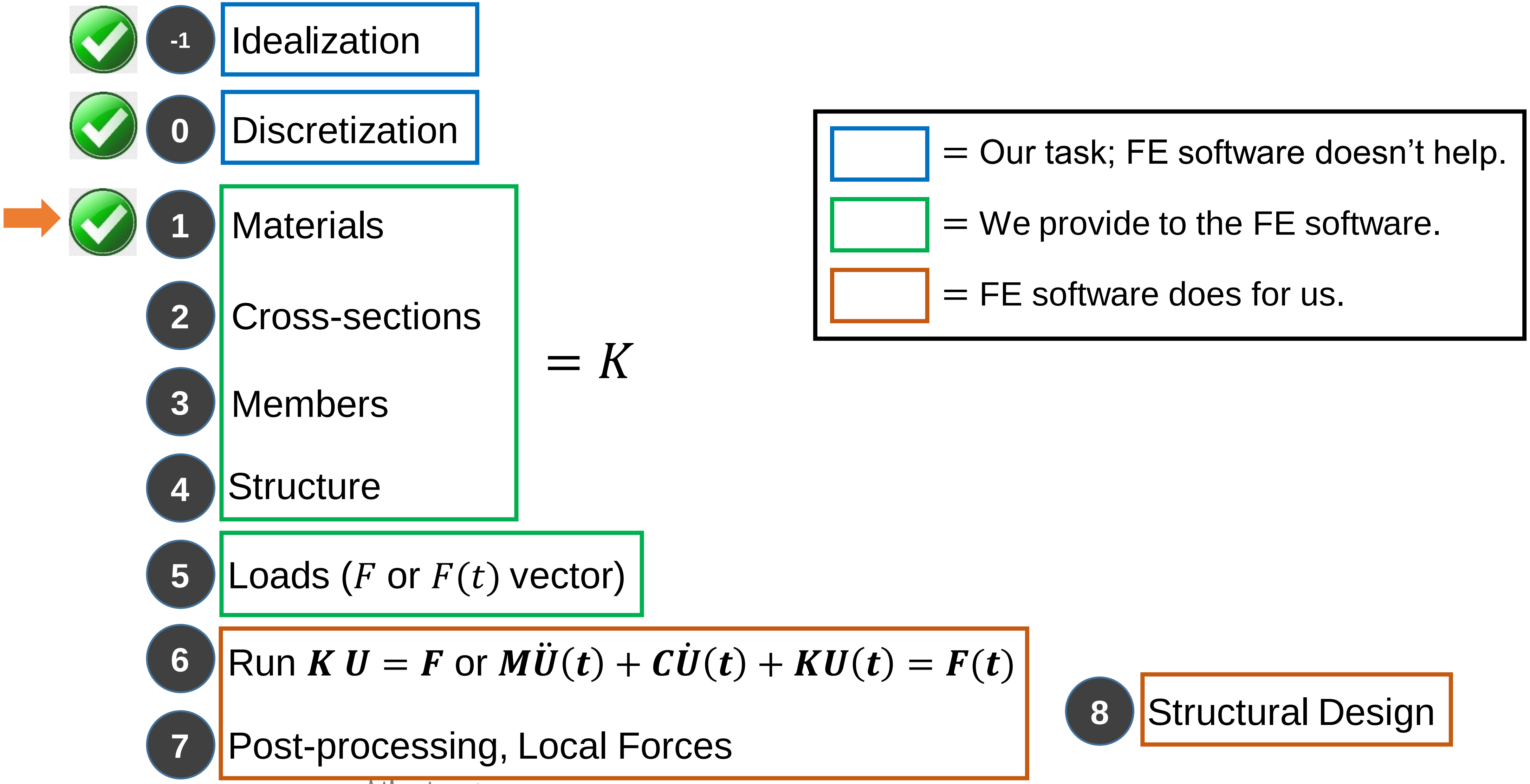


Principal Compressive Stress Contours of Two Piles-Pile Cap



Principal Tensile & Compressive Stress Contours of Four Piles-Pile Cap

Finite Element Modeling, Analysis and Design Process



- ✓ -1 Idealization
- ✓ 0 Discretization
- ➡ ✓ 1 Materials
- 2 Cross-sections
- 3 Members
- 4 Structure

$= K$

- 5 Loads (F or $F(t)$ vector)
- 6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$
- 7 Post-processing, Local Forces
- 8 Structural Design

🟦 = Our task; FE software doesn't help.

🟩 = We provide to the FE software.

🟧 = FE software does for us.

Modeling of Structural Materials

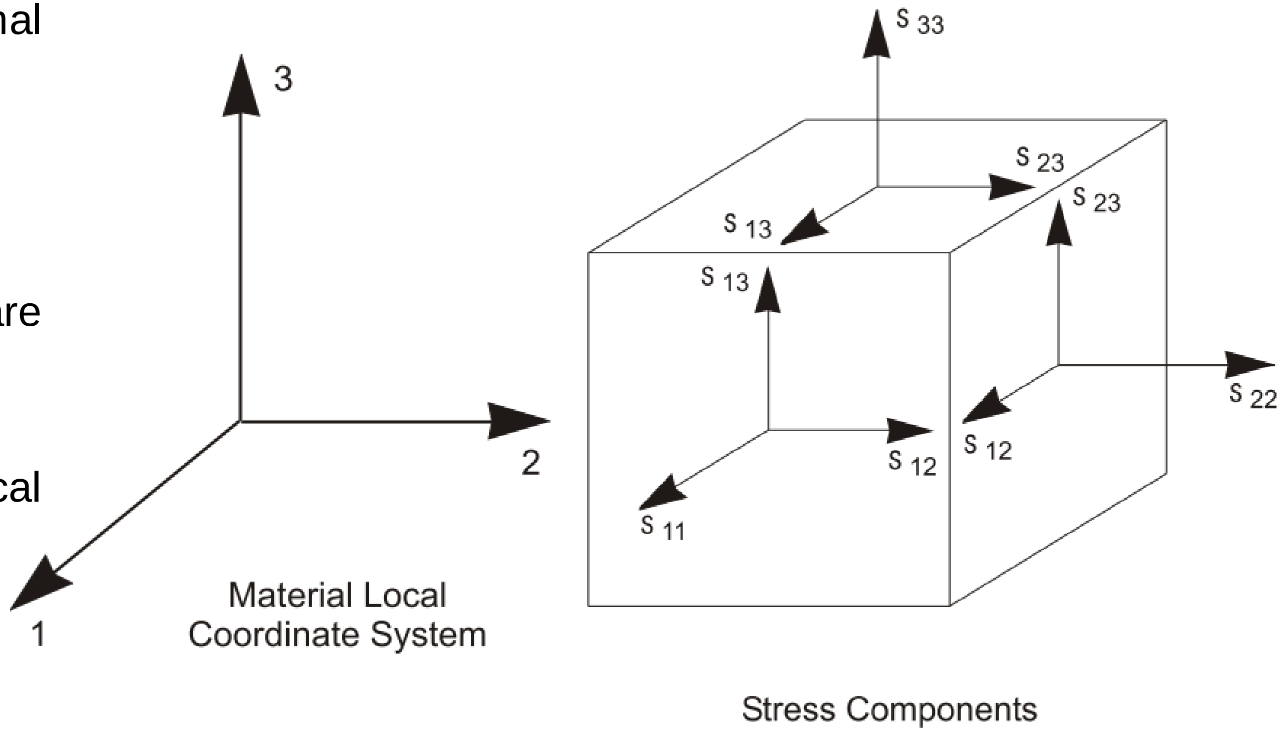
Stresses and Strains – General Notation

- Each material has its own material local coordinate system, which is used to define the elastic and thermal properties.

- Significant only for orthotropic and anisotropic materials.

- The axes of the material local coordinate system are denoted as 1, 2, and 3.

- The material coordinate system is aligned with the local coordinate system for each element.



Definition of Stress Components in the Material Local Coordinate System

Stress and Strain – One Dimensional Case

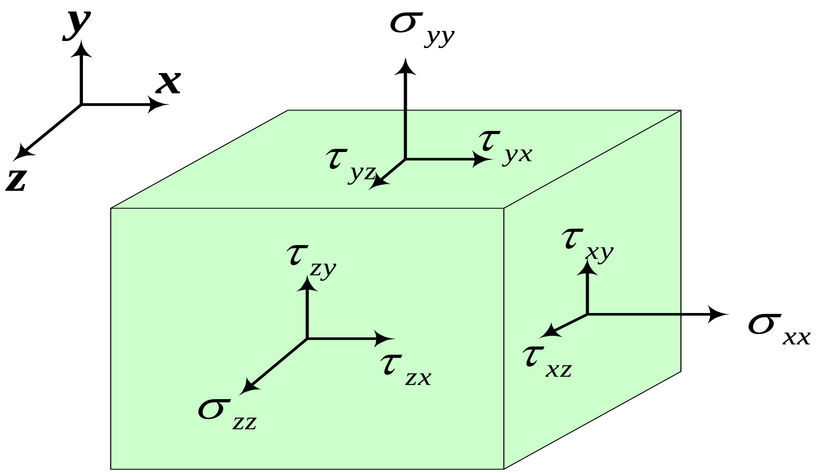
- The Hook's law states that within the elastic limits, the stress is proportional to the strain

$$\sigma = E\ \varepsilon$$

- This is valid for only **very limited cases**
- In reality, the Modulus of Elasticity, **E is NOT a constant**
- There are many stress and strain components, and many properties.

A Bigger Picture of Stress-Strain Components

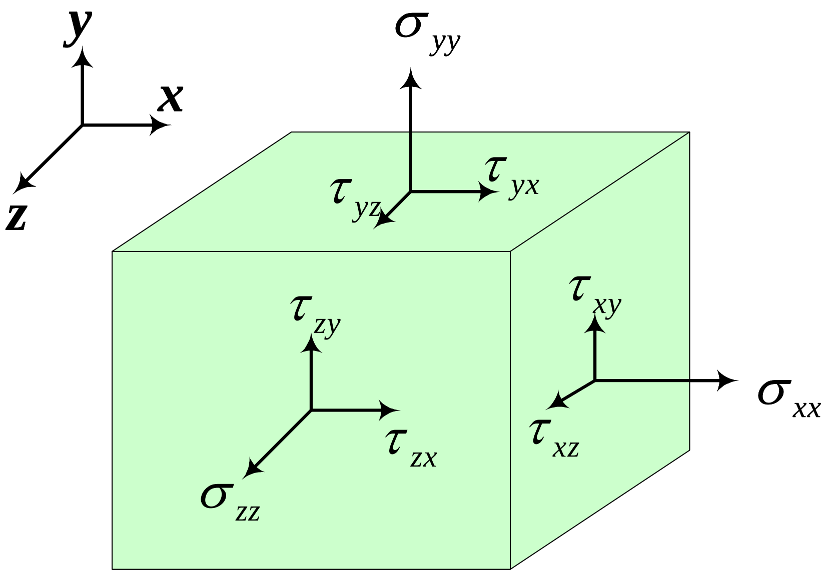
For a general 3D Isotropic Material



At any point in a continuum, or solid, the stress state can be completely defined in terms of six stress components and six corresponding strains.

$$\begin{bmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{zx}\end{bmatrix} = \frac{E}{(1+v)(1-2v)}\begin{bmatrix}1-v & v & v & 0 & 0 & 0\\ v & 1-v & v & 0 & 0 & 0\\ v & v & 1-v & 0 & 0 & 0\\ 0 & 0 & 0 & \frac{1-2v}{2} & 0 & 0\\ 0 & 0 & 0 & 0 & \frac{1-2v}{2} & 0\\ 0 & 0 & 0 & 0 & 0 & \frac{1-2v}{2}\end{bmatrix}\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_z\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{bmatrix}$$

Stress and Strains Components



At any point in a continuum, or solid, the stress state can be completely defined in terms of six stress components and six corresponding strains.

- The Hook's law is a simplified form of Stress-Strain relationship.
- Ultimately, the six stress and strain components can be represented by 3 principal summations.

Directional Behaviors of Materials

Isotropic Materials

- The behavior of an isotropic material is independent of the direction of loading or the orientation of the material.
- Shearing behavior is uncoupled from ex-tensional behavior and is not affected by temperature change.

Orthotropic Materials

- The behavior of an orthotropic material can be different in each of the three local coordinate directions.
- Shearing behavior is un-coupled from extensional behavior and is not affected by temperature change.

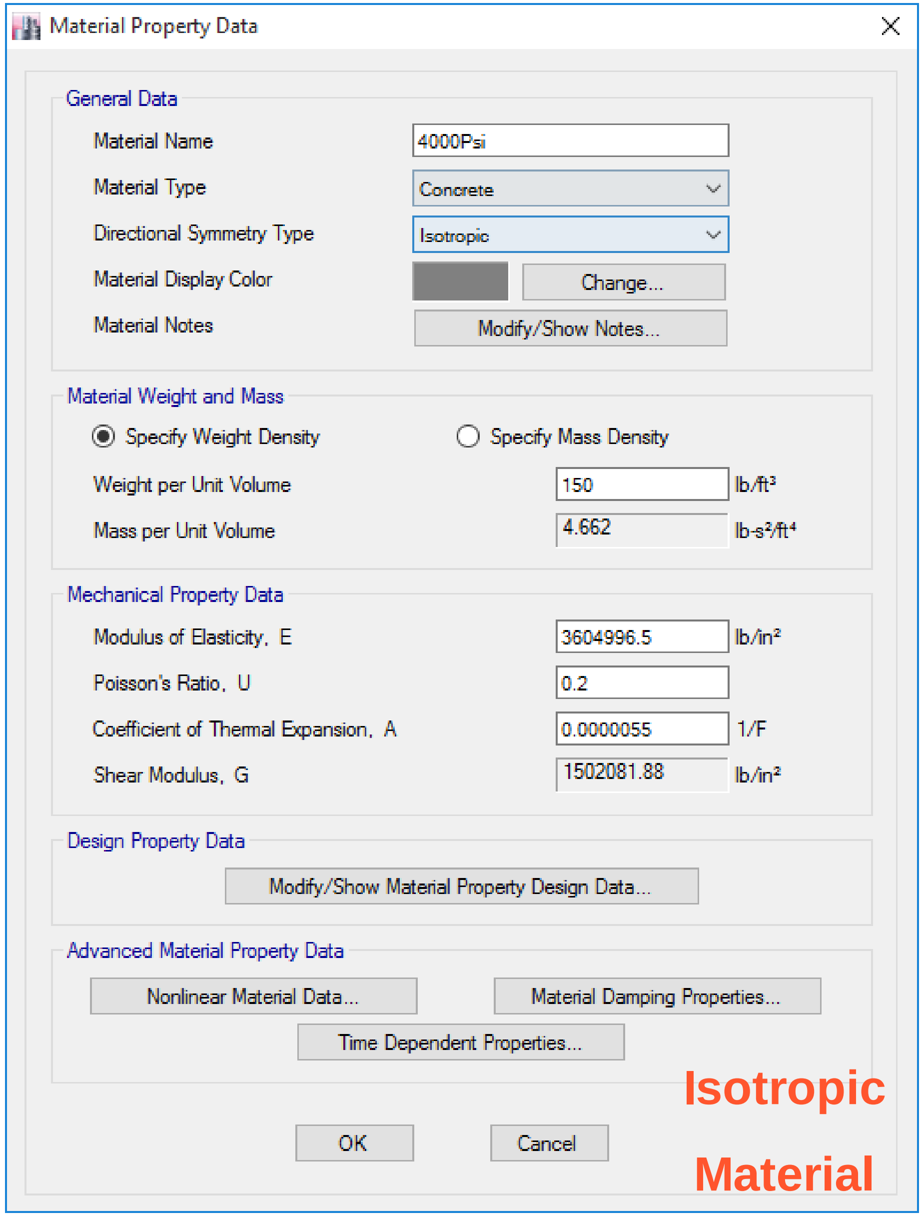
Anisotropic Materials

- The behavior of an anisotropic material can be different in each of the three local coordinate directions
- Shearing behavior can be fully coupled with extensional behavior and can be affected by temperature change.

Basic Properties

- For analysis
  - Modulus of elasticity, E
  - Poisons ratio, ν
  - Shear modulus, G
  - Thermal expansion coefficient, α

- For design
  - Yield stress, F_y
  - Failure stress, F_u, F_c, F_t etc.
  - Yield strain ε_y
  - Failure strain ε_u



Material Property Data

General Data

Material Name: 4000Psi

Material Type: Concrete

Directional Symmetry Type: Isotropic

Material Display Color

Material Notes

Material Weight and Mass

● Specify Weight Density ○ Specify Mass Density

Weight per Unit Volume: 150 lb/ft³

Mass per Unit Volume: 4.662 lb-s²/ft⁴

Mechanical Property Data

Modulus of Elasticity, E: 3604996.5 lb/in²

Poisson's Ratio, U: 0.2

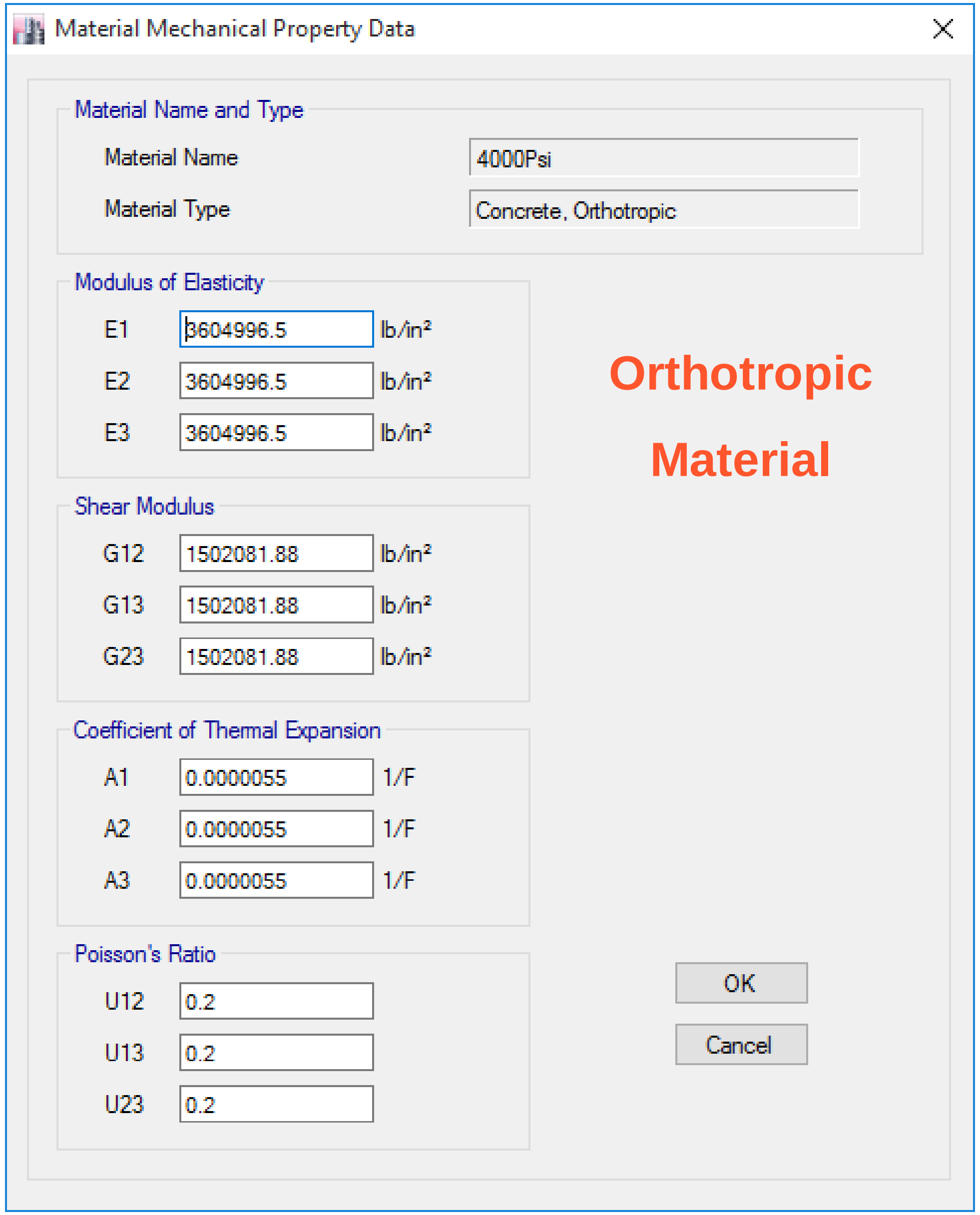
Coefficient of Thermal Expansion, A: 0.0000055 1/F

Shear Modulus, G: 1502081.88 lb/in²

Design Property Data

Advanced Material Property Data

Isotropic Material



Material Mechanical Property Data

Material Name and Type

Material Name: 4000Psi

Material Type: Concrete, Orthotropic

Modulus of Elasticity

E1: 3604996.5 lb/in²

E2: 3604996.5 lb/in²

E3: 3604996.5 lb/in²

Shear Modulus

G12: 1502081.88 lb/in²

G13: 1502081.88 lb/in²

G23: 1502081.88 lb/in²

Coefficient of Thermal Expansion

A1: 0.0000055 1/F

A2: 0.0000055 1/F

A3: 0.0000055 1/F

Poisson's Ratio

U12: 0.2

U13: 0.2

U23: 0.2

Orthotropic Material

Material Modelling for Steel and Concrete

- Modulus of elasticity based on nominal strengths
- Unit weight of materials
- Unit mass of materials
- Poisson’s ratios

Modulus of elasticity of concrete,

$$E_c = 57000\sqrt{f_c{}'} \text{ for } f_c' \leq 6000 \text{ psi}$$

$$E_c = 40000\sqrt{f_c{}'} + 1 \times 10^6 \text{ for } f_c{}' > 6000 \text{ psi}$$

Modulus of elasticity of steel,

$$E_s = 199{,}947 \, MPa$$

Weight per unit volume

- Reinforced concrete = 23.56 kN/m^3
- Steel = 76.97 kN/m^3

Poisson’s ratio

- Reinforced concrete = 0.2
- Steel = 0.3

Other Specific properties

- Relaxation
- Fatigue
- Creep
- Shrinkage
- Confinement based properties

Dependence of Behavior

- Relationship between **Stress and Strain** depends on
  - Basic material composition
  - Initial conditions
  - State of strain
  - Direction of strain
  - History of strain
  - Time since initial strain
  - Temperature
  - Cyclic strain
  - Rate of strain change, velocity and acceleration

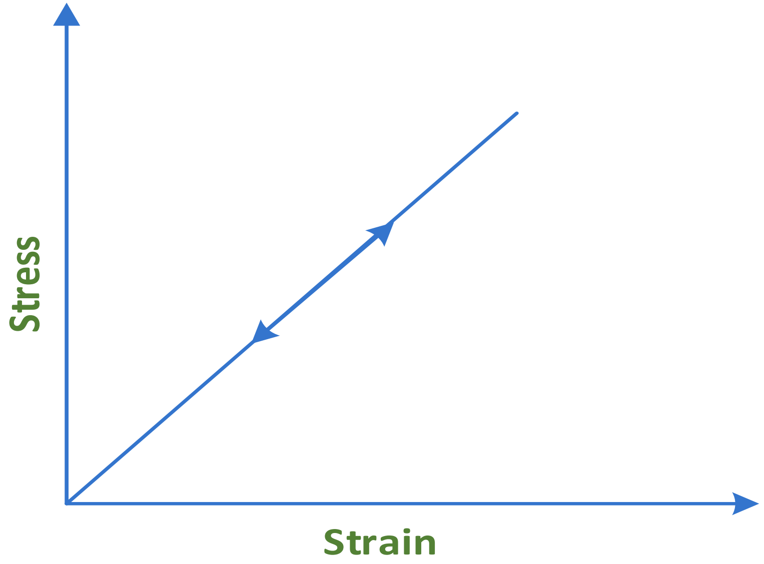
Linearity and Elasticity

- Material behavior depends on level of strain
  - Linear
  - Non-linear

- Material behavior depends on loading history
  - Elastic
  - Plastic
  - Inelastic
  - Hysteretic

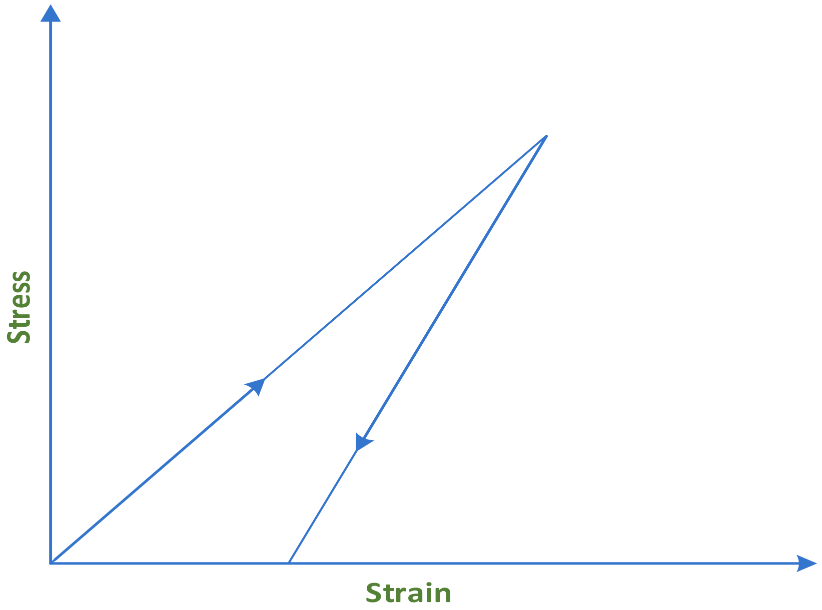
Linear Elastic Material

- A linear elastic material is one in which the strain is proportional to stress
- Both "loading" and "unloading" curves are same (straight lines).



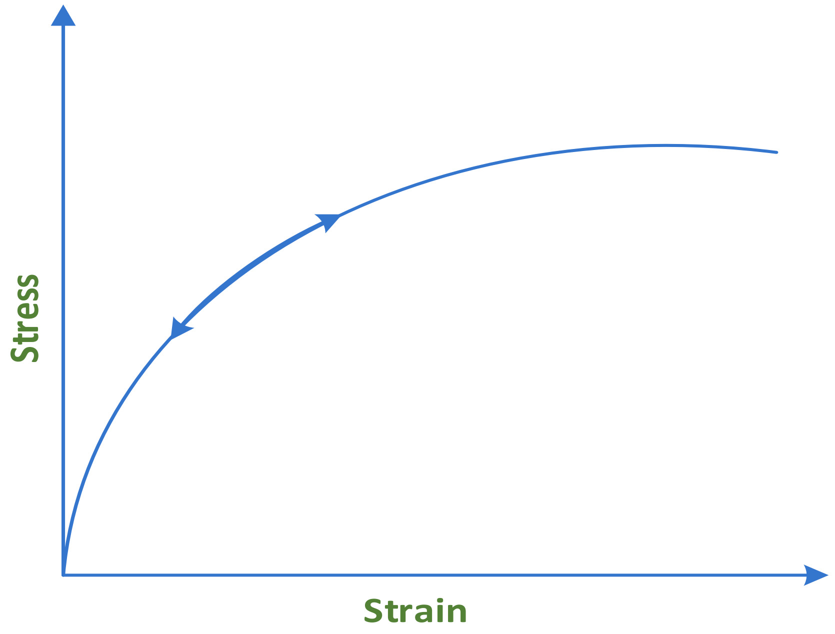
Linear Inelastic Material

- A linear inelastic material is one in which the strain is proportional to stress
- "Loading" and "unloading" curves are not same (although straight lines).



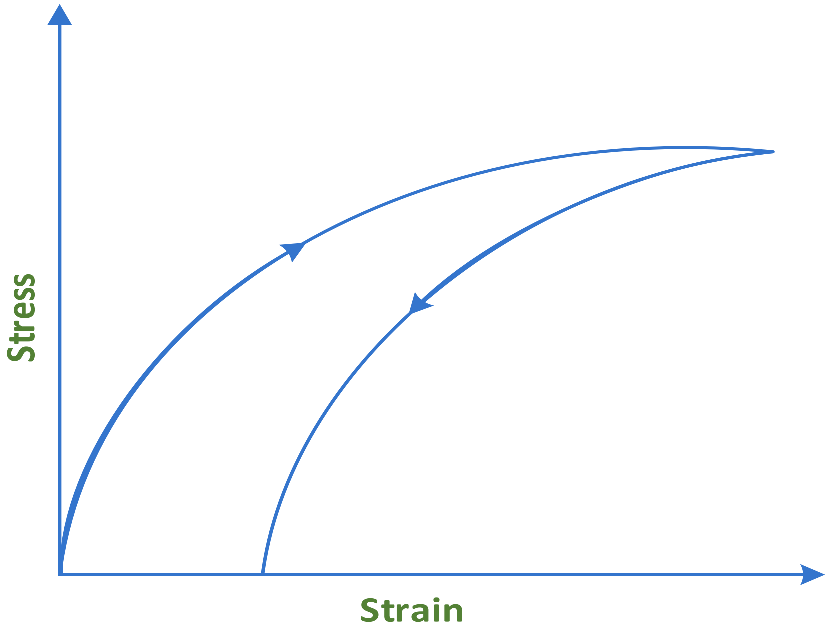
Nonlinear Elastic material

- For a nonlinear elastic material, strain is not proportional to stress as shown in figure.
- Both “loading” and “unloading” curves are same but are not straight lines.



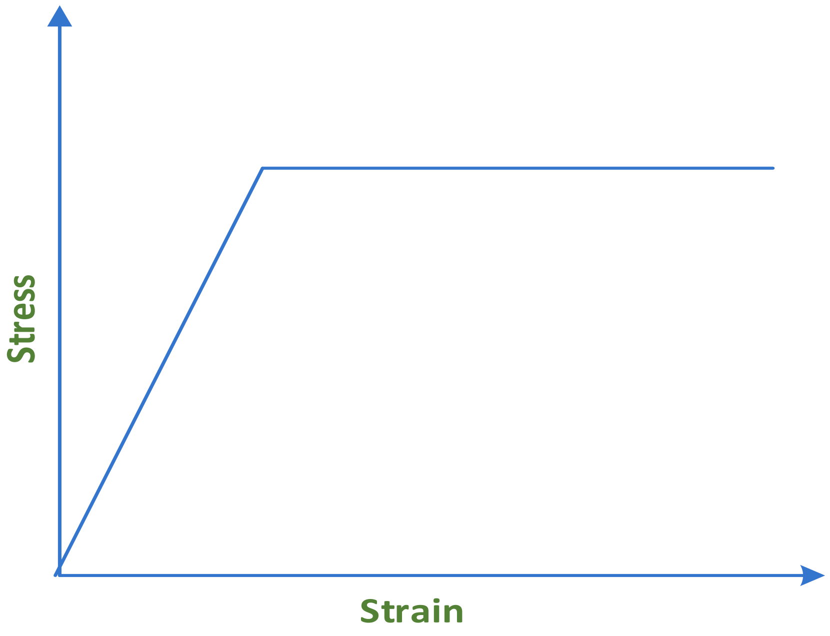
Nonlinear Inelastic Material

- For a nonlinear inelastic material, strain is not proportional to stress as shown in figure.
- “Loading” and “unloading” curves are not same in this case.



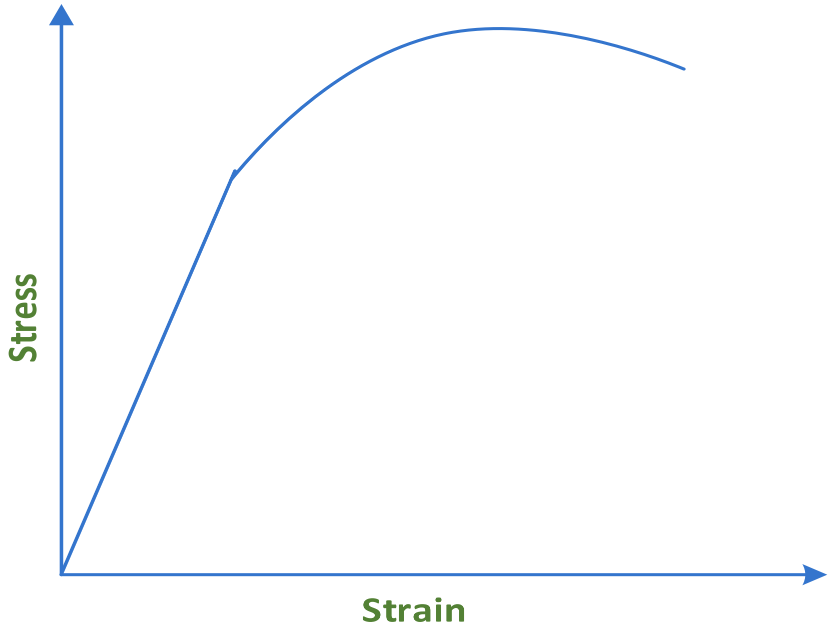
Elastic–Perfectly Plastic (Non-strain Hardening)

- The behavior of an elastic-perfectly plastic (non-strain hardening) material



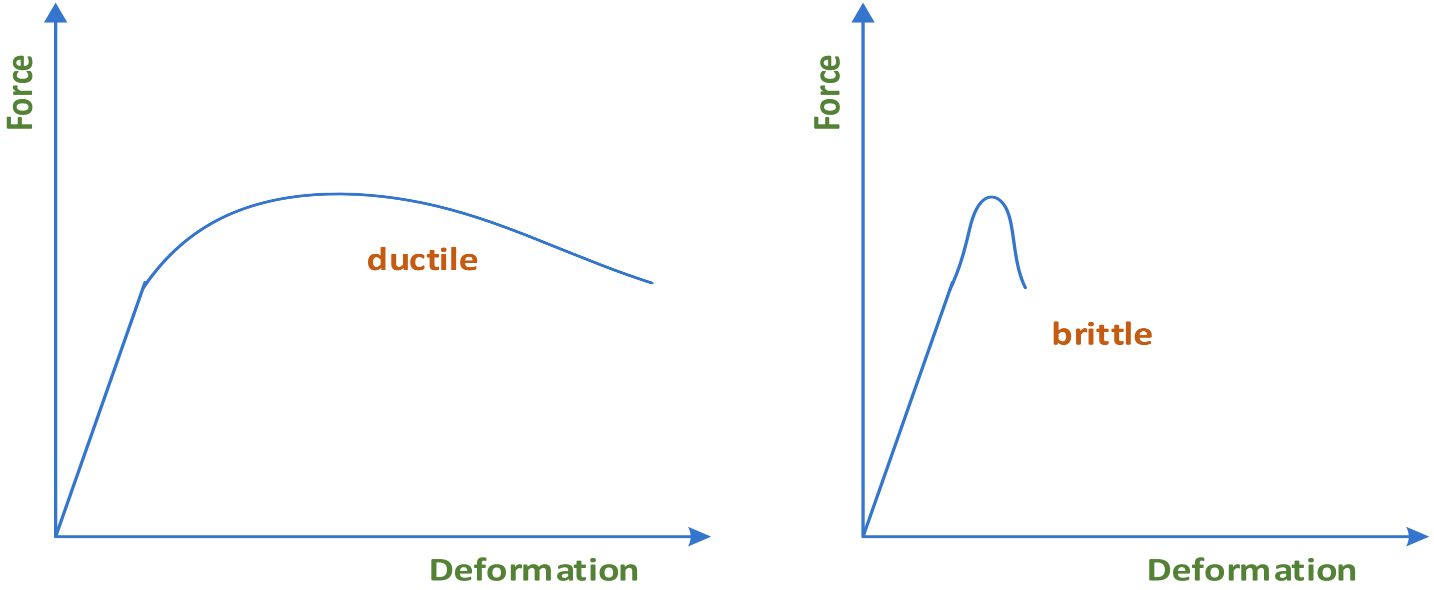
Elastic – Plastic Material

- The elastic plastic material exhibits a stress – strain behavior as depicted in the figure



Ductile and Brittle Materials

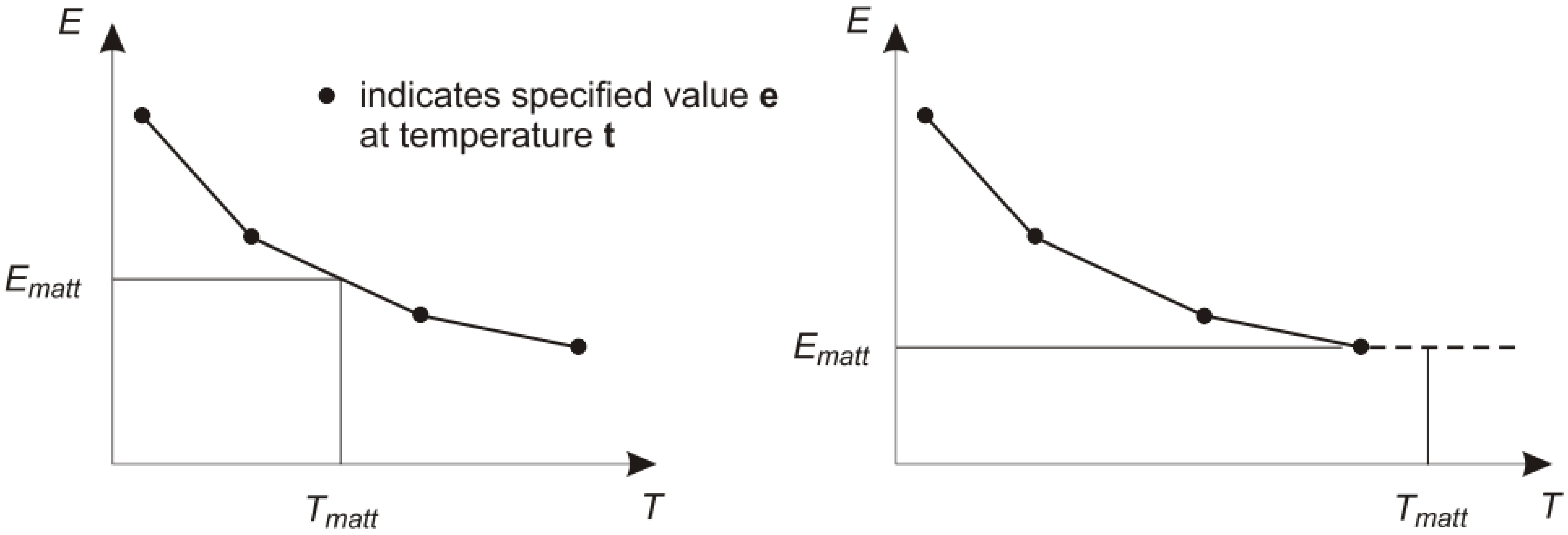
- Ductile materials:
  - able to deform significantly into the inelastic range
- Brittle materials:
  - fail suddenly by cracking or splintering
  - much weaker in tension than in compression



Temperature Dependent Properties

- These properties are given at **a series of specified material temperatures, *t***.
- Properties at other temperatures are obtained by **linear interpolation** between the two nearest specified temperatures.
- Properties at temperatures outside the specified range use the properties at the nearest specified temperature.

Temperature Dependent Properties

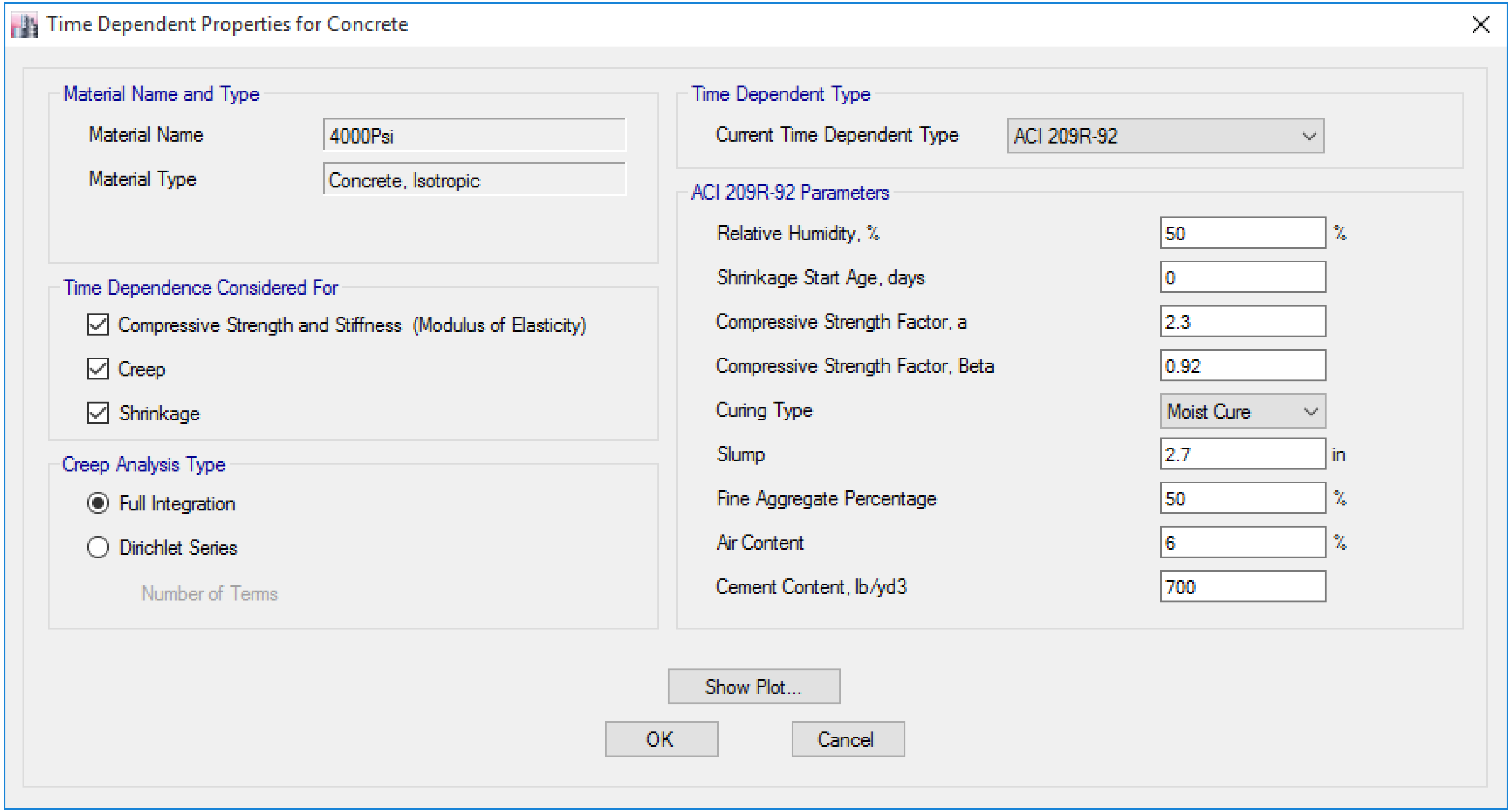


Interpolated Value

Extrapolated Value

Determination of Property E_{matt} at Temperature T_{matt} from Function E(T)

Temperature Dependent Properties



Time Dependent Properties for Concrete

Material Name and Type

Material Name: 4000Psi

Material Type: Concrete, Isotropic

Time Dependence Considered For

☑ Compressive Strength and Stiffness (Modulus of Elasticity)

☑ Creep

☑ Shrinkage

Creep Analysis Type

● Full Integration

○ Dirichlet Series

Number of Terms

Time Dependent Type

Current Time Dependent Type: ACI 209R-92

ACI 209R-92 Parameters

Relative Humidity, %: 50 %

Shrinkage Start Age, days: 0

Compressive Strength Factor, a: 2.3

Compressive Strength Factor, Beta: 0.92

Curing Type: Moist Cure

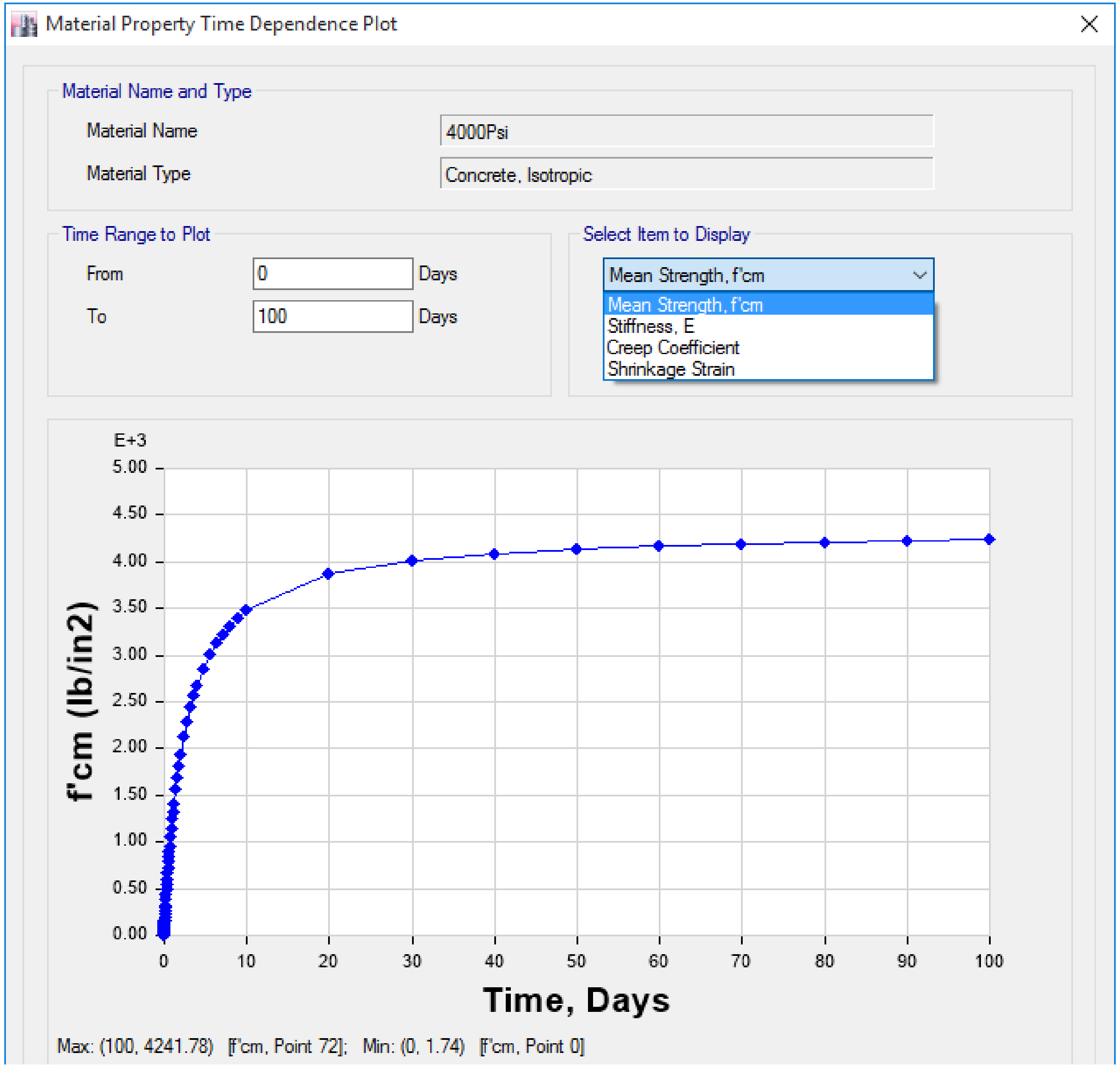
Slump: 2.7 in

Fine Aggregate Percentage: 50 %

Air Content: 6 %

Cement Content, lb/yd3: 700

Temperature Dependent Properties



Material Name: 4000Psi

Material Type: Concrete, Isotropic

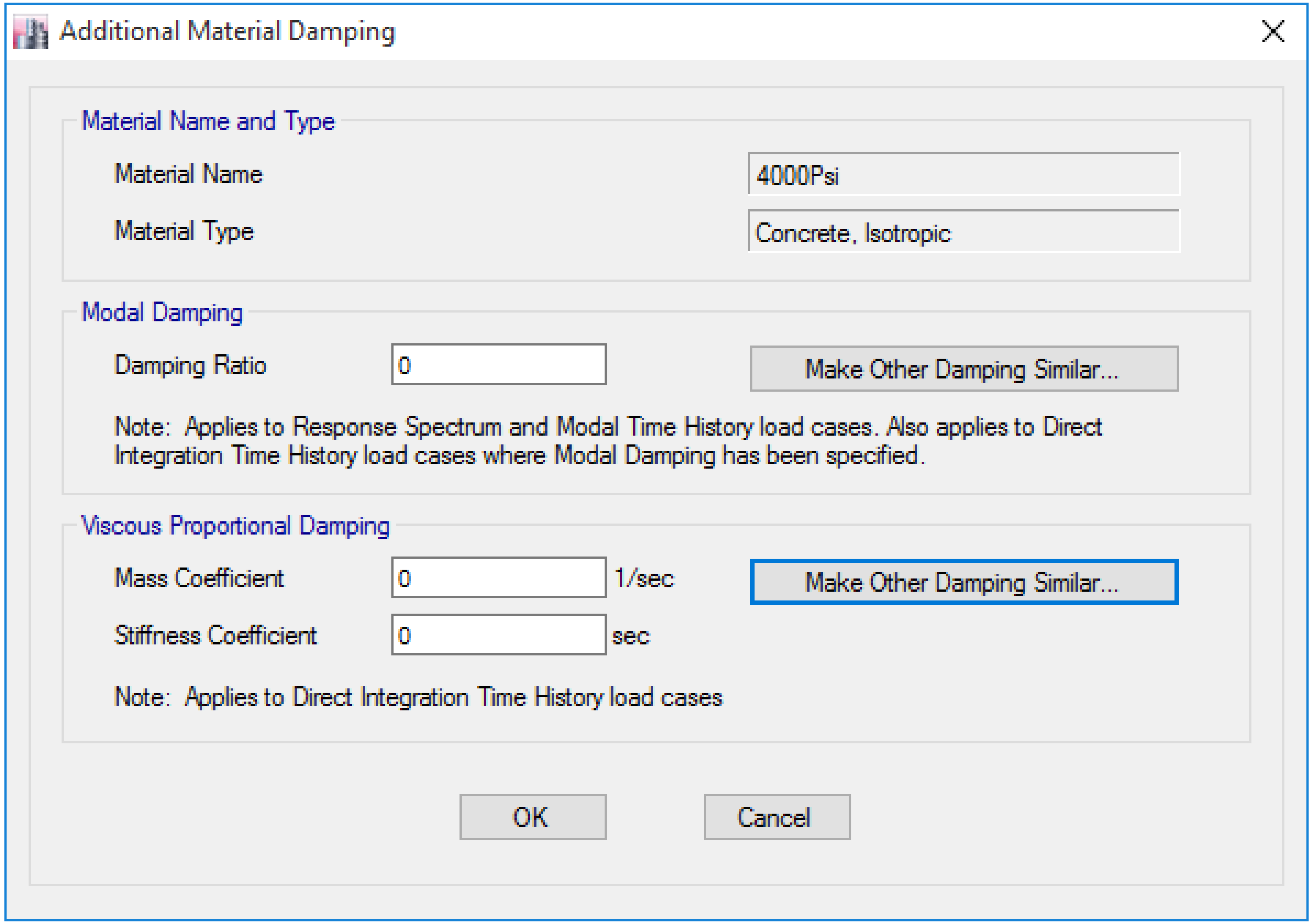
From: 0 Days

To: 100 Days

Select Item to Display: Mean Strength, f'cm

Max: (100, 4241.78) [f'cm, Point 72]; Min: (0, 1.74) [f'cm, Point 0]

Material Damping



Additional Material Damping

Material Name and Type

Material Name: 4000Psi

Material Type: Concrete, Isotropic

Modal Damping

Damping Ratio: 0

Note: Applies to Response Spectrum and Modal Time History load cases. Also applies to Direct Integration Time History load cases where Modal Damping has been specified.

Viscous Proportional Damping

Mass Coefficient: 0 1/sec

Stiffness Coefficient: 0 sec

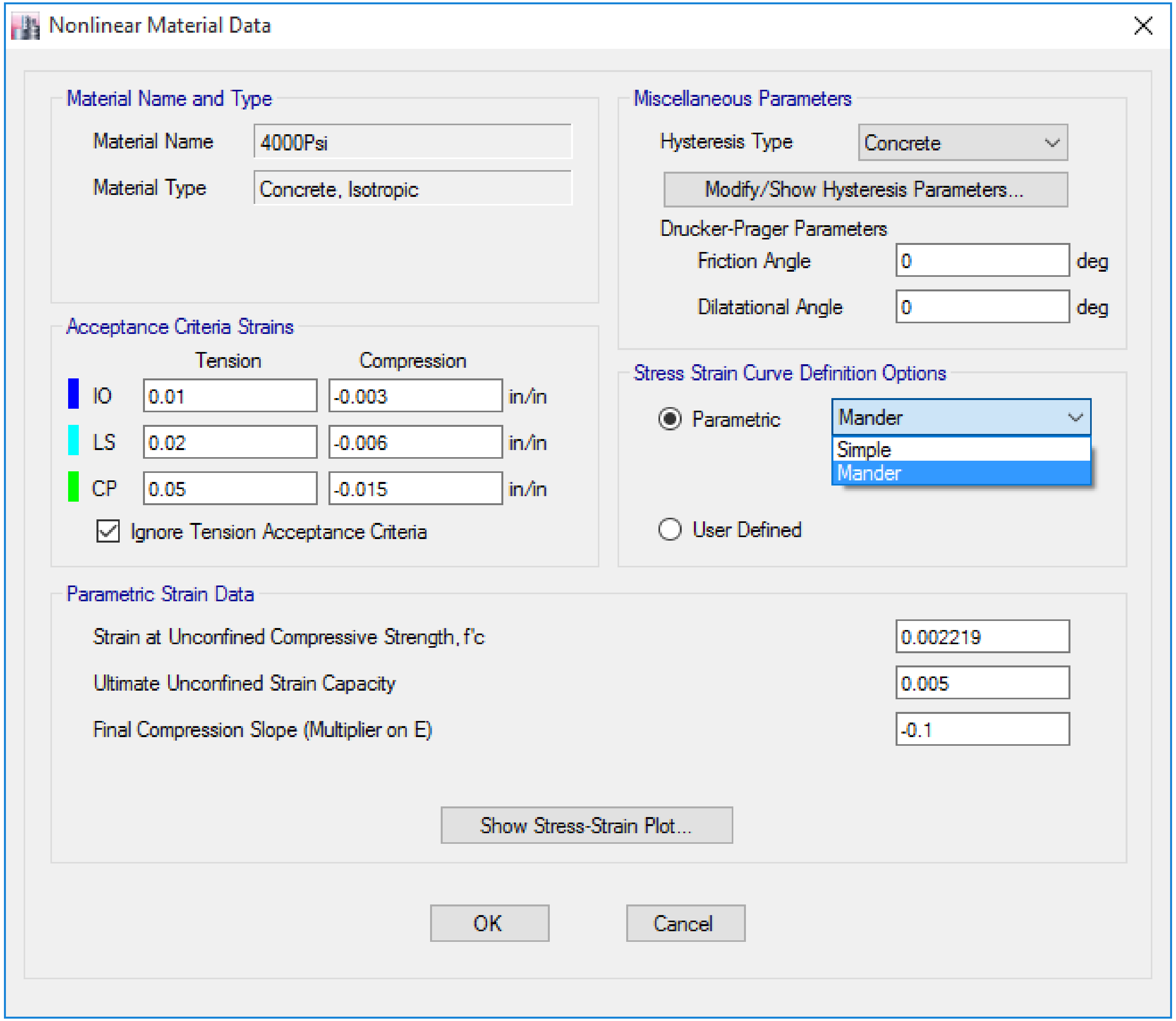
Note: Applies to Direct Integration Time History load cases

Nonlinear Material Properties

Used in the nonlinear modeling of elements using the

- **Fiber Hinges**
- **Layered Shell Element**

Will be discussed in Lecture 6(b): Nonlinear Modeling of Structures



Nonlinear Material Data

Material Name and Type

Material Name: 4000Psi

Material Type: Concrete, Isotropic

Acceptance Criteria Strains

		Tension	Compression	
🟦	IO	0.01	-0.003	in/in
🟦	LS	0.02	-0.006	in/in
🟩	CP	0.05	-0.015	in/in

☑ Ignore Tension Acceptance Criteria

Miscellaneous Parameters

Hysteresis Type: Concrete

Drucker-Prager Parameters

Friction Angle: 0 deg

Dilatational Angle: 0 deg

Stress Strain Curve Definition Options

● Parametric: Mander (Simple, Mander)

○ User Defined

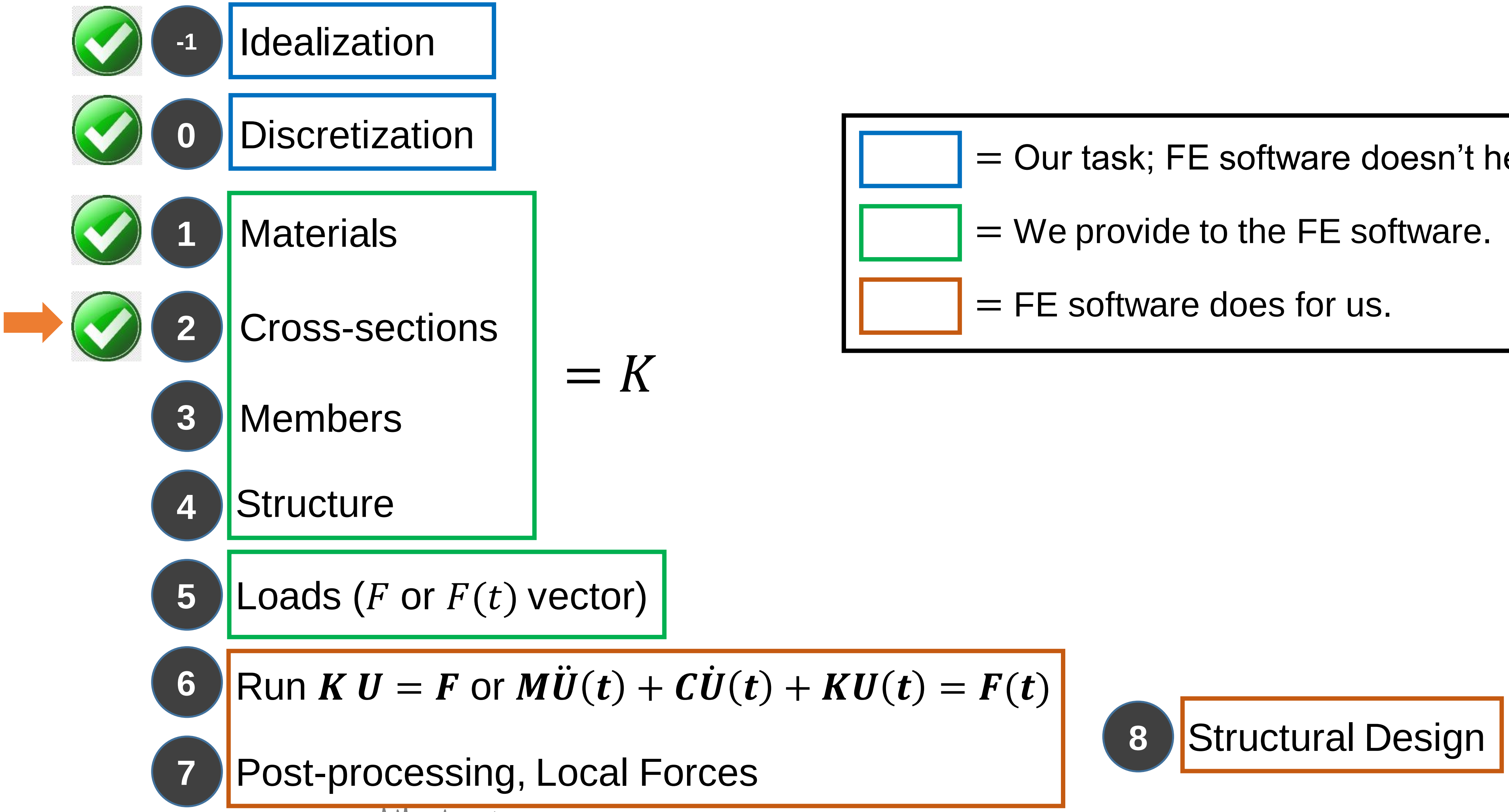
Parametric Strain Data

Strain at Unconfined Compressive Strength, f'c: 0.002219

Ultimate Unconfined Strain Capacity: 0.005

Final Compression Slope (Multiplier on E): -0.1

Finite Element Modeling, Analysis and Design Process



- ✓ -1 Idealization
- ✓ 0 Discretization
- ✓ 1 Materials
- ✓ 2 Cross-sections
- 3 Members
- 4 Structure

$= K$

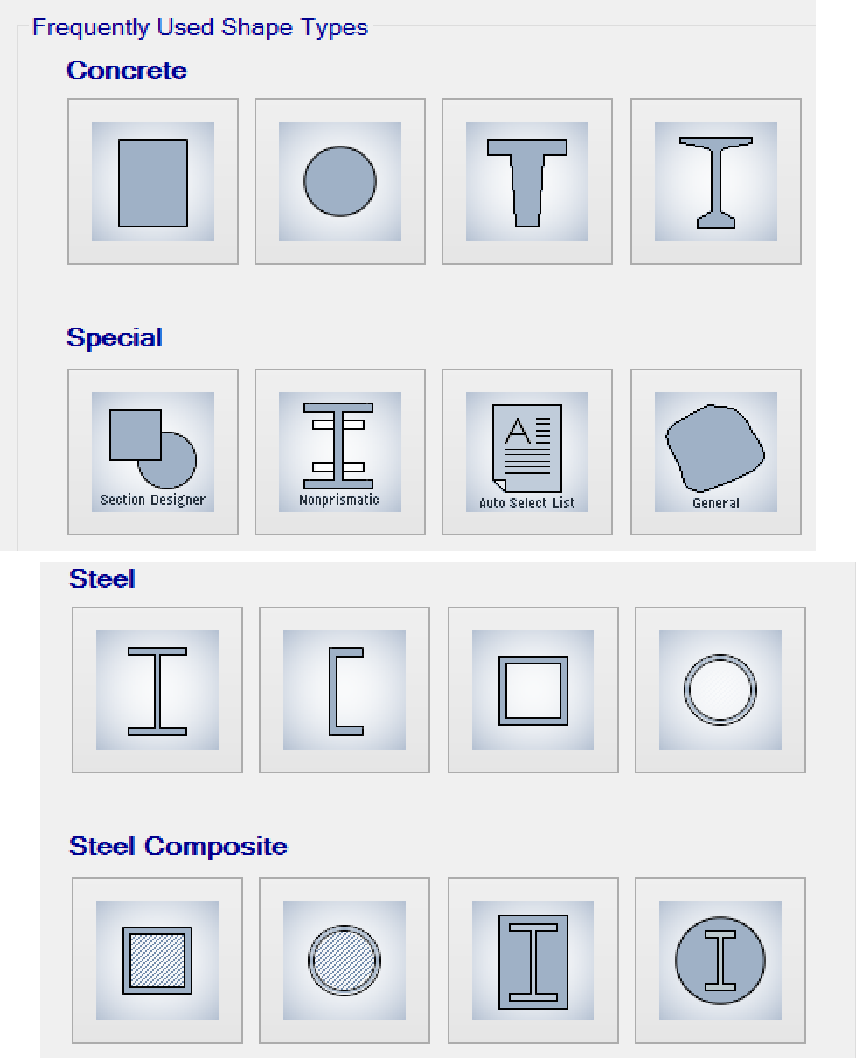
- 5 Loads (F or $F(t)$ vector)
- 6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$
- 7 Post-processing, Local Forces
- 8 Structural Design

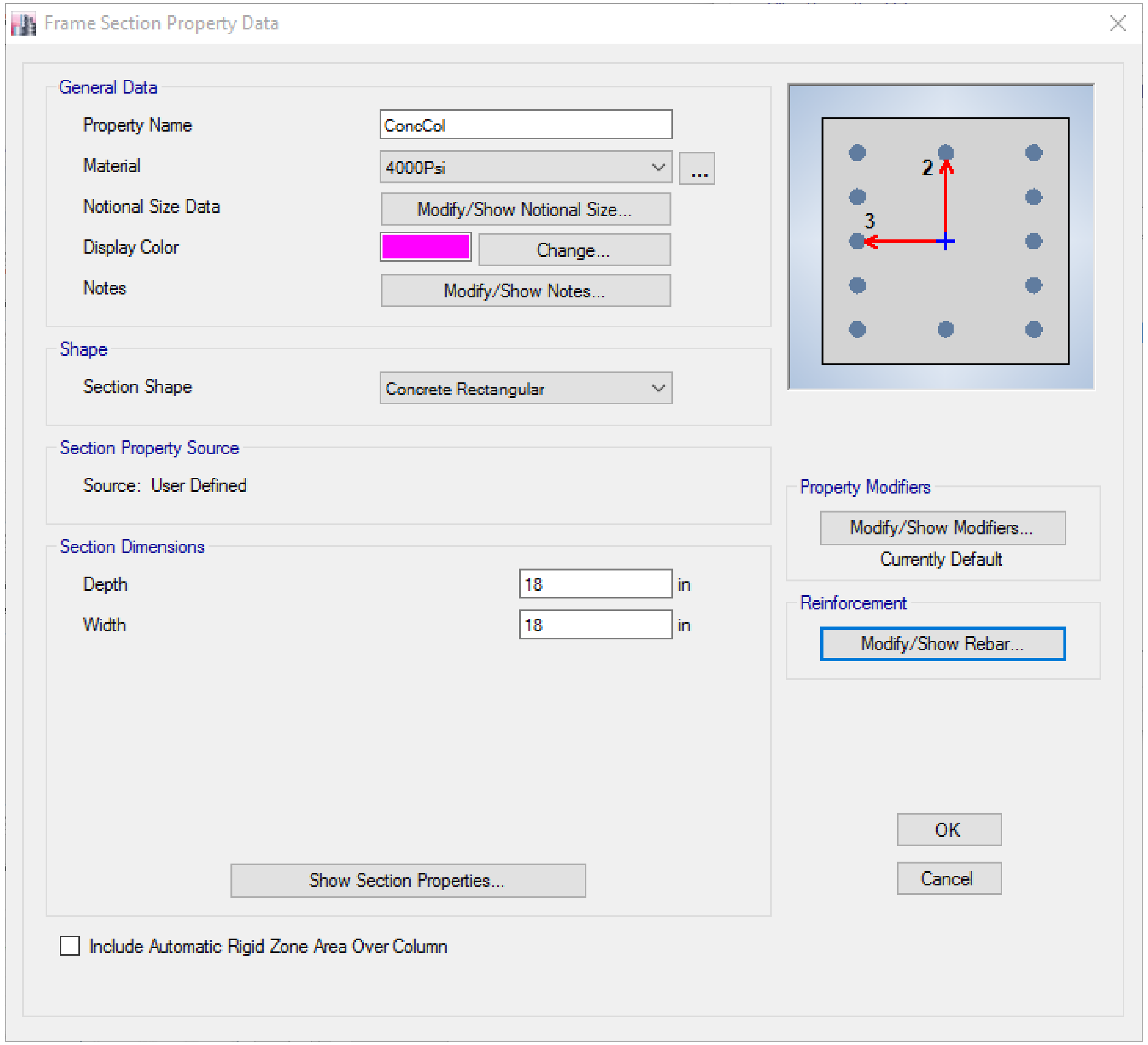
🟦 = Our task; FE software doesn’t help.

🟩 = We provide to the FE software.

🟧 = FE software does for us.

Cross-section Properties of Frames





General Data

Property Name: ConcCol

Material: 4000Psi

Notional Size Data

Display Color

Notes

Shape

Section Shape: Concrete Rectangular

Section Property Source

Source: User Defined

Section Dimensions

Depth: 18 in

Width: 18 in

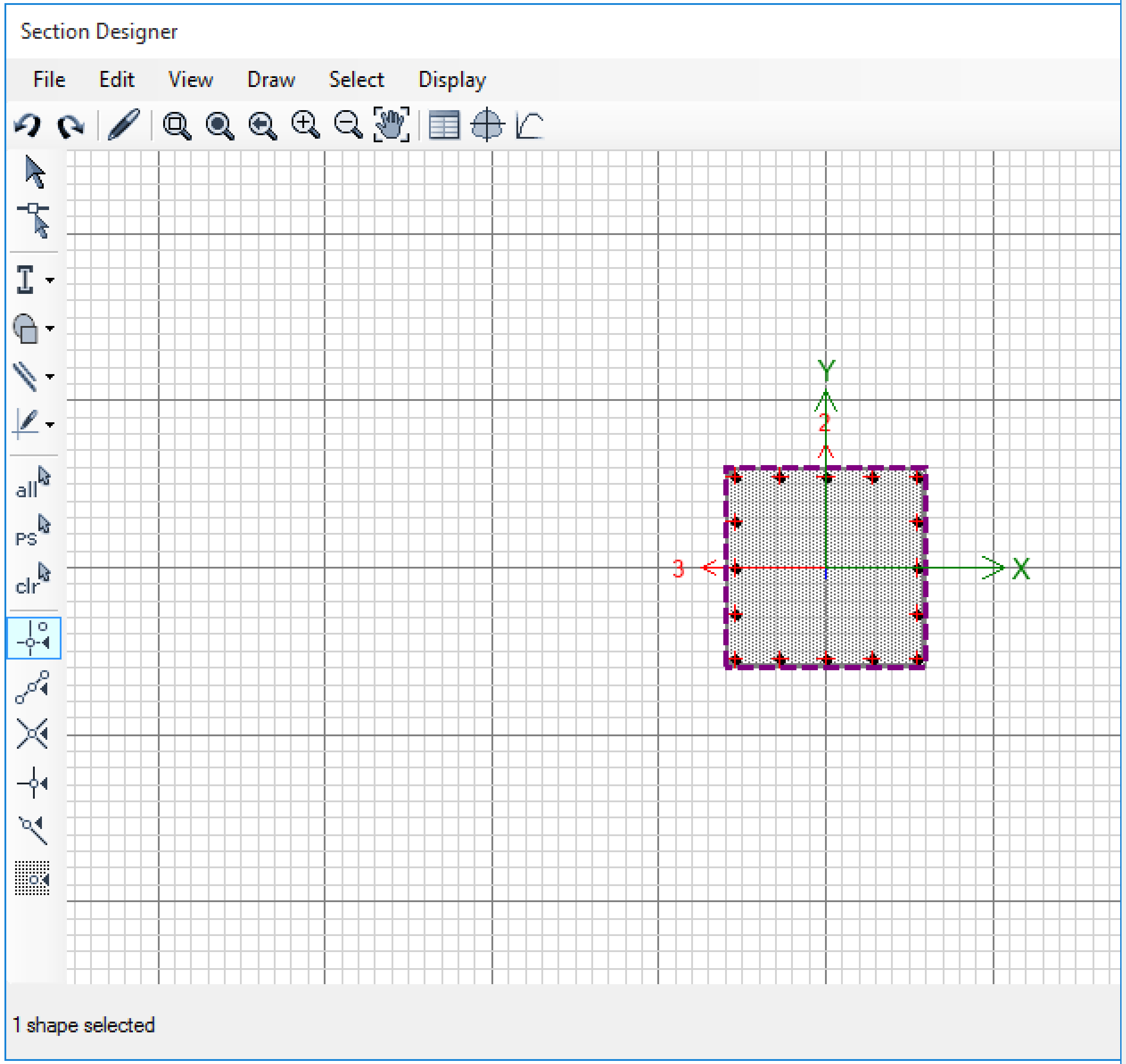
Property Modifiers

Currently Default

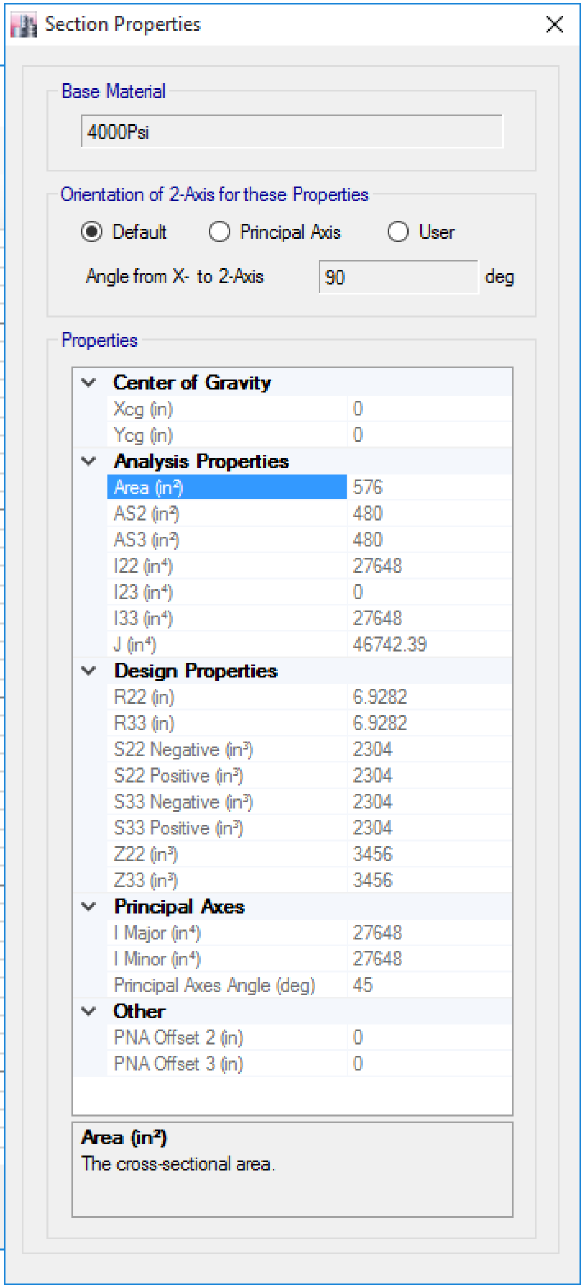
Reinforcement

☐ Include Automatic Rigid Zone Area Over Column

Section Designer



1 shape selected



Base Material: 4000Psi

Orientation of 2-Axis for these Properties

● Default ○ Principal Axis ○ User

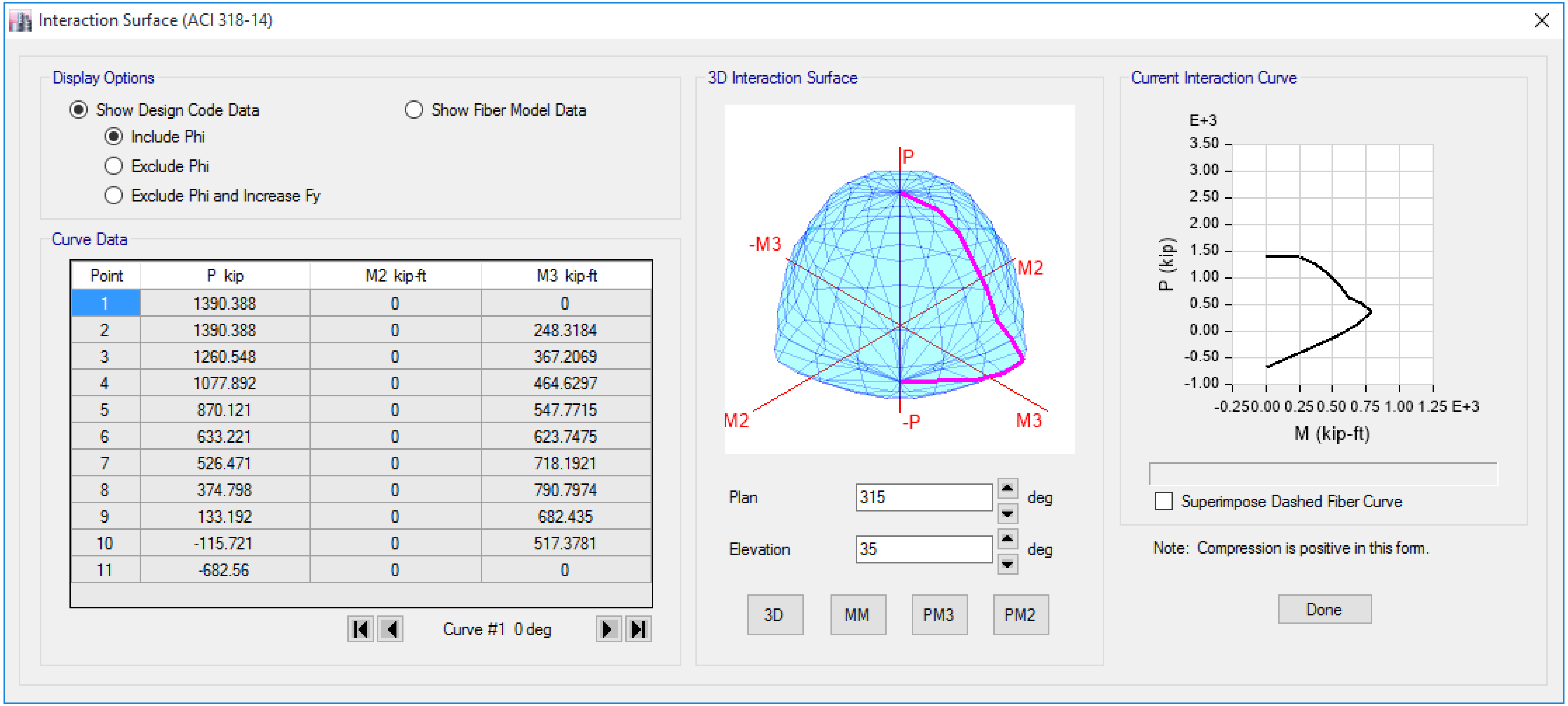
Angle from X- to 2-Axis: 90 deg

Properties

Center of Gravity	
Xcg (in)	0
Ycg (in)	0
Analysis Properties	
Area (in²)	576
AS2 (in²)	480
AS3 (in²)	480
I22 (in⁴)	27648
I23 (in⁴)	0
I33 (in⁴)	27648
J (in⁴)	46742.39
Design Properties	
R22 (in)	6.9282
R33 (in)	6.9282
S22 Negative (in³)	2304
S22 Positive (in³)	2304
S33 Negative (in³)	2304
S33 Positive (in³)	2304
Z22 (in³)	3456
Z33 (in³)	3456
Principal Axes	
I Major (in⁴)	27648
I Minor (in⁴)	27648
Principal Axes Angle (deg)	45
Other	
PNA Offset 2 (in)	0
PNA Offset 3 (in)	0

Area (in²)
The cross-sectional area.

P–M2–M3 Interaction Surface (Capacity Surface)



Interaction Surface (ACI 318-14)

Display Options

● Show Design Code Data

- ● Include Phi
- ○ Exclude Phi
- ○ Exclude Phi and Increase Fy

○ Show Fiber Model Data

Curve Data

Point	P kip	M2 kip-ft	M3 kip-ft
1	1390.388	0	0
2	1390.388	0	248.3184
3	1260.548	0	367.2069
4	1077.892	0	464.6297
5	870.121	0	547.7715
6	633.221	0	623.7475
7	526.471	0	718.1921
8	374.798	0	790.7974
9	133.192	0	682.435
10	-115.721	0	517.3781
11	-682.56	0	0

Curve #1 0 deg

3D Interaction Surface

Plan 315 deg

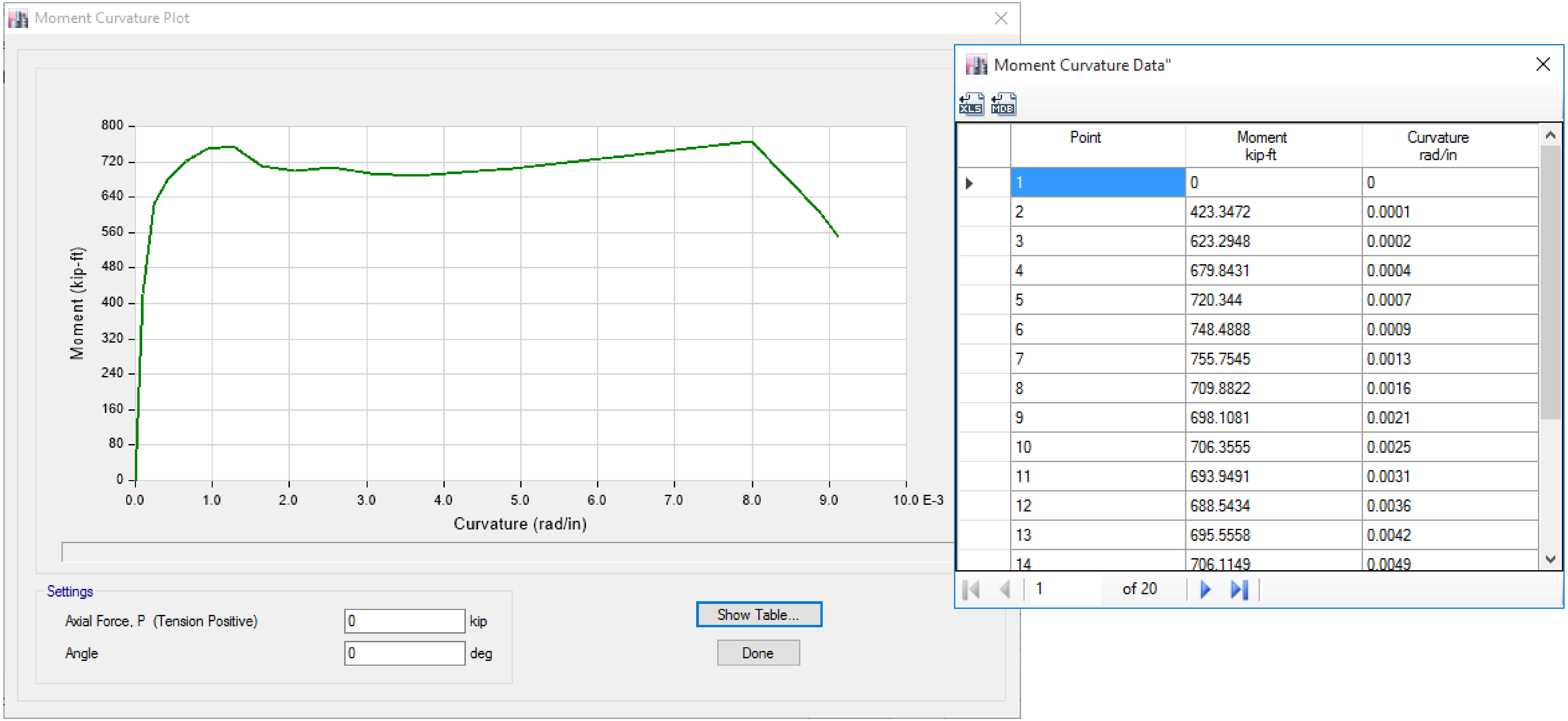
Elevation 35 deg

Current Interaction Curve

☐ Superimpose Dashed Fiber Curve

Note: Compression is positive in this form.

Moment-Curvature Relationship of a Cross-section



Settings

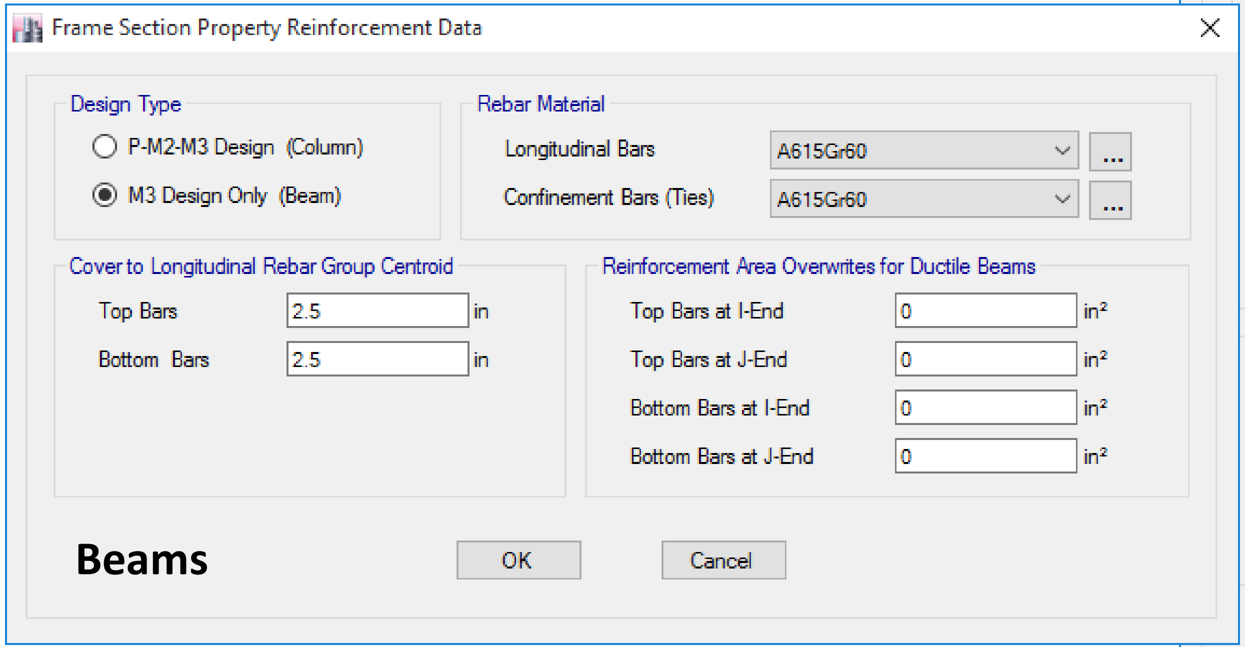
Axial Force, P (Tension Positive) 0 kip

Angle 0 deg

Point	Moment kip-ft	Curvature rad/in
1	0	0
2	423.3472	0.0001
3	623.2948	0.0002
4	679.8431	0.0004
5	720.344	0.0007
6	748.4888	0.0009
7	755.7545	0.0013
8	709.8822	0.0016
9	698.1081	0.0021
10	706.3555	0.0025
11	693.9491	0.0031
12	688.5434	0.0036
13	695.5558	0.0042
14	706.1149	0.0049

1 of 20

Reinforcement Data for Beams (M3) and Columns (P-M2-M3)



Frame Section Property Reinforcement Data

Design Type
- ○ P-M2-M3 Design (Column)
- ● M3 Design Only (Beam)

Rebar Material
- Longitudinal Bars: A615Gr60
- Confinement Bars (Ties): A615Gr60

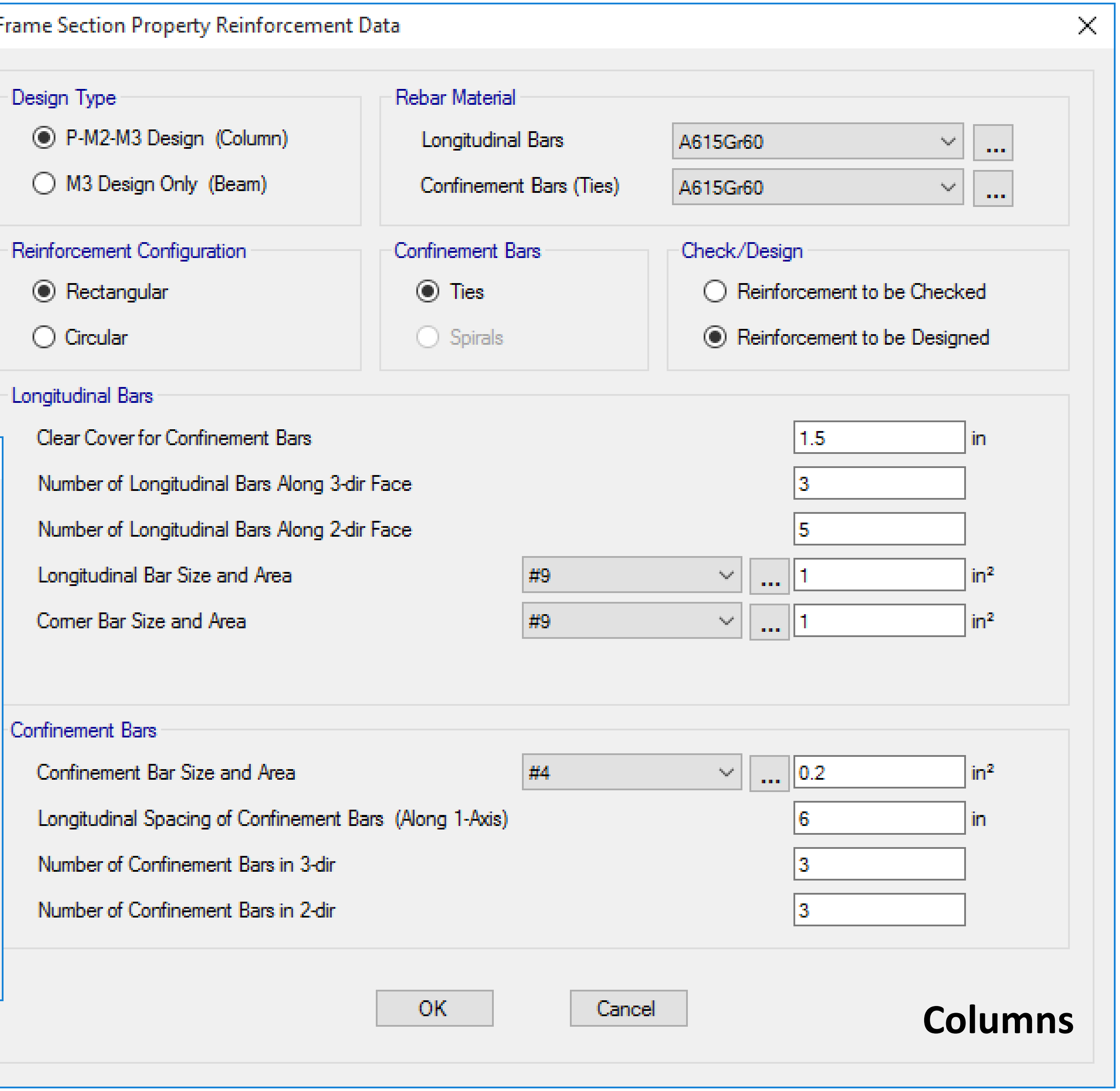
Cover to Longitudinal Rebar Group Centroid

Top Bars	2.5	in
Bottom Bars	2.5	in

Reinforcement Area Overwrites for Ductile Beams

Top Bars at I-End	0	in²
Top Bars at J-End	0	in²
Bottom Bars at I-End	0	in²
Bottom Bars at J-End	0	in²

Beams



Frame Section Property Reinforcement Data

Design Type
- ● P-M2-M3 Design (Column)
- ○ M3 Design Only (Beam)

Rebar Material
- Longitudinal Bars: A615Gr60
- Confinement Bars (Ties): A615Gr60

Reinforcement Configuration
- ● Rectangular
- ○ Circular

Confinement Bars
- ● Ties
- ○ Spirals

Check/Design
- ○ Reinforcement to be Checked
- ● Reinforcement to be Designed

Longitudinal Bars

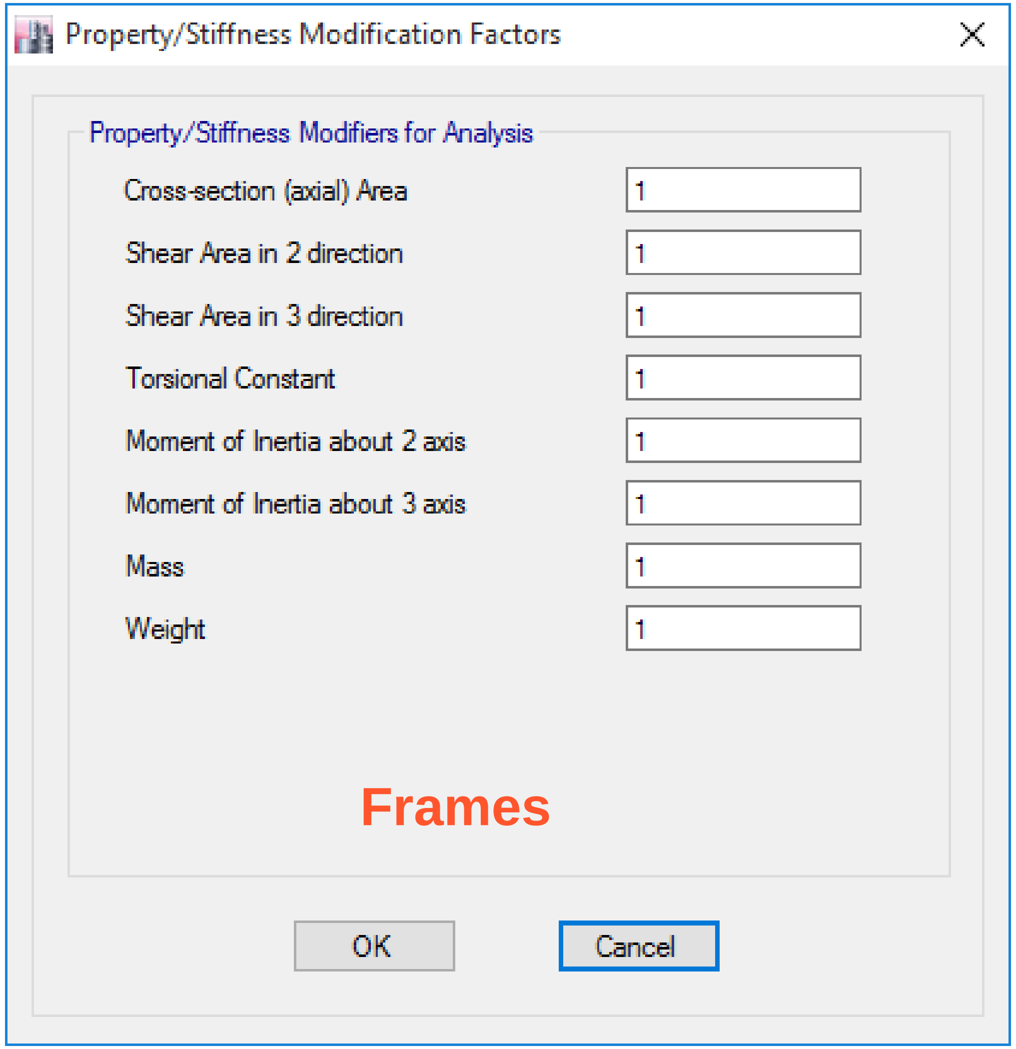
Clear Cover for Confinement Bars		1.5	in
Number of Longitudinal Bars Along 3-dir Face		3	
Number of Longitudinal Bars Along 2-dir Face		5	
Longitudinal Bar Size and Area	#9	1	in²
Corner Bar Size and Area	#9	1	in²

Confinement Bars

Confinement Bar Size and Area	#4	0.2	in²
Longitudinal Spacing of Confinement Bars (Along 1-Axis)		6	in
Number of Confinement Bars in 3-dir		3	
Number of Confinement Bars in 2-dir		3	

Columns

Stiffness Modifiers for Cracked Sections

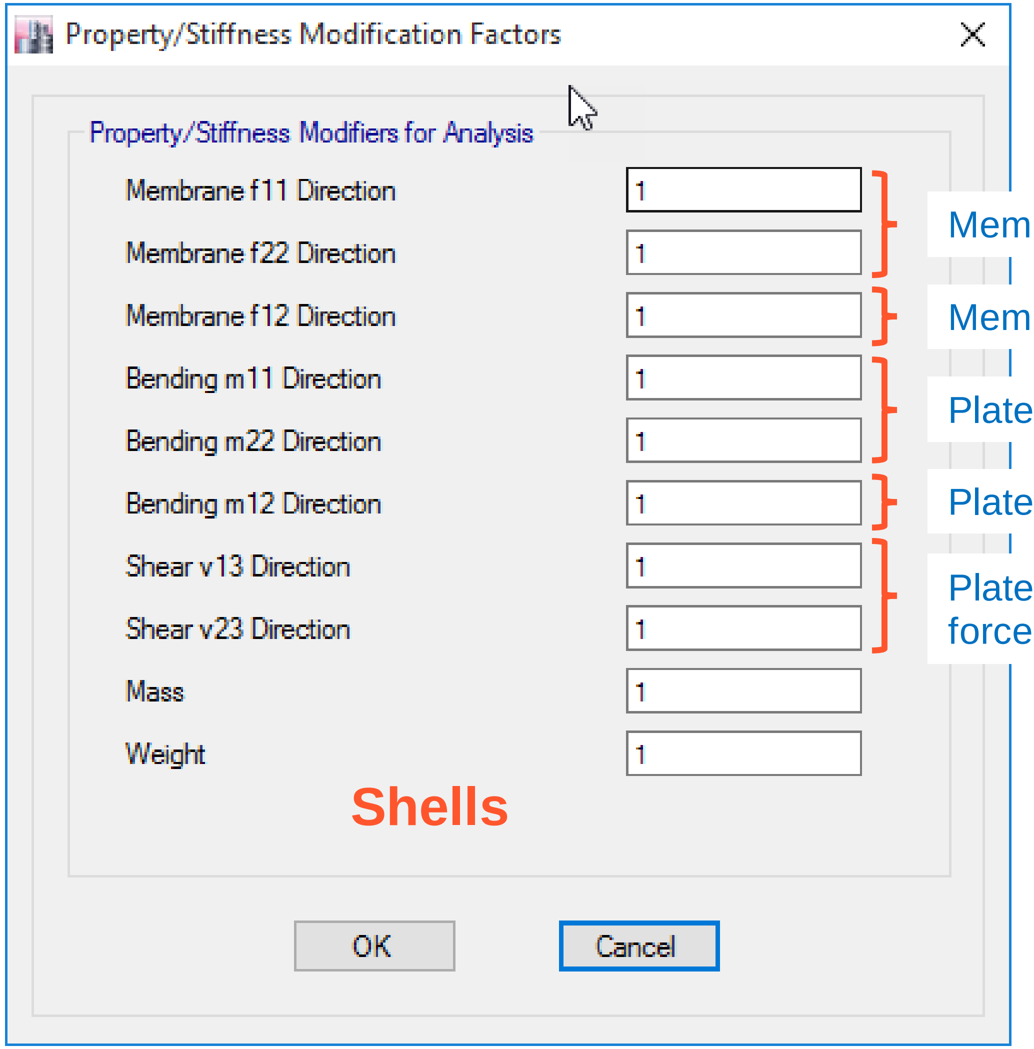


Property/Stiffness Modification Factors

Property/Stiffness Modifiers for Analysis

Cross-section (axial) Area	1
Shear Area in 2 direction	1
Shear Area in 3 direction	1
Torsional Constant	1
Moment of Inertia about 2 axis	1
Moment of Inertia about 3 axis	1
Mass	1
Weight	1

Frames



Property/Stiffness Modification Factors

Property/Stiffness Modifiers for Analysis

Membrane f11 Direction	1
Membrane f22 Direction	1
Membrane f12 Direction	1
Bending m11 Direction	1
Bending m22 Direction	1
Bending m12 Direction	1
Shear v13 Direction	1
Shear v23 Direction	1
Mass	1
Weight	1

Shells

Membrane direct forces

Membrane shear force

Plate bending moments

Plate twisting moment

Plate transverse shear forces

Cracked-section Properties for Shear Walls

- Flexural and axial behaviours are modified by **either f11 or f22** depending on the orientation of the local axis and the shear behaviour is controlled by f12.

- In ETABS, the default is to have the 1-axis horizontal and the 2-axis vertical. This means that the **flexural modifier for EI should be applied to f22 for wall piers**.

- **Generally not designed for out-of-plane bending** to avoid excessive longitudinal reinforcement. Use a small modifier of 0.25 for m11, m22 and m12 so numerical instabilities could be avoided.

Stiffness Assumptions in Finite Element Models for Strength Design

Source: Mr. Thaung Htut Aung (AIT Solutions, AIT)

Concrete Element	Seismic	ETABS
Core walls/shear walls	Flexural (In-plane) – 0.7 I_g Flexural (Out-of-plane) – 0.25 I_g Shear (In-plane) – 1.0 A_g	f22 = 0.7 m11 = m22 = m12 = 0.25
Basement walls	Flexural (In-plane) – 0.8 I_g Shear (In-plane) – 0.5 A_g	m11 = m22 = m12 = 0.8 f12 = 0.5
Coupling beams (Diagonal-reinforced)	Flexural – 0.07 (L/h) $I_g \leq 0.3 I_g$ Shear – 1.0 A_g	I33 = 0.07 (L/h) ≤ 0.3
Coupling beams (Conventional-reinforced)	Flexural – 0.07 (L/h) $I_g \leq 0.3 I_g$ Shear – 1.0 A_g	I33 = 0.07 (L/h) ≤ 0.3
Transfer diaphragms (In-plane)	Flexural – 0.1 I_g Shear – 0.1 A_g	f11 = f22 = 0.1 f12 = 0.1
Tower diaphragms (In-plane)	Flexural – 0.5 I_g Shear – 0.5 A_g	f11 = f22 = 0.5 f12 = 0.5
Moment frame beams	Flexural – 0.35 I_g Shear – 1.0 A_g	I33 = 0.35
Columns	Flexural – 0.7 I_g Shear – 1.0 A_g	I22 = I33 = 0.7
Flat slabs (Out-of-plane) Slab with beams (Out-of-plane)	Flexural – 0.1 I_g Flexural – 0.35 I_g	m11 = m22 = m12 = 0.1 m11 = m22 = m12 = 0.35

Table A.8.4—Effective stiffness values[1]

Component		Axial	Flexural	Shear
Beams	nonprestressed	$1.0E_cA_g$	$0.3E_cI_g$	$0.4E_cA_g$
	prestressed	$1.0E_cA_g$	$1.0E_cI_g$	$0.4E_cA_g$
Columns with compression caused by design gravity loads[2]	$\geq 0.5A_gf_c'$	$1.0E_cA_g$	$0.7E_cI_g$	$0.4E_cA_g$
	$\leq 0.1A_gf_c'$ or with tension	$1.0E_cA_g$ (compression) $1.0E_sA_{st}$ (tension)	$0.3E_cI_g$	$0.4E_cA_g$
Structural walls[3]	in-plane	$1.0E_cA_g$	$0.35E_cI_g$	$0.2E_cA_g$
	out-of-plane	$1.0E_cA_g$	$0.25E_cI_g$	$0.4E_cA_g$
Diaphragms (in-plane only)[4]	nonprestressed	$0.25E_cA_g$	$0.25E_cI_g$	$0.25E_cA_g$
	prestressed	$0.5E_cA_g$	$0.5E_cI_g$	$0.4E_cA_g$
Coupling beams	with or without diagonal reinforcement	$1.0E_cA_g$	$0.07\left(\frac{\ell_n}{h}\right)E_cI_g$ $\leq 0.3E_cI_g$	$0.4E_cA_g$
Mat foundations	in-plane	$0.5E_cA_g$	$0.5E_cI_g$	$0.4E_cA_g$
	out-of-plane[5]		$0.5E_cI_g$	

[1] Tabulated values for axial, flexural, and shear shall be applied jointly in defining effective stiffness of an element, unless alternative combinations are justified.

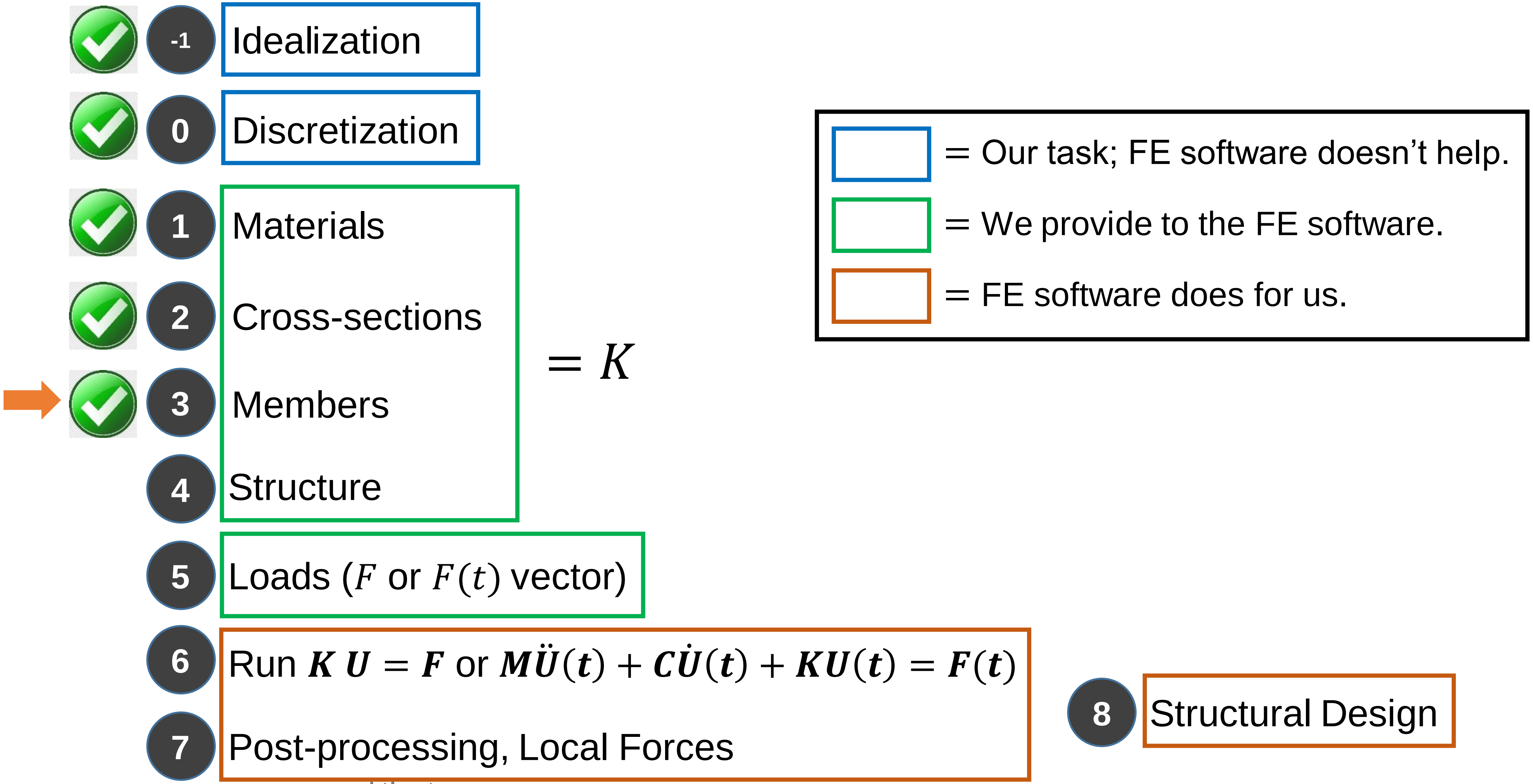
[2] For columns with axial compression falling between the limits provided, flexural stiffness shall be determined by linear interpolation.

[3] Tabulated values are appropriate where members are modeled using line elements to represent their properties.

[4] Diaphragms shall be permitted to be modeled as rigid in-plane if this does not result in differences in analysis outcomes.

[5] Specified stiffness values for mat foundations pertain for the general condition of the mat. Where the wall or other vertical members imposed sufficiently large forces, including local force reversals across stacked wall openings, the stiffness values may need to be reduced.

Finite Element Modeling, Analysis and Design Process



- ✓ -1 Idealization
- ✓ 0 Discretization
- ✓ 1 Materials
- ✓ 2 Cross-sections
- ➡ ✓ 3 Members
- 4 Structure

$= K$

- 5 Loads (F or $F(t)$ vector)
- 6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\boldsymbol{\ddot{U}}(\boldsymbol{t}) + \boldsymbol{C}\boldsymbol{\dot{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$
- 7 Post-processing, Local Forces
- 8 Structural Design

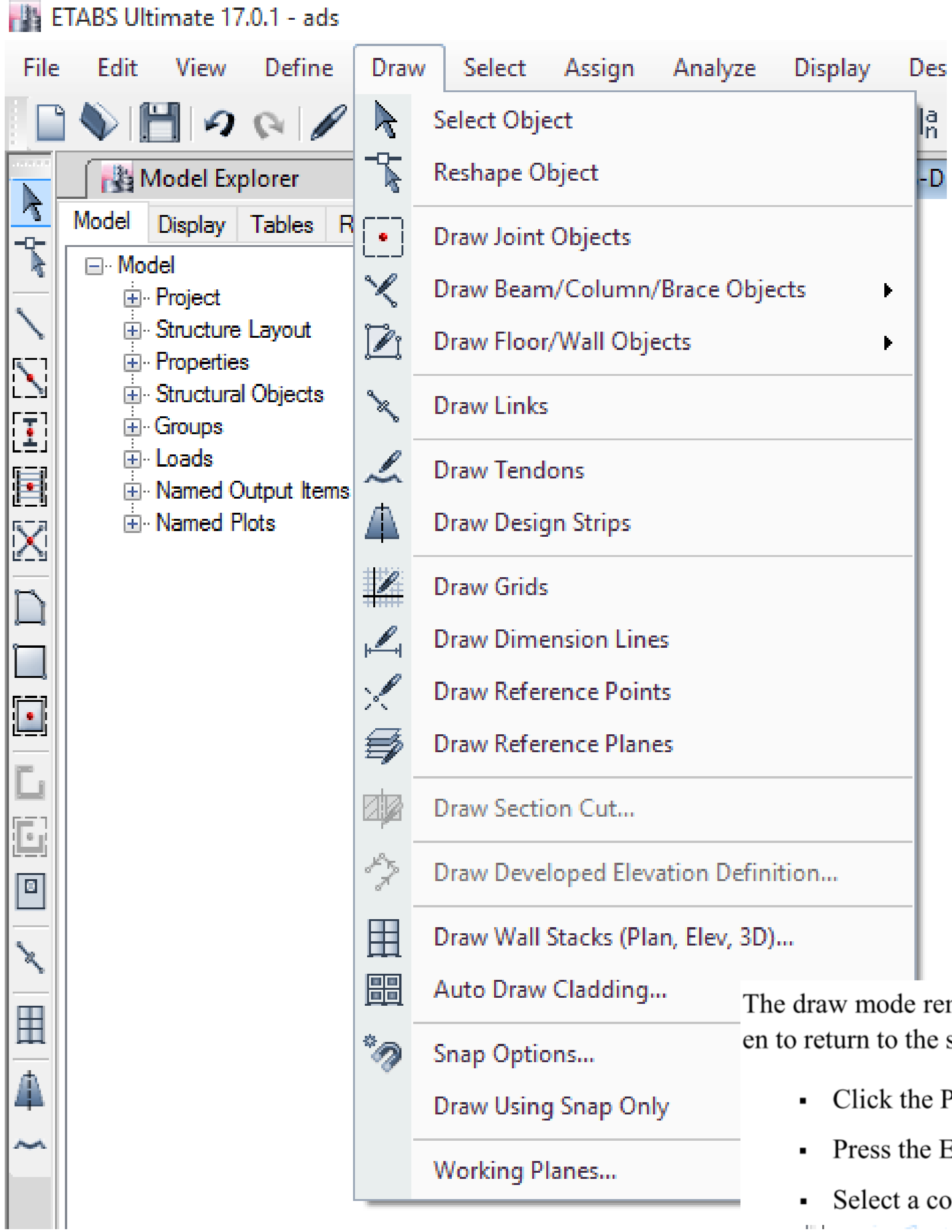
🟦 = Our task; FE software doesn’t help.

🟩 = We provide to the FE software.

🟧 = FE software does for us.

"Draw" Members

- Draw Joint Objects
- Draw Beam/Column/Brace Objects
    - Draw Beam/Column/Brace (Plan, Elev, 3D)
    - Quick Draw Beams/Columns (Plan, Elev, 3D)
    - Quick Draw Columns (Plan)
    - Quick Draw Secondary Beams (Plan)
    - Quick Draw Braces (Elev)
- Draw Floor/Wall Objects
    - Draw Floor/Wall (Plan, Elev, 3D)
    - Draw Rectangular Floor/Wall (Plan, Elev)
    - Quick Draw Floor/Wall (Plan, Elev)
    - Draw Walls (Plan)
    - Quick Draw Walls (Plan)
    - Draw Wall Openings (Plan, Elev, 3D)
- Draw Links

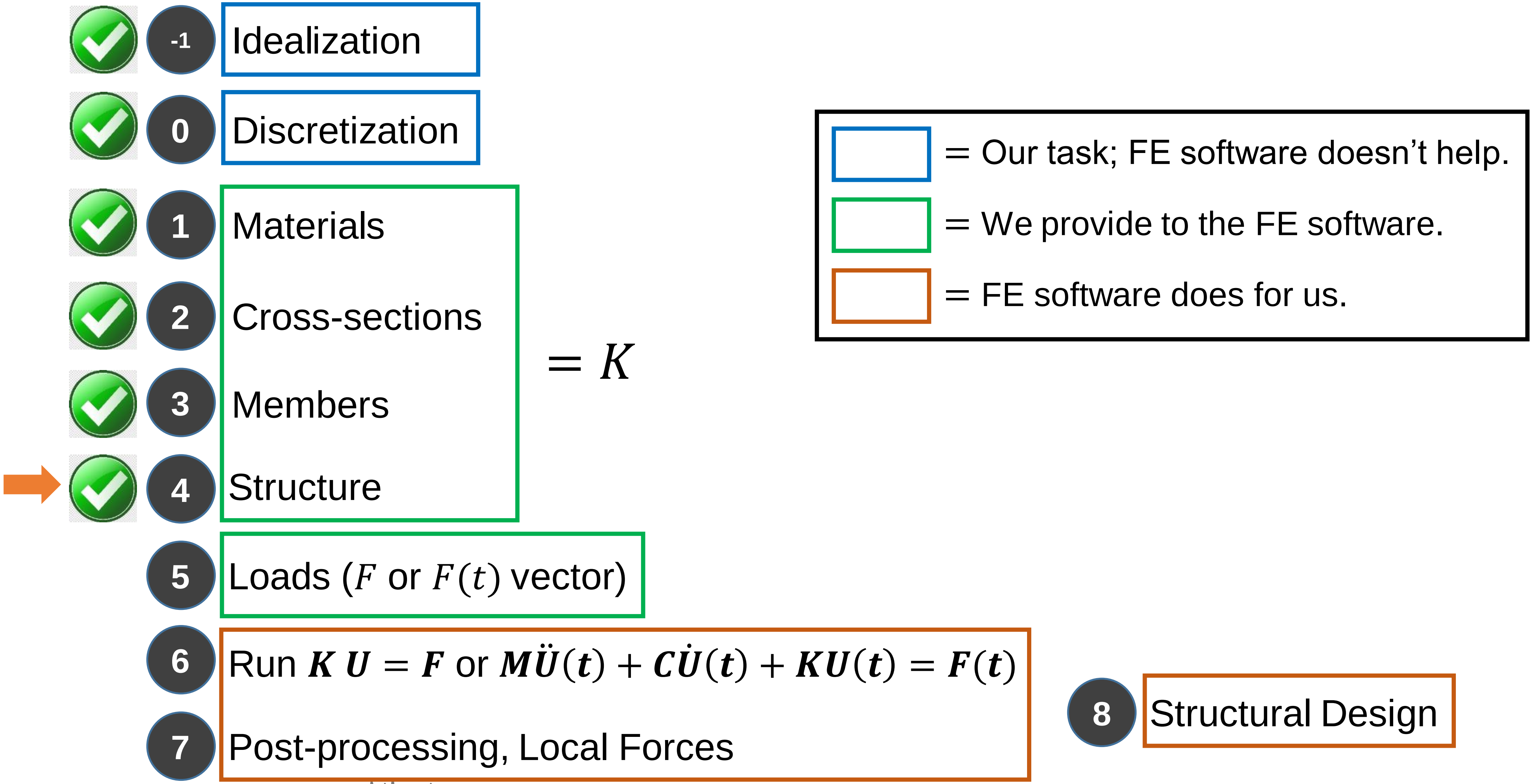


- Draw Tendons
- Draw Design Strips
- Draw Grids
- Draw Dimension Lines
- Draw Reference Points
- Draw Reference Planes
- Draw Section Cut
- Draw Developed Elevation Definition
- Draw Wall Stacks (Plan, Elev, 3D)
- Auto Draw Cladding
- Snap Options

The draw mode remains enabled until one of the following actions is taken to return to the select mode:

- Click the Pointer button on the toolbar .
- Press the Esc key on the keyboard.
- Select a command from the Select menu.

Finite Element Modeling, Analysis and Design Process



✓ -1 Idealization

✓ 0 Discretization

✓ 1 Materials

✓ 2 Cross-sections

✓ 3 Members

✓ 4 Structure

$= K$

5 Loads (F or $F(t)$ vector)

6 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$

7 Post-processing, Local Forces

8 Structural Design

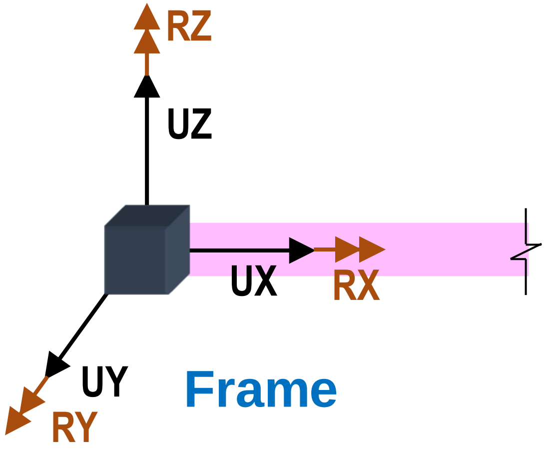
🟦 = Our task; FE software doesn't help.

🟩 = We provide to the FE software.

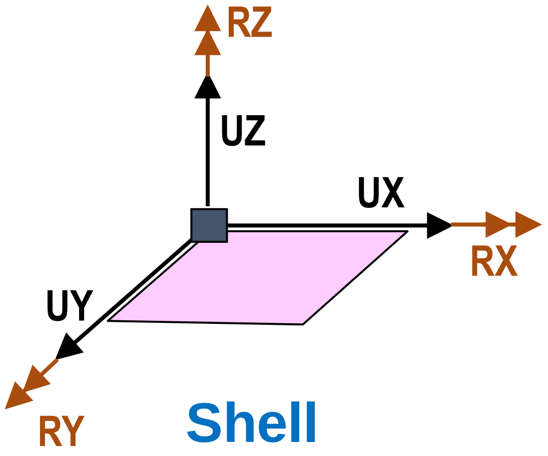
🟧 = FE software does for us.

Connecting Different Types of Elements

RZ = Drilling DOF (Let's assume that it is available in membrane and shell)



Frame



Shell

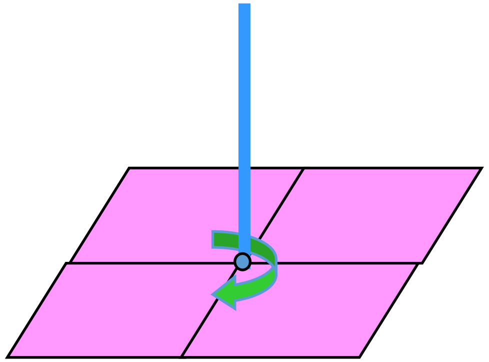
	Available DOFs/node	Truss	Frame	Membrane	Plate	Shell	Solid
Truss	UX, UY, UZ	OK	OK	UZ	OK	OK	OK
Frame	UX, UY, UZ, RX, RY, RZ	RX, RY, RZ	OK	RX, RY, UZ	UX, UY, RZ	OK	RX, RY, RZ
Membrane	UX, UY, RZ	RZ	OK	OK	UX, UY, RZ	OK	RZ
Plate	UZ, RX, RY	RX, RY	OK	UZ, RX, RY	OK	OK	RX, RY
Shell	UX, UY, UZ, RX, RY, RZ	RX, RY, RZ	OK	UZ, RX, RY	UX, UY, RZ	OK	RX, RY, RZ
Solid	UX, UY, UZ	OK	OK	UZ	UX, UY	OK	OK

Orphan Degrees of Freedom

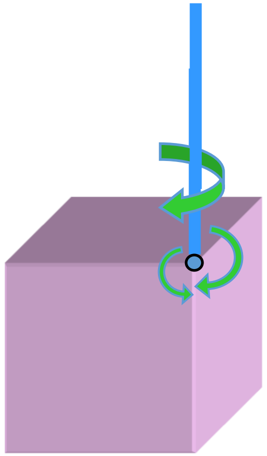
0 1 2 3 4

Connecting Dissimilar Elements

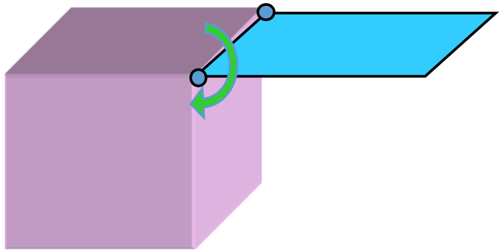
- When elements with different degree of freedom at ends connect with each other, special measures may need to be taken to provide proper connectivity **depending on the Software Capability**.



Beams to Plates

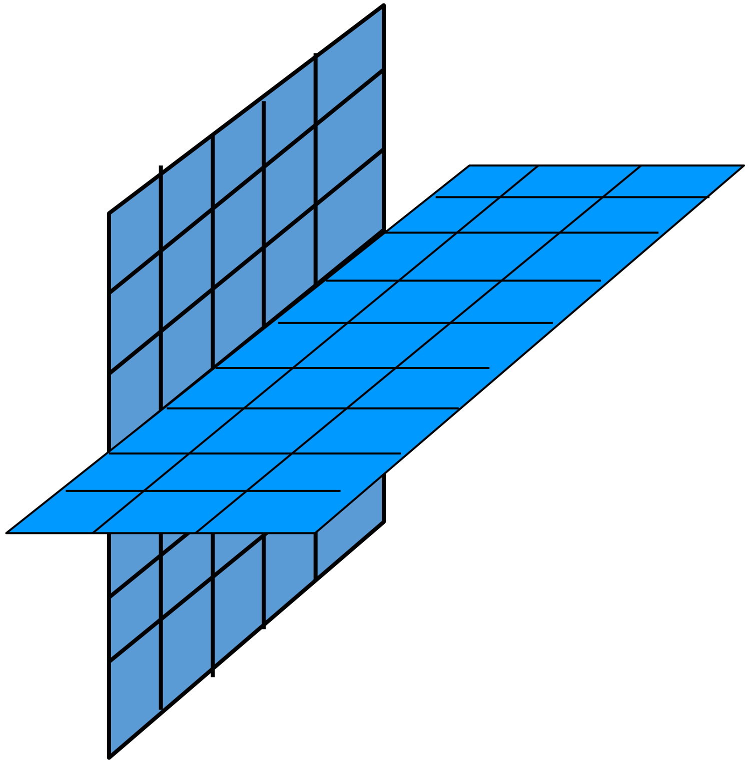


Beam to Brick

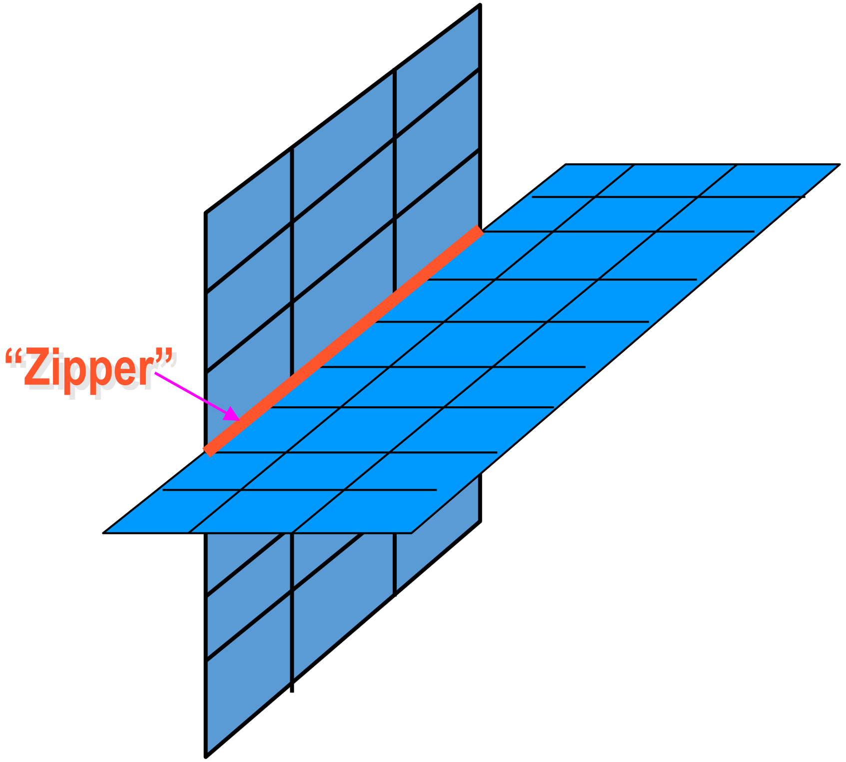


Plates to Brick

Connecting Walls to Slab

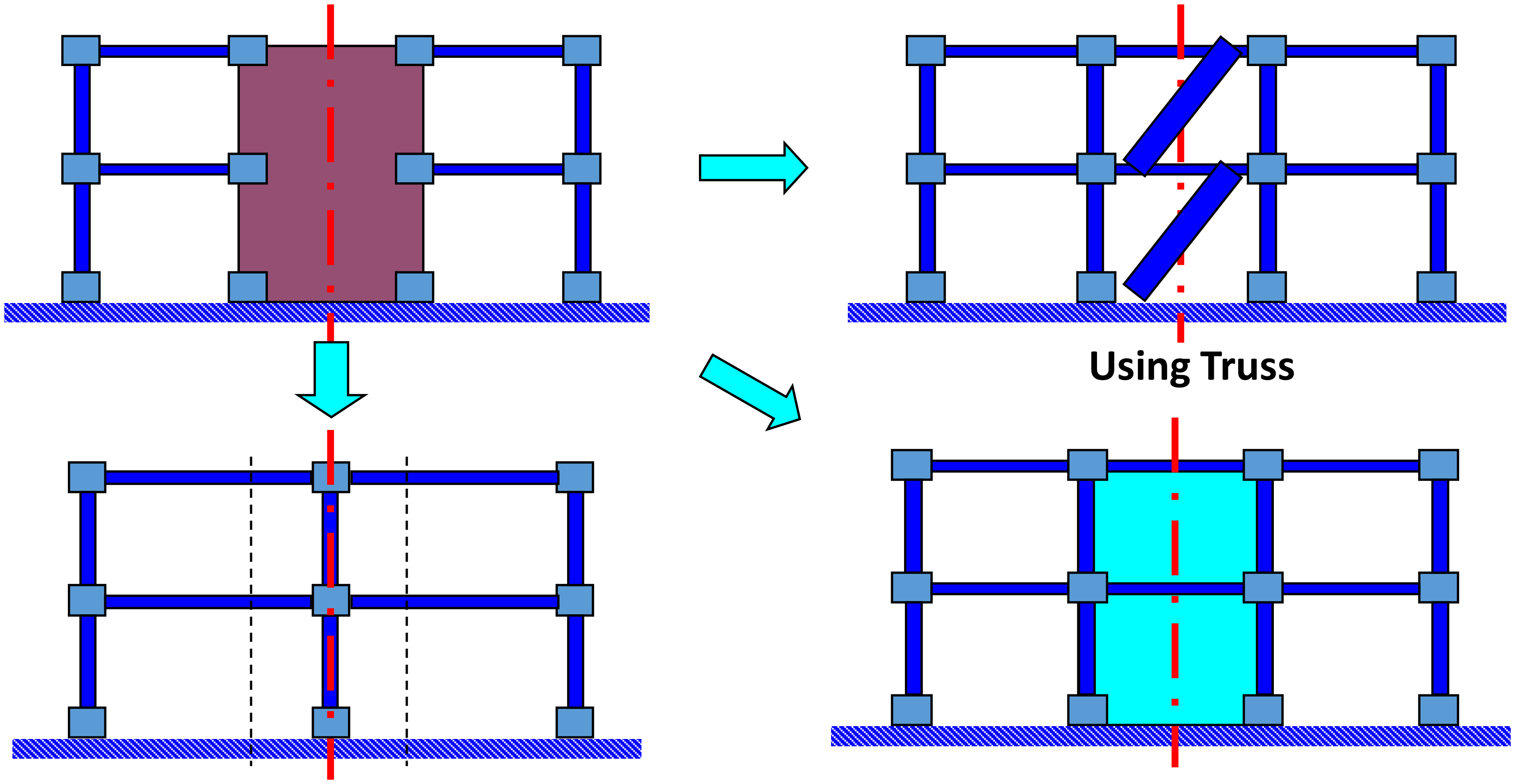


In general the mesh in the slab should match with mesh in the wall to establish connection.



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

Modeling of Planer Walls

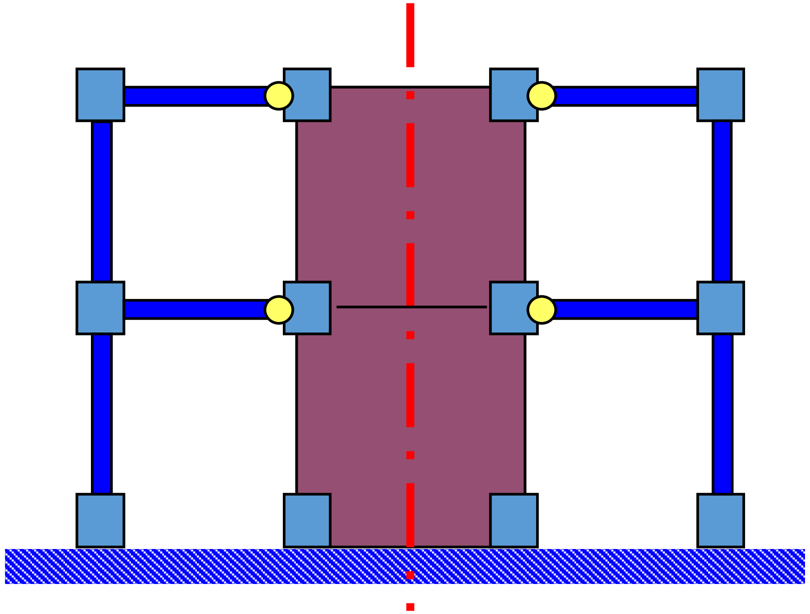


Using Truss

Using Beam and Column

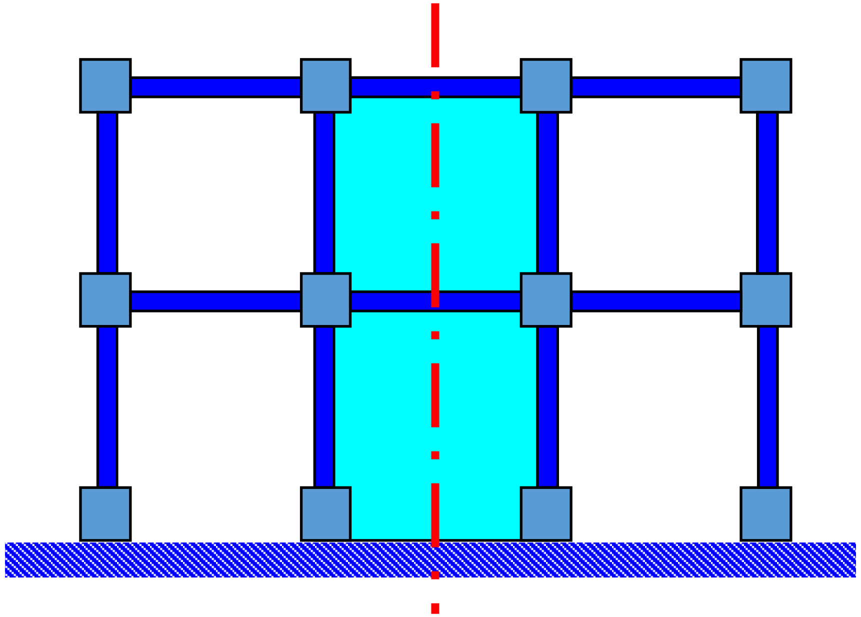
Using 2D Elements and Beams

Modeling Shear Walls Using 2D Elements



Modeling Shear-Walls using 2D Elements only

(No Moment continuity
with Beams and Columns unless
6 DOF Shell is used)

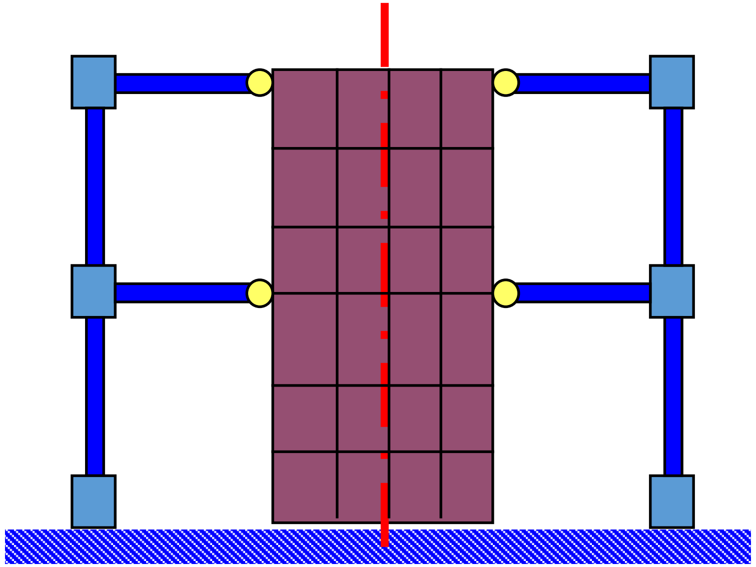


Modeling Shear-Walls using 2D Elements, Beams, Columns

(Full Moment continuity
with Beams and Columns is restored
by using additional beams)

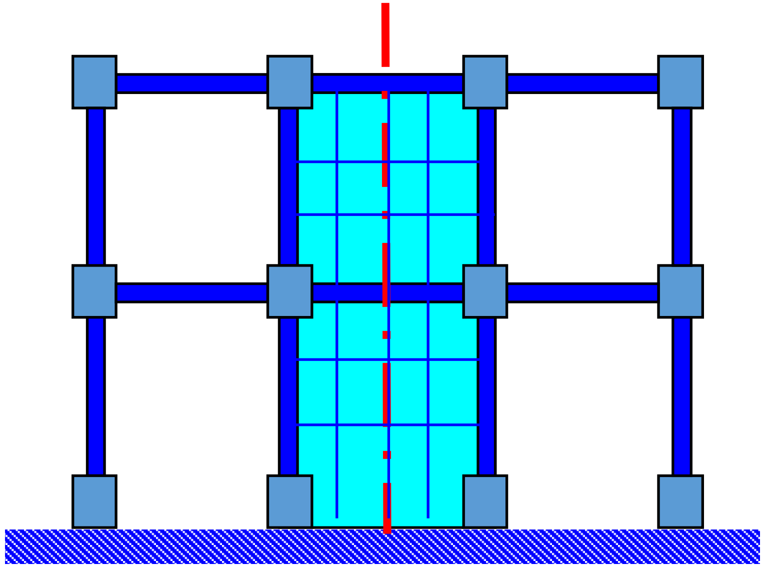
Using Plates to Model Shear Walls

- Multiple elements results in greater accuracy in determination of stress distribution and allow easy modeling of openings.



Using 2D Elements only

(No Moment continuity
with Beams and Columns unless 6
DOF Shell is used)

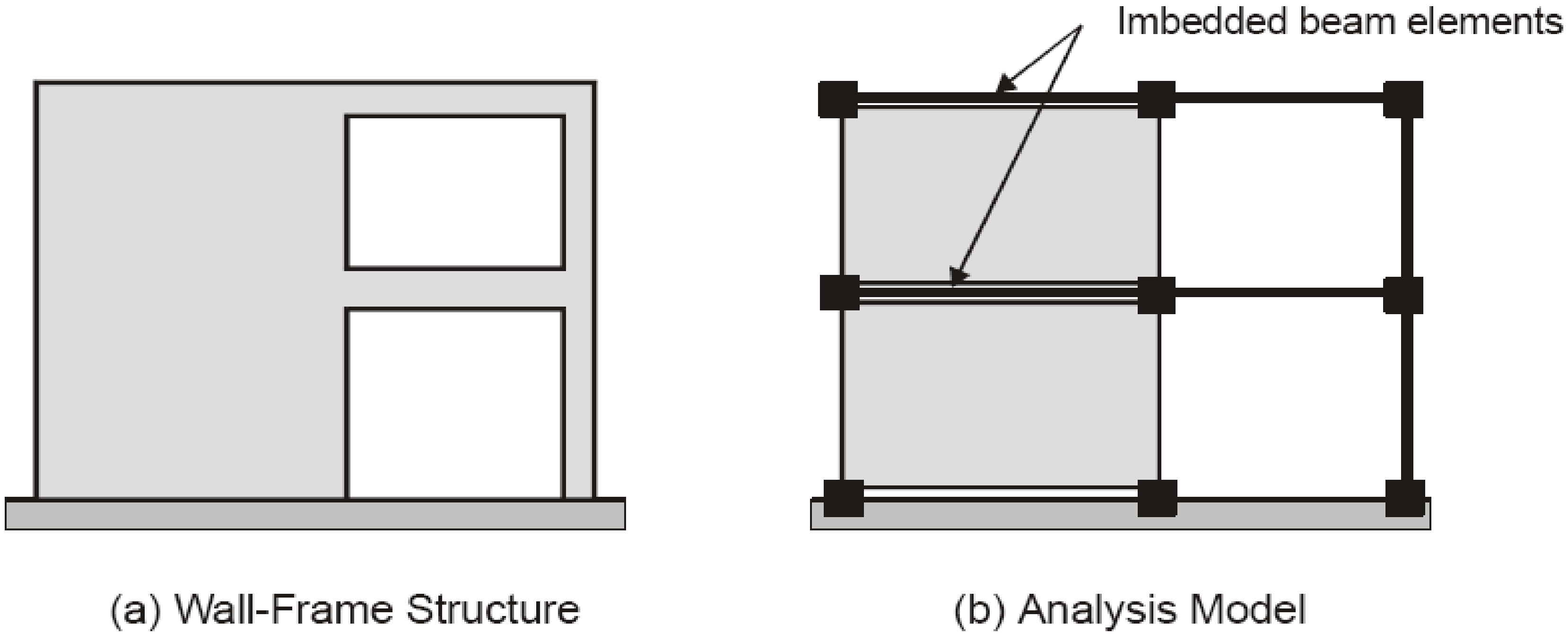


**Using 2D Elements with
Beams, Columns**

(Full Moment continuity
with Beams and Columns)

Connecting a Beam to a Shear Wall

- If a beam element is connected to a shear wall, a beam element must be imbedded in the wall



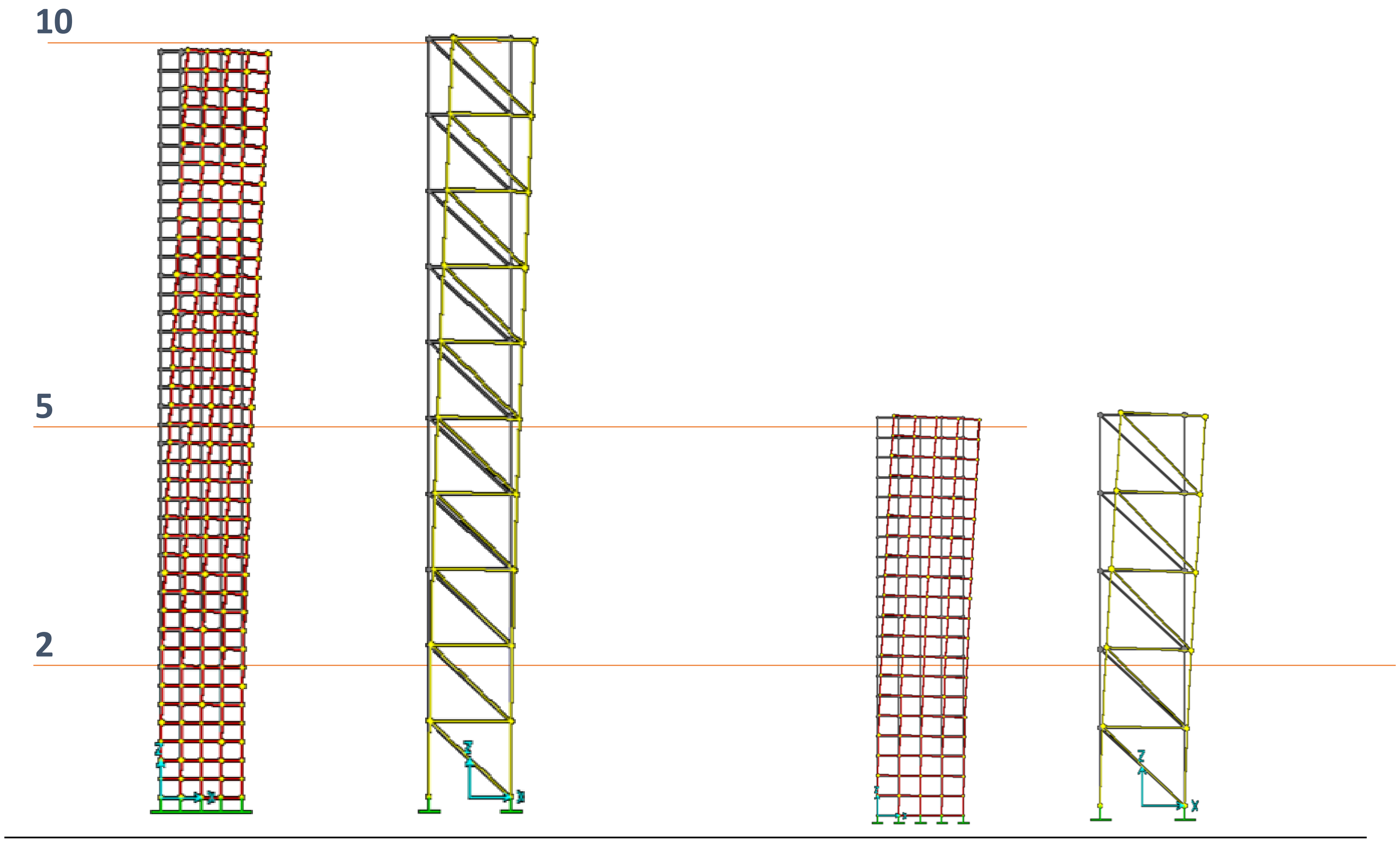
(a) Wall-Frame Structure (b) Analysis Model

- Coupling Beams are modelled either using frame elements or shell elements

Modeling of Shear Walls Using Truss Models

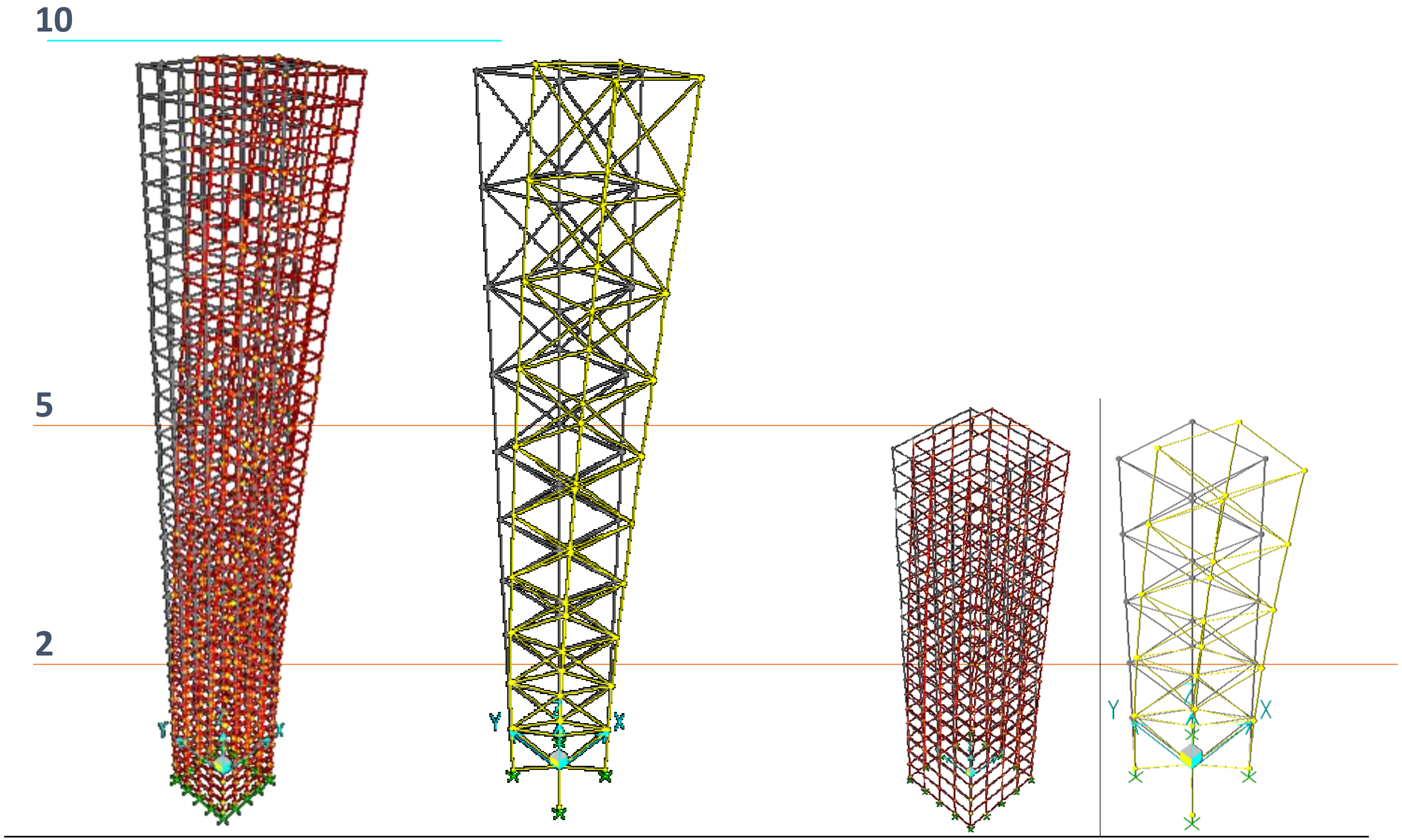
- The behavior of shear walls can be closely approximated by **truss models**:
  - The vertical elements provide the axial-flexural resistance
  - The diagonal elements provide the shear resistance
- Truss models are derived from the "**strut-and-tie**" concepts (extensively used for **deep beams** and **shear walls**).
- This model represents the "cracked" state of the wall where all tension is taken by ties and compression by concrete.
- Difficult to determine the size and reinforcement in diagonal elements.
- Nonlinear axial load-deformation hinges can be used. Hinges in diagonal struts should be force-controlled to **detect shear failure** or the diagonals may be forced to remain elastic.

Truss Model for Shear Walls



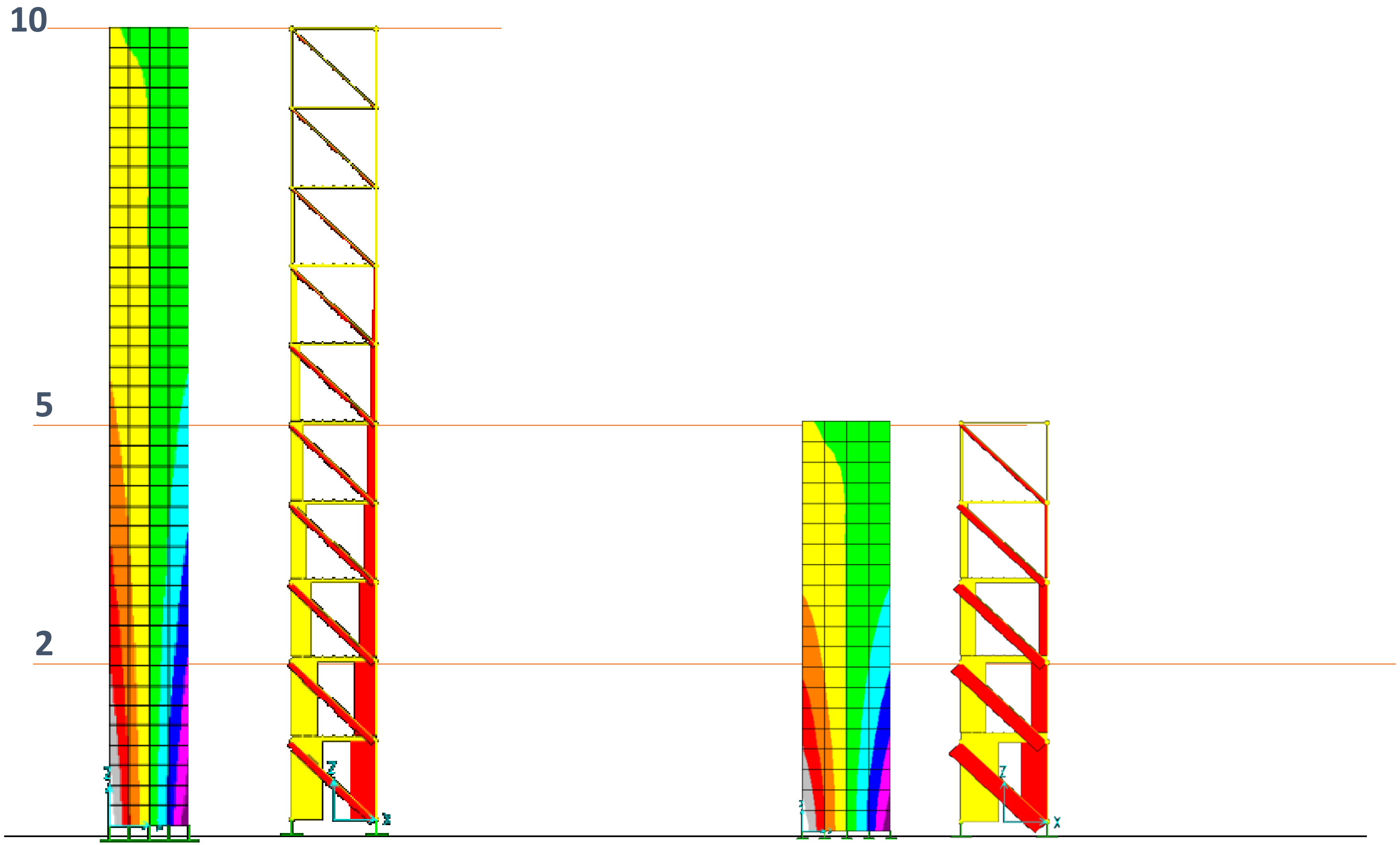
Comparing Deformation and Deflections of Shell Model with Truss Model

Truss Model for Shear Walls



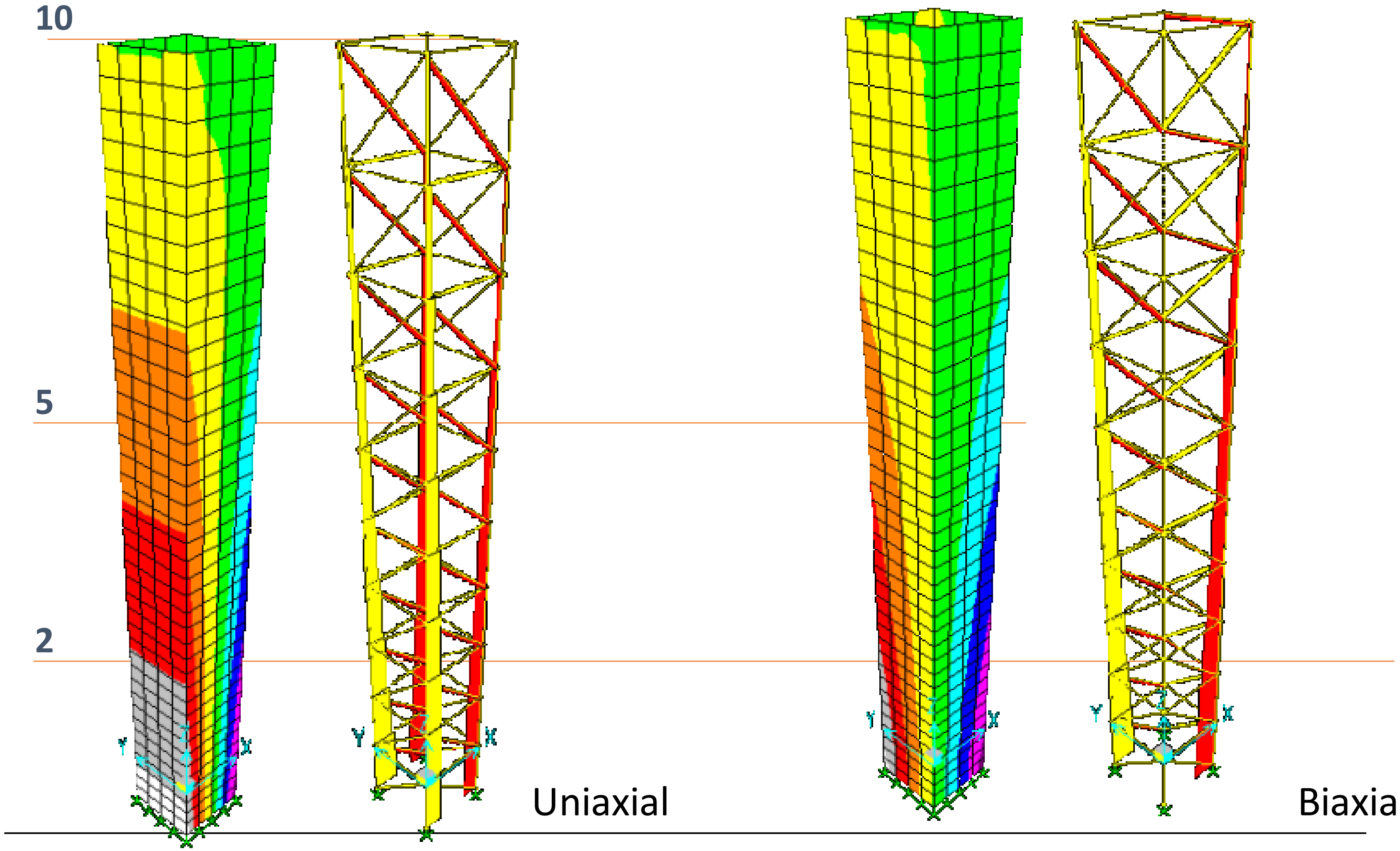
Comparing Deformation and Deflections of Shell Model with Truss Model

Truss Models for Shear Walls

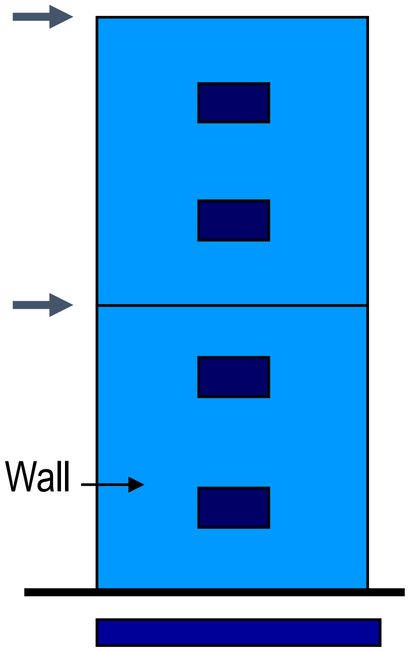


Comparing Axial Stress and Axial Force Patterns

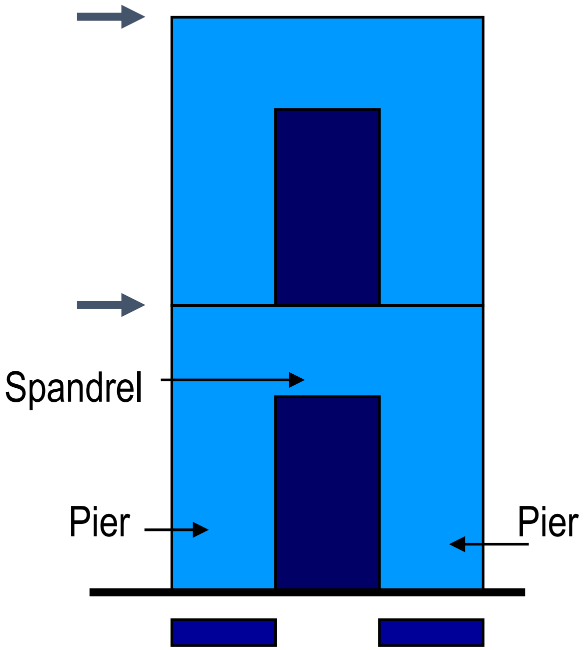
Truss Models for Shear Walls



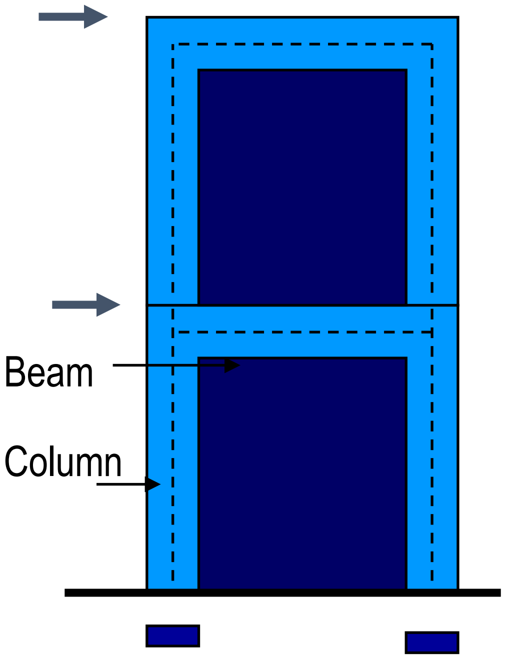
Modeling of Openings in Shear Walls



Very Small Openings may not alter wall behaviour

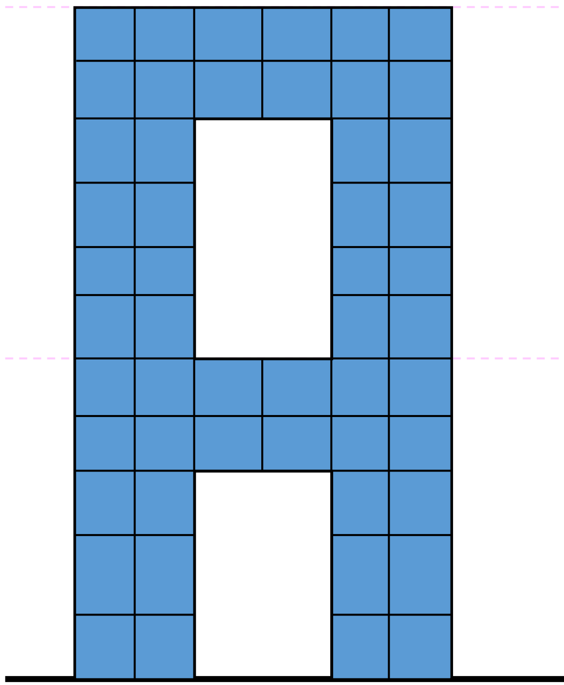


Medium Openings may convert shear wall to Pier and Spandrel System

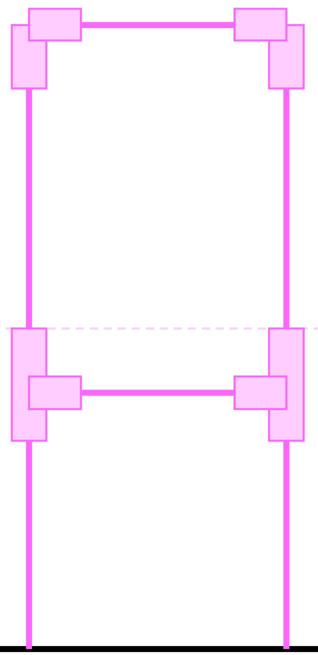


Very Large Openings may convert the Wall to Frame

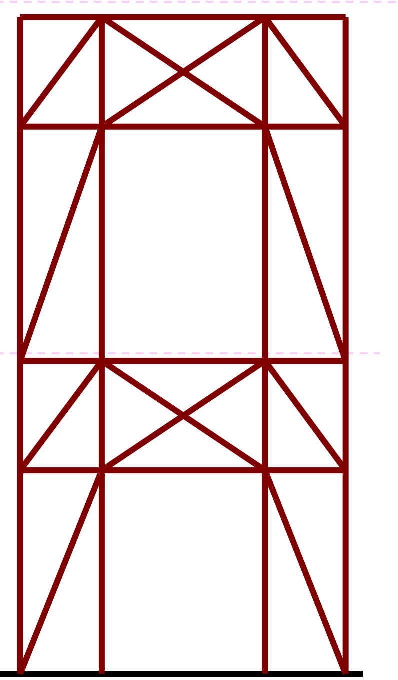
Modeling Walls with Opening



Shell Model



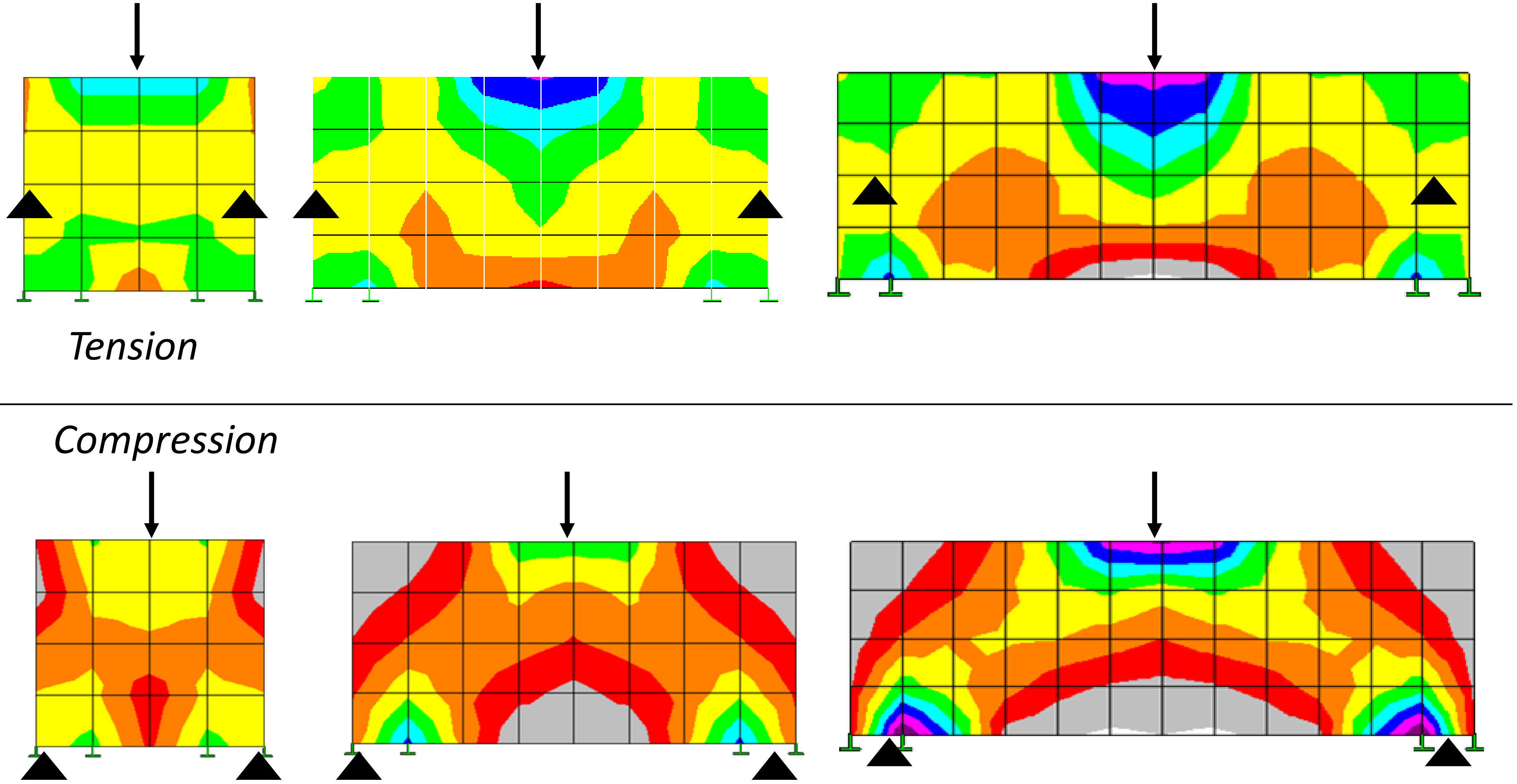
Rigid Frame Model

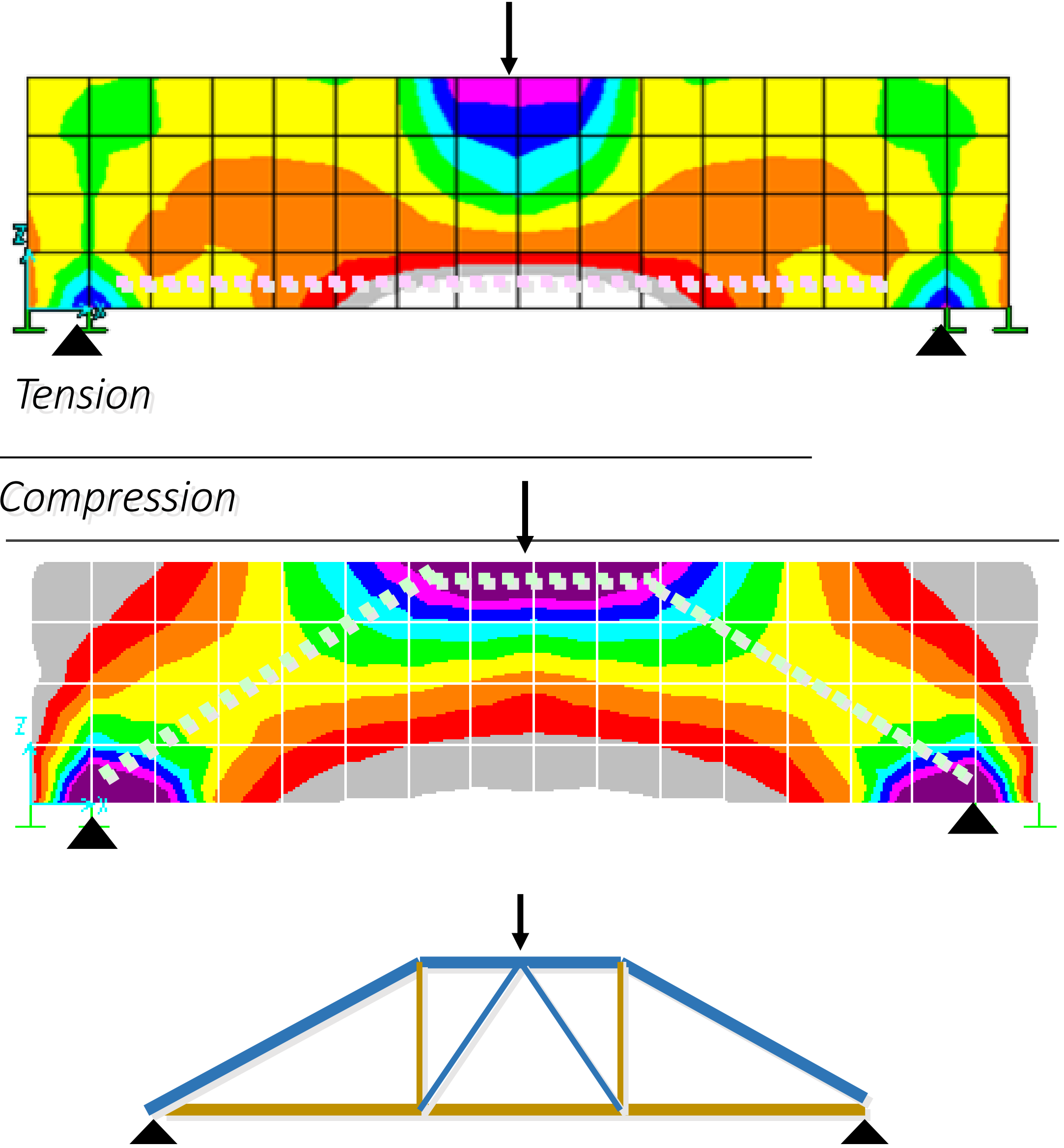


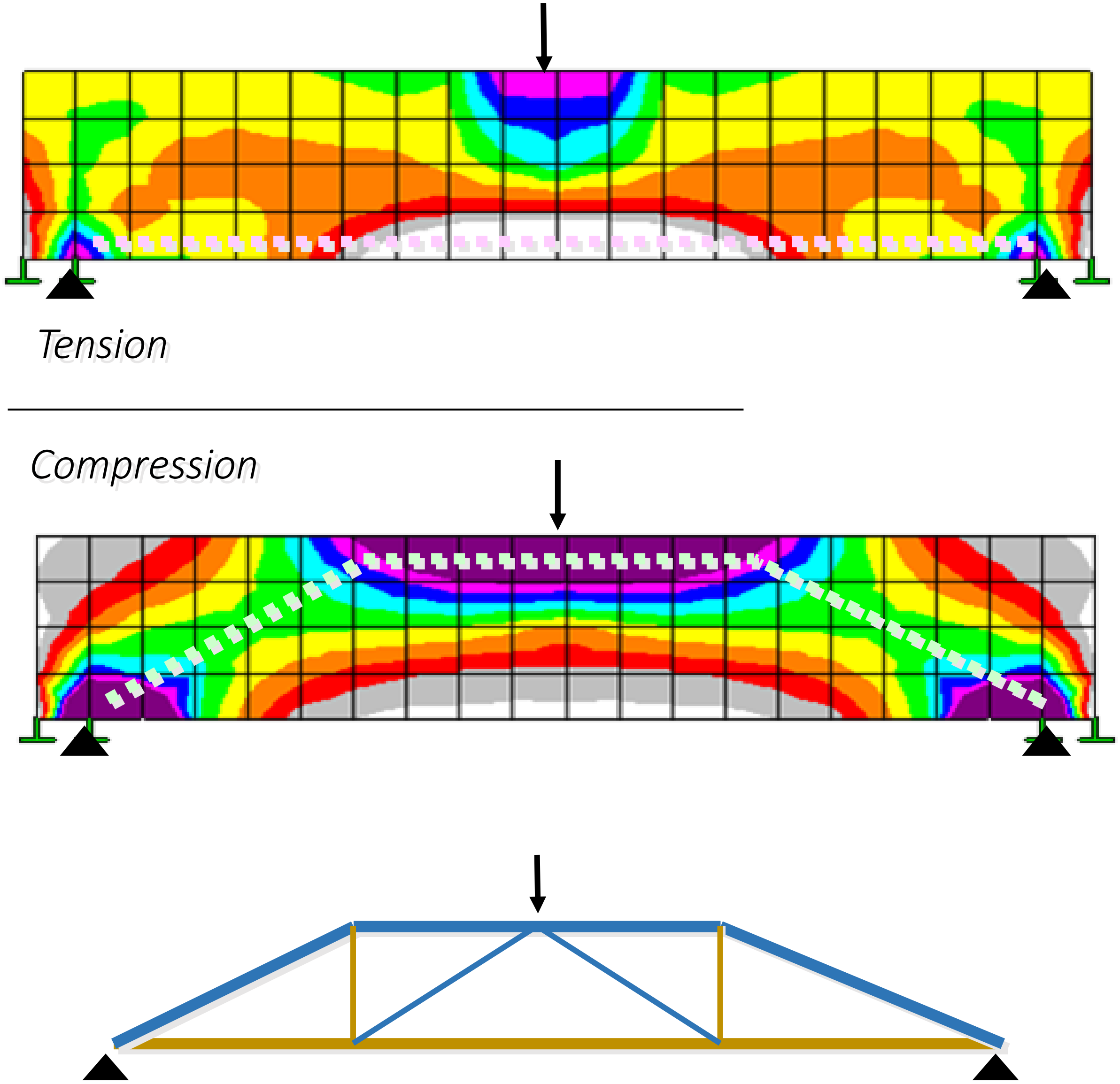
Truss Model

Special Modeling Techniques for Transfer Girders and Deep Beams

The Axial Stresses



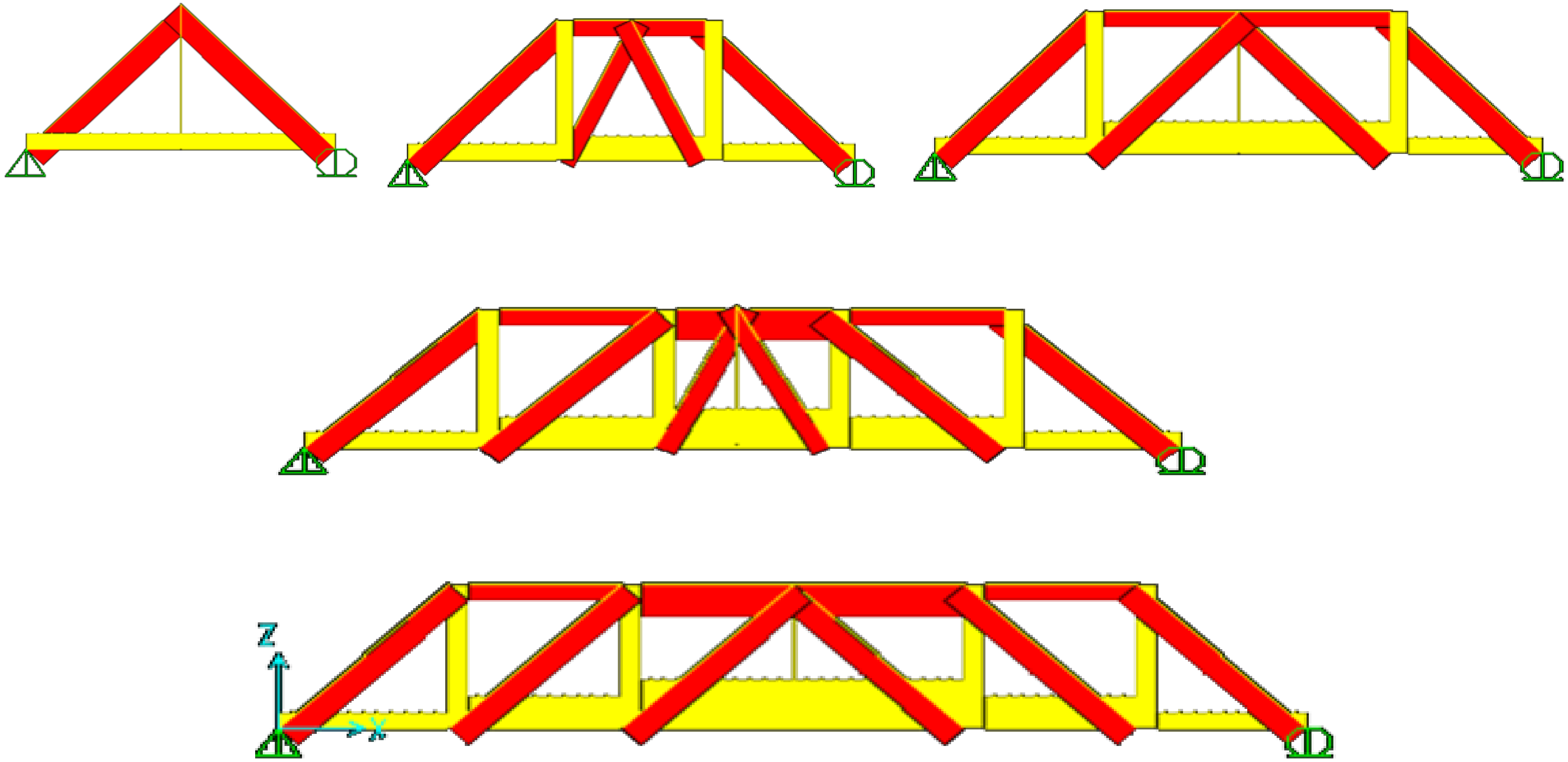




The Hidden Truss in Members

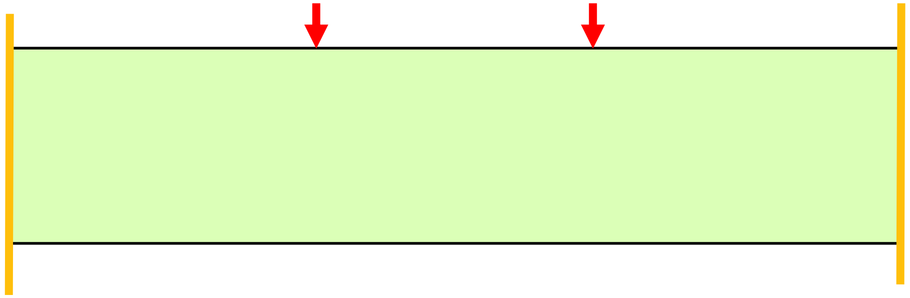
Truss Models for Deep Beams

The Axial Forces in Truss Elements

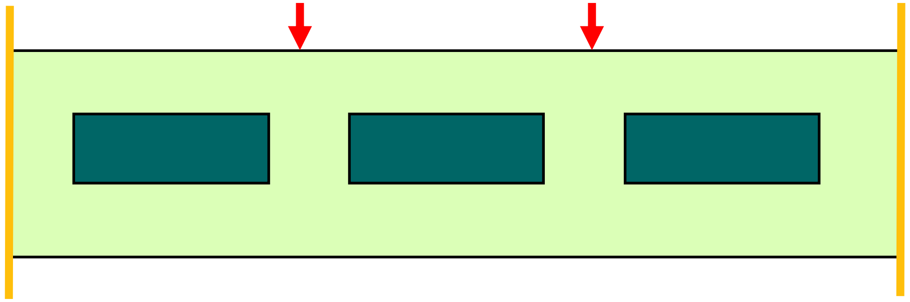


Deep Beam or Veirendel Girders

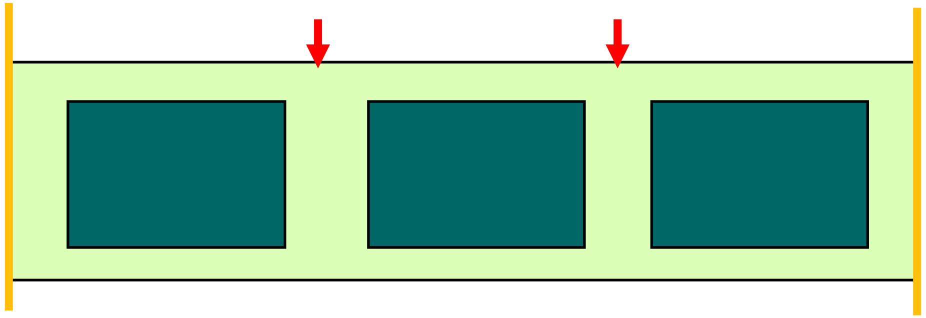
- Deep Beam



- Deep Beam or Veirendel Girder



- Veirendel Girder



Modeling Openings in Beams

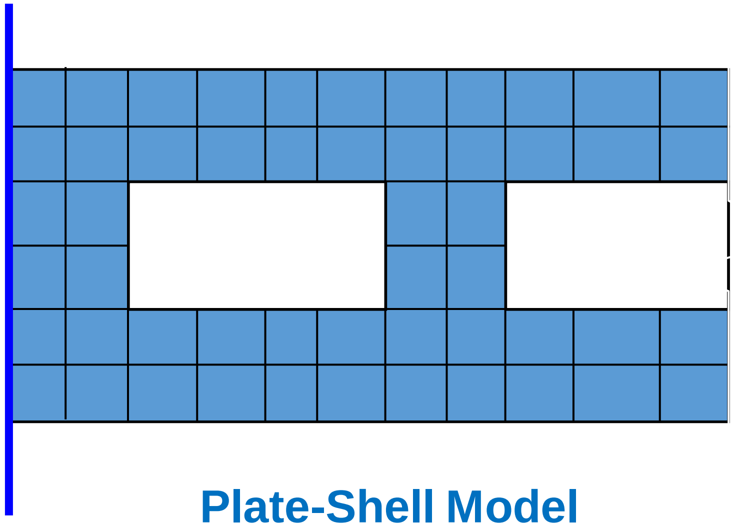
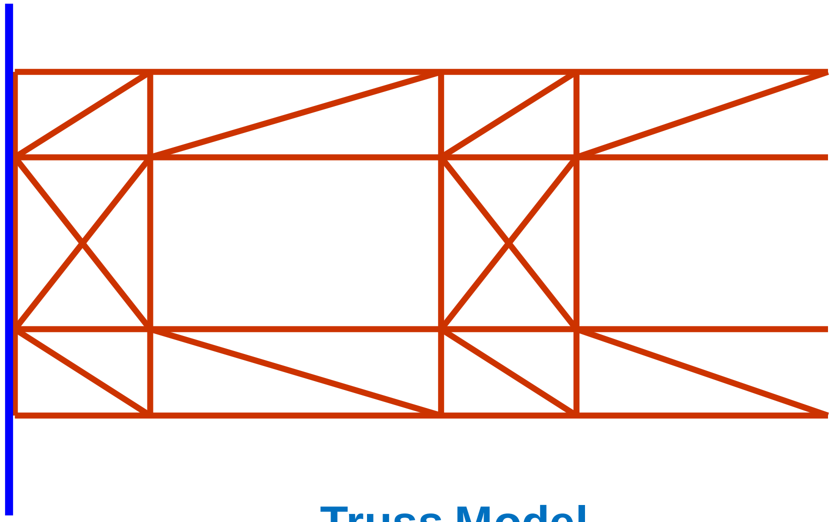


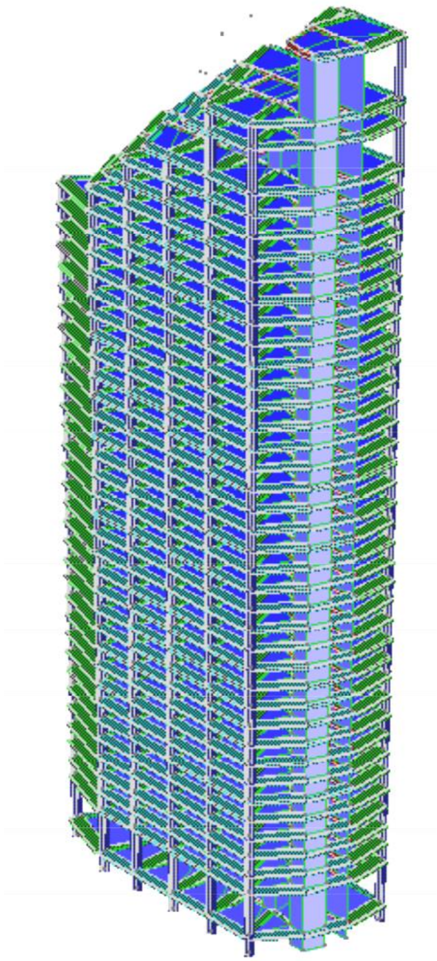
Plate-Shell Model



Truss Model

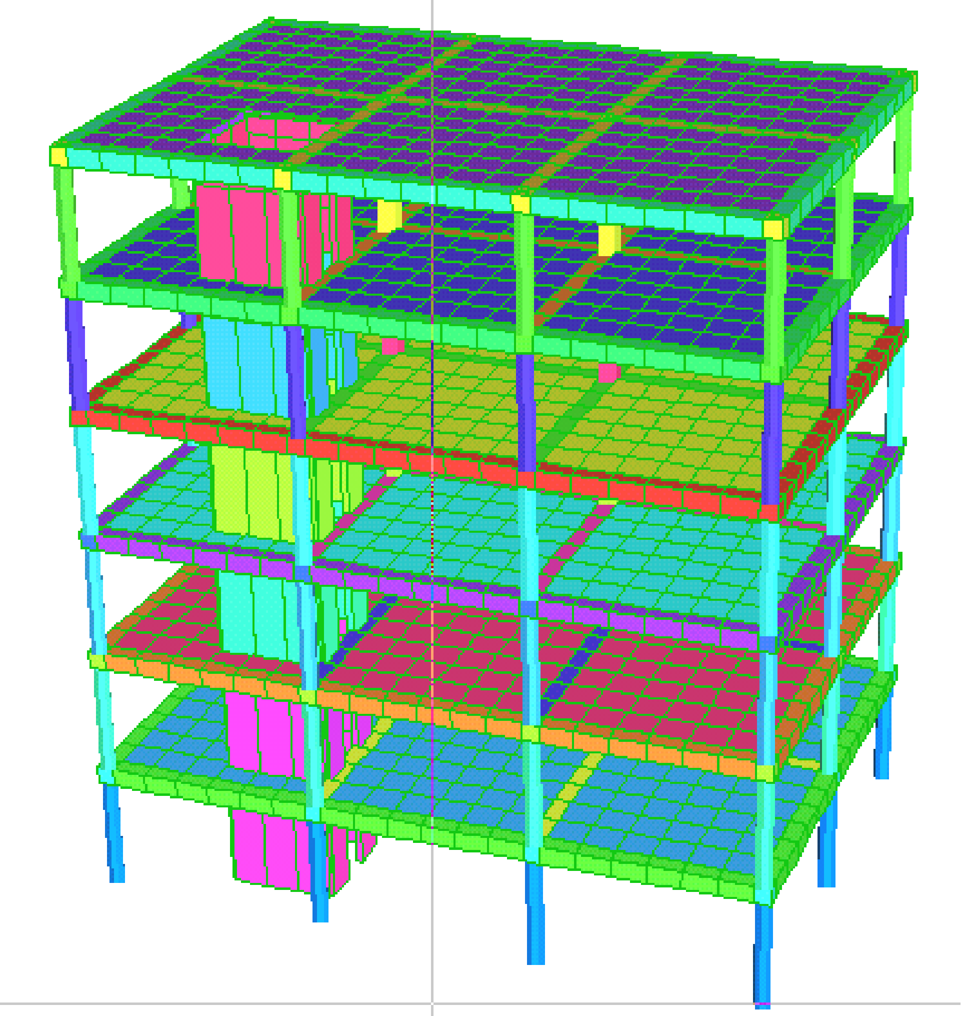
Full 3D Finite Element Model

- The columns and beams are modeled by using frame elements.
- The slabs and shear walls are modeled by using shell elements.
    - At least 9 or 16 elements in each slab panel must be used if gravity loads are applied to the slabs.
    - If the model is only for lateral analysis, one element per slab panel may be sufficient to model the in-plane stiffness.
    - Shear walls may be modeled by membrane or shell or plane stress element. The out of plane bending is not significant.



Full 3D Finite Element Model

- Example:
  - Uses more than 4000 beam and shell elements.
  - Suitable for analysis for gravity and lateral loads.
  - Results can be used for design of columns and beams.
  - Slab reinforcement difficult to determine from shell results.



Finite Element Modeling, Analysis and Design Process

- ✓ -1 🟦 Idealization
- ✓ 0 🟦 Discretization
- ✓ 1 🟩 Materials
- ✓ 2 🟩 Cross-sections
- ✓ 3 🟩 Members
- ✓ 4 🟩 Structure

$= K$

- ➡ ✓ 5 🟩 Loads (F or $F(t)$ vector)
- 6 🟧 Run $\boldsymbol{K}\ \boldsymbol{U} = \boldsymbol{F}$ or $\boldsymbol{M}\ddot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{C}\dot{\boldsymbol{U}}(\boldsymbol{t}) + \boldsymbol{K}\boldsymbol{U}(\boldsymbol{t}) = \boldsymbol{F}(\boldsymbol{t})$
- 7 🟧 Post-processing, Local Forces
- 8 🟧 Structural Design

🟦 = Our task; FE software doesn’t help.

🟩 = We provide to the FE software.

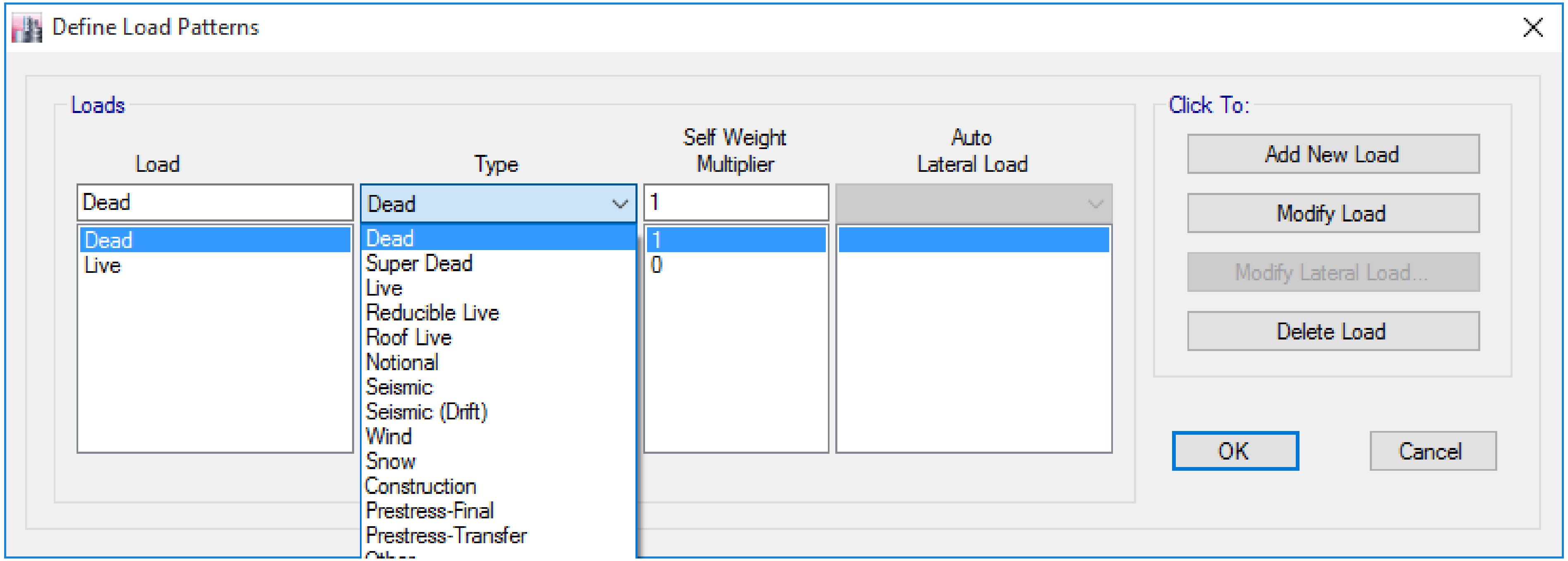
🟧 = FE software does for us.

Load Patterns, Load Cases and Load Combinations

Load Patterns, Load Cases and Load Combinations

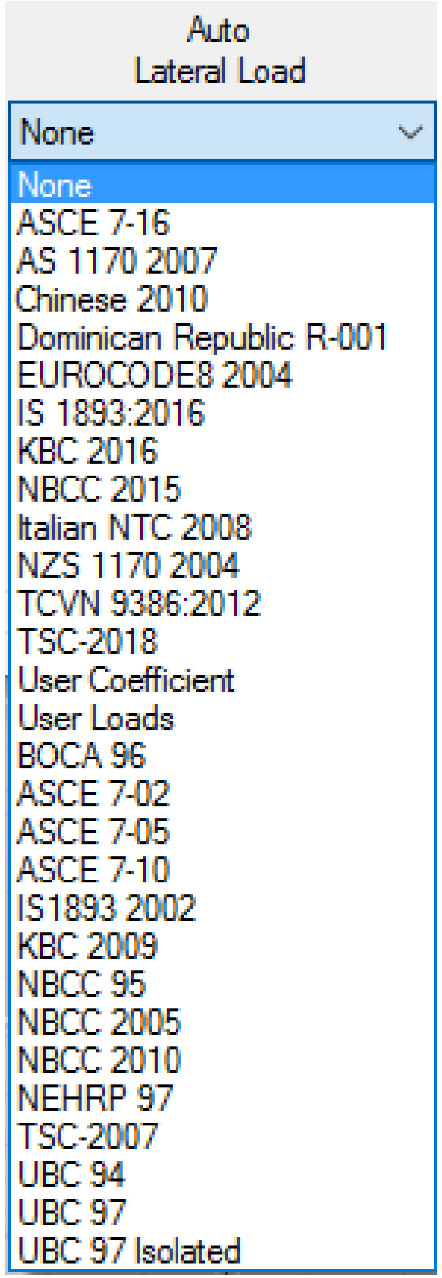
- A **Load Pattern** is a specified spatial distribution of forces, displacements, temperatures, and other effects that act upon the structure.
- A Load Pattern by itself does not cause any response of the structure. Load Patterns must be applied in Load Cases in order to produce results.
- A **Load Case** defines how the Load Patterns are to be applied (e.g., statically or dynamically), how the structure responds (e.g., linearly or nonlinearly), and how the analysis is to be performed (e.g., modally or by direct-integration.)
- A Load Case may apply a single Load Pattern or a combination of Load Patterns.
- The results of Load Cases can be combined after analysis by defining **Load Combinations**. A Load Combination is a sum or envelope of the results from different Load Cases.
- For linear problems, algebraic-sum types of Load Combinations make sense. For nonlinear problems, it is usually best to combine Load Patterns in the Load Cases, and use Load Combinations only for computing envelopes.
- When performing **design**, only the results from **Load Combinations** are used. Load Combinations can be automatically created by the design algorithms, or you can create your own. If necessary, you can define Load Combinations that contain only a single Load Case.

Load Patterns



Load	Type	Self Weight Multiplier	Auto Lateral Load
Dead	Dead	1	
Live		0	

Type options: Dead, Super Dead, Live, Reducible Live, Roof Live, Notional, Seismic, Seismic (Drift), Wind, Snow, Construction, Prestress-Final, Prestress-Transfer



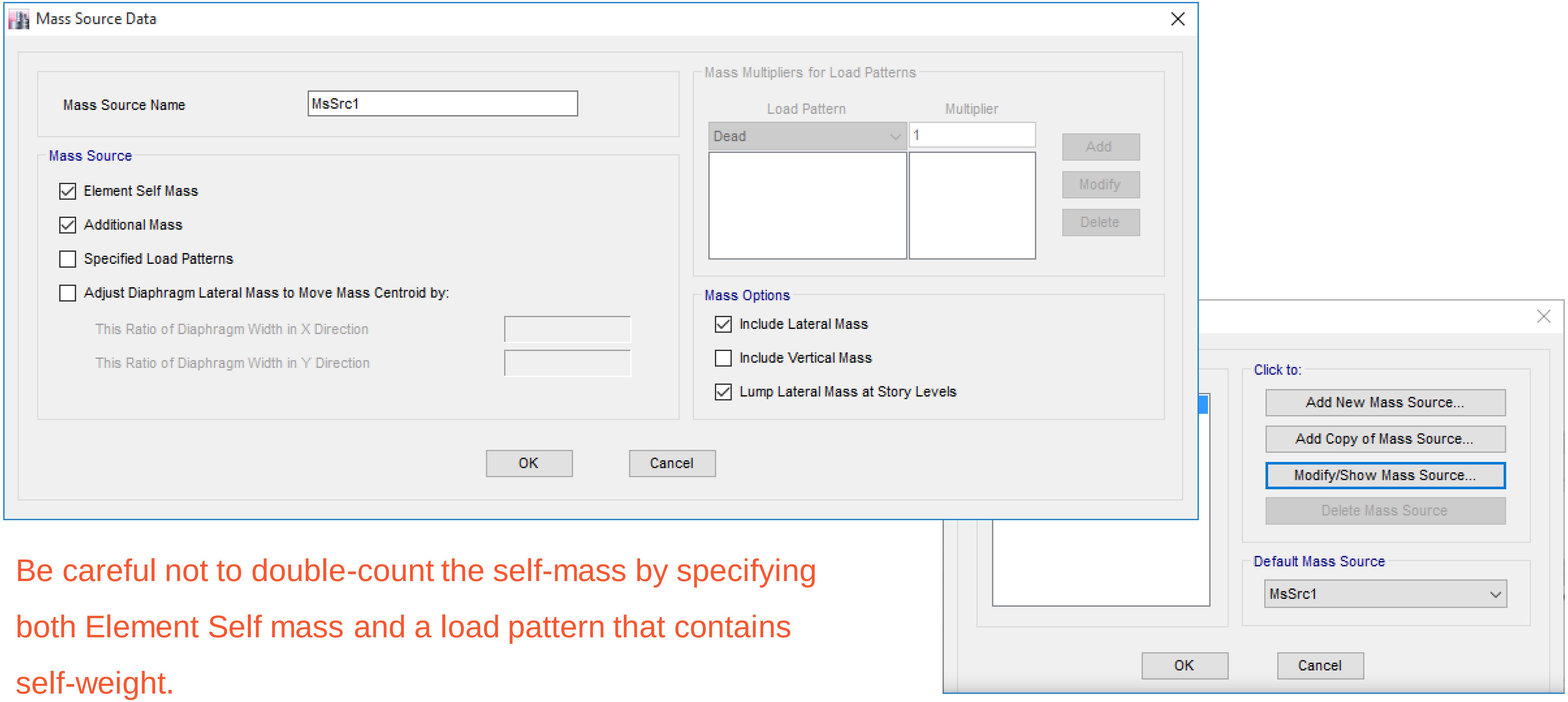
Auto Lateral Load: None

- None
- ASCE 7-16
- AS 1170 2007
- Chinese 2010
- Dominican Republic R-001
- EUROCODE8 2004
- IS 1893:2016
- KBC 2016
- NBCC 2015
- Italian NTC 2008
- NZS 1170 2004
- TCVN 9386:2012
- TSC-2018
- User Coefficient
- User Loads
- BOCA 96
- ASCE 7-02
- ASCE 7-05
- ASCE 7-10
- IS1893 2002
- KBC 2009
- NBCC 95
- NBCC 2005
- NBCC 2010
- NEHRP 97
- TSC-2007
- UBC 94
- UBC 97
- UBC 97 Isolated

Mass Source

- The mass used as inertia in dynamic analyses or to calculate certain types of loads can be controlled by specifying the Mass Source.
- Multiple Mass Sources can be defined so that different load cases can use a different mass distributions for loading and inertia. Examples could include e.g.
    - Modeling a structure supporting **different configurations of equipment**, or
    - Explicitly considering the effect of **different eccentricities of the story mass** on the mode shapes.
- There are three possible contributions to a Mass Source:
    - **Element Self Mass** — This includes the mass from the section properties used by the elements. For the Link/Support elements, this mass is explicitly defined in the section property. For all other elements the mass comes from the material property referenced by the section property.
    - **Additional Mass** — This includes mass as signed to the joints and any additional mass as signed to the frame or shell elements.
    - **Specified Load Patterns** — Mass is computed from the gravity load as computed from a specified linear combination of load patterns.

Mass Source



Mass Source Data

Mass Source Name: MsSrc1

Mass Source

- ☑ Element Self Mass
- ☑ Additional Mass
- ☐ Specified Load Patterns
- ☐ Adjust Diaphragm Lateral Mass to Move Mass Centroid by:
  - This Ratio of Diaphragm Width in X Direction
  - This Ratio of Diaphragm Width in Y Direction

Mass Multipliers for Load Patterns

Load Pattern	Multiplier
Dead	1

Mass Options

- ☑ Include Lateral Mass
- ☐ Include Vertical Mass
- ☑ Lump Lateral Mass at Story Levels

Default Mass Source: MsSrc1

Be careful not to double-count the self-mass by specifying both Element Self mass and a load pattern that contains self-weight.

Different Mass Sources in Different Cases

- The default mass source will be used for all Load Cases unless specified otherwise.
- A specified Mass Source can be selected for the following types of load cases:
  - Nonlinear static
  - Nonlinear staged-construction
  - Nonlinear direct-integration time-history
- Response-spectrum and modal time-history load cases use the Mass Source of their corresponding modal load case.

Different Mass Sources in Different Cases

- For example, consider the case where a response-spectrum analysis is to be carried out for a tower both with and without a significant equipment load. You could do the following:
  - Define two load patterns
    - **DEAD**, which includes the self-weight of the tower structure
    - **LIVE**, which includes only the weight of the equipment
  - Define two mass sources
    - **MASSDEAD** (only the load pattern **DEAD** with a scale factor of 1, no Element Self Mass or Additional Mass
    - **MASSDEADLIVE**, (both load patterns **DEAD** and **LIVE**, each with a scale of 1, no Element or Additional Mass)
  - Define two nonlinear static load cases
    - **DEAD**, which specifies mass source **MASSDEAD**
    - **DEADLIVE**, which specifies mass source **MASSDEADLIVE**
  - Define two modal load cases
    - **MODALDEAD**, which uses the stiffness of load case **DEAD**
    - **MODALDEADLIVE**, which uses the stiffness of load case **DEADLIVE**

Different Mass Sources in Different Cases

- Define two response-spectrum load cases
  - – One which uses the modes of load case **MODALDEAD**
  - – the other which uses the modes of load case **MODALDEADLIVE**

- Note that in the above example the nonlinear static load cases were used only to **specify the Mass Source**. However, in most practical cases you would also want to apply the corresponding load patterns as loads and consider P-delta effects, as these would likely also have an effect on the modes.
- Note that the response-spectrum cases, in addition to considering the inertial effect of the different masses on the modes, also **apply acceleration loads that are based on mass**. These loads will automatically be based on the mass from the same Mass Source used to calculate the modes.

Geometric Nonlinearity

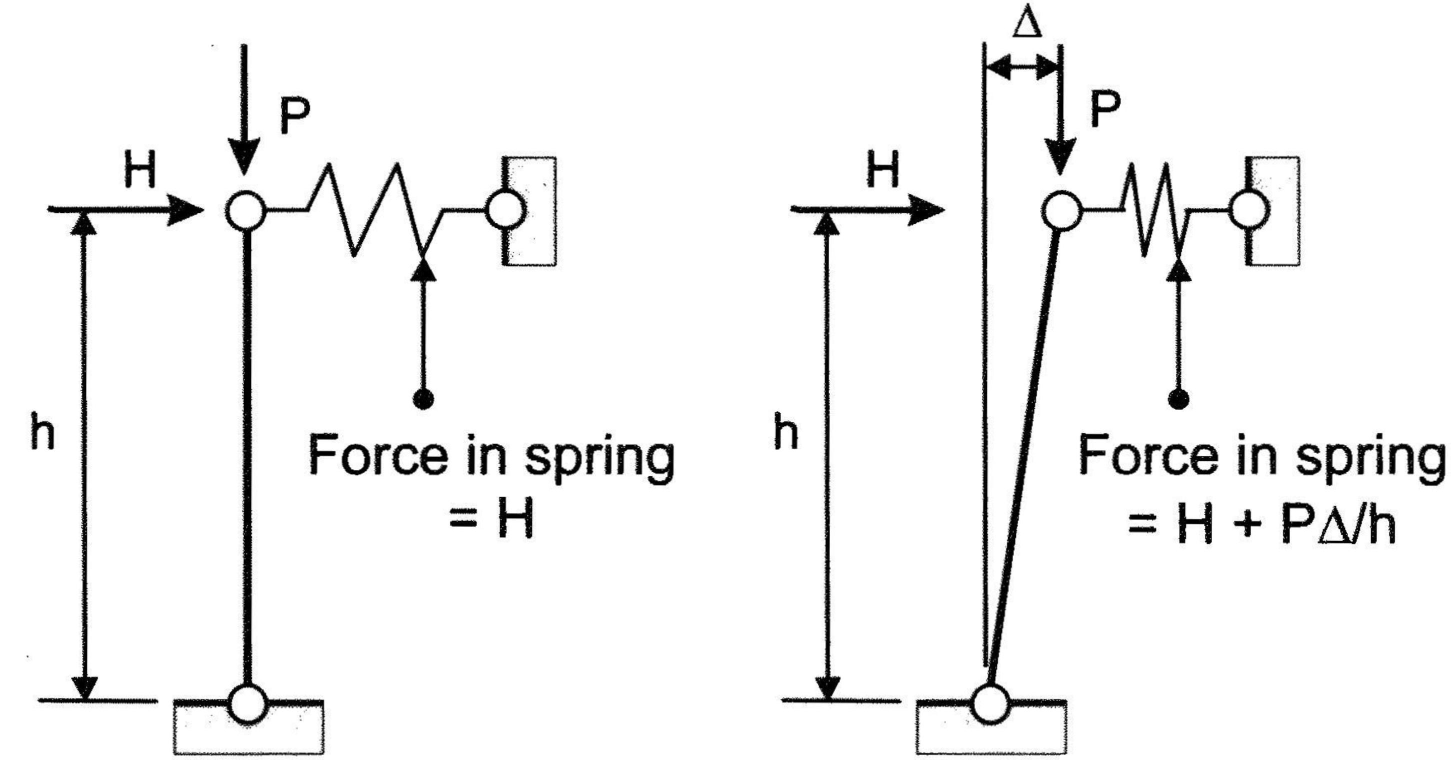
- If the load on the structure and/or the resulting deflections are large, then the load-deflection behavior may become nonlinear.
- Several causes of this **nonlinear behavior** can be identified:
  - **P-delta effect:** when large stresses (or forces and moments) are present within a structure, equilibrium equations written for the original and the deformed geometries may differ significantly, even if the deformations are very small.
  - **Large-displacement effect:** when a structure undergoes large deformation (in particular, large strains and rotations), the usual engineering stress and strain measures no longer apply, and the equilibrium equations must be written for the deformed geometry. This is true even if the stresses are small.
  - **Material nonlinearity:** when a material is strained beyond its proportional limit, the stress-strain relationship is no longer linear. Plastic materials strained beyond the yield point may exhibit history-dependent behavior. Material nonlinearity may affect the load-deflection behavior of a structure even when the equilibrium equations for the original geometry are still valid.
  - **Other effects:** Other sources of nonlinearity are also possible, including nonlinear loads, boundary conditions and constraints.

Causes of Geometric Nonlinearity

- There are two causes of geometric nonlinearity, the first based on equilibrium and the second on compatibility (continuity).
- Geometric nonlinearity occurs when the displacements of a structure are large enough to affect one or both of the following.
  - **(1) The equilibrium relationships.** Equilibrium in the deformed position of the structure may be significantly different from that in the undeformed position.
  - **(2) The compatibility relationships.** The relationships between element deformations and element end displacements may be significantly nonlinear.

Causes of Geometric Nonlinearity: Equilibrium

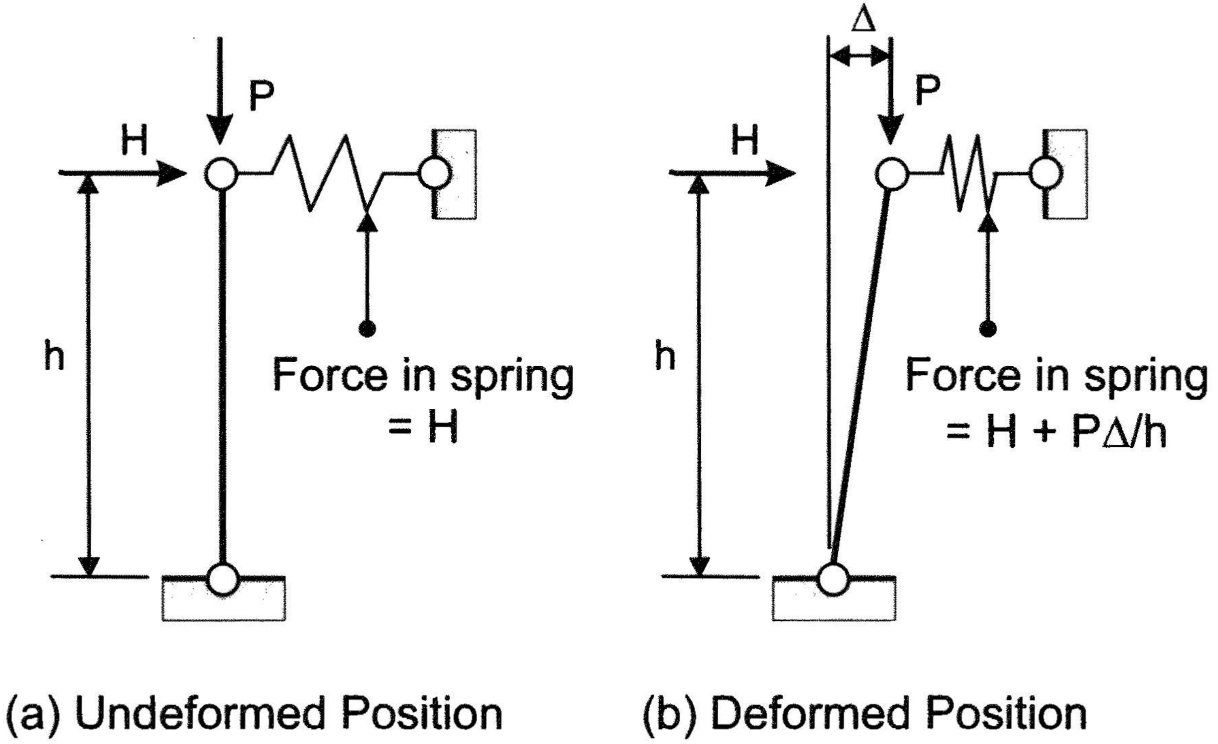
- Strictly speaking, equilibrium between external loads and internal forces must be satisfied in the **deformed position of the structure**.
- However, if the displacements are small, it can be a reasonable approximation to consider equilibrium in the initial, undeformed position.
- Since this position is fixed, the equilibrium relationships are linear. For example, doubling the external loads exactly doubles the internal forces (assuming no material nonlinearity).



(a) Undeformed Position (b) Deformed Position

Causes of Geometric Nonlinearity: Equilibrium

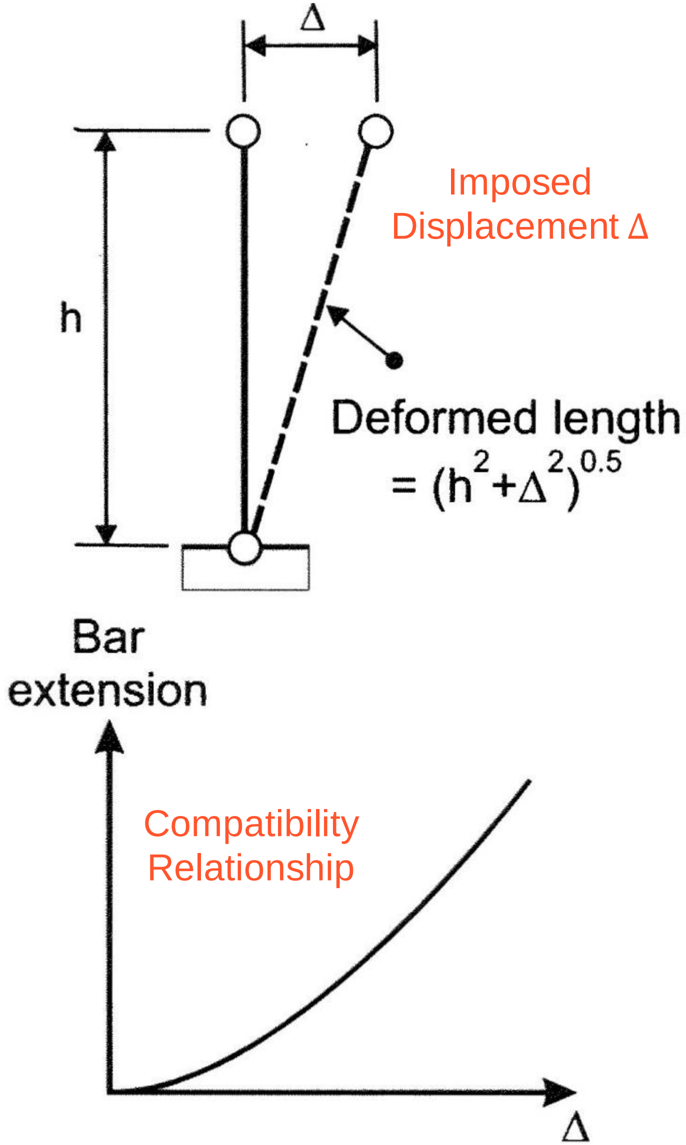
- Figure (a) shows the undeformed position. The bending moment at the pinned base must be zero, so by simple equilibrium the force in the spring is equal to the horizontal load.
- Figure (b) shows the deformed position, assuming that the spring compresses and the top of the bar moves horizontally by an amount Δ. Again, the bending moment at the base is zero, so to satisfy equilibrium the force in the spring must be larger than the applied load. Also, the spring force is not proportional to the load. For example, if P and H are doubled, the force in the spring more than doubles.



(a) Undeformed Position (b) Deformed Position

Causes of Geometric Nonlinearity: Compatibility (Continuity)

- There is a geometrical relationship between the displacements of a structure and the deformations of its components. Figure shows such a relationship.
- In Figure, the top of the bar moves horizontally. Hence, the bar must extend to maintain continuity. Figure (b) shows the relationship between displacement and bar extension. The bar extension is the deformed length minus the undeformed length, h.
- For a very small horizontal displacement the bar extension is close to zero (in the limit, for a vanishingly small displacement, the bar extension is exactly zero). For larger displacements the bar extends, with a nonlinear relationship between displacement and extension.



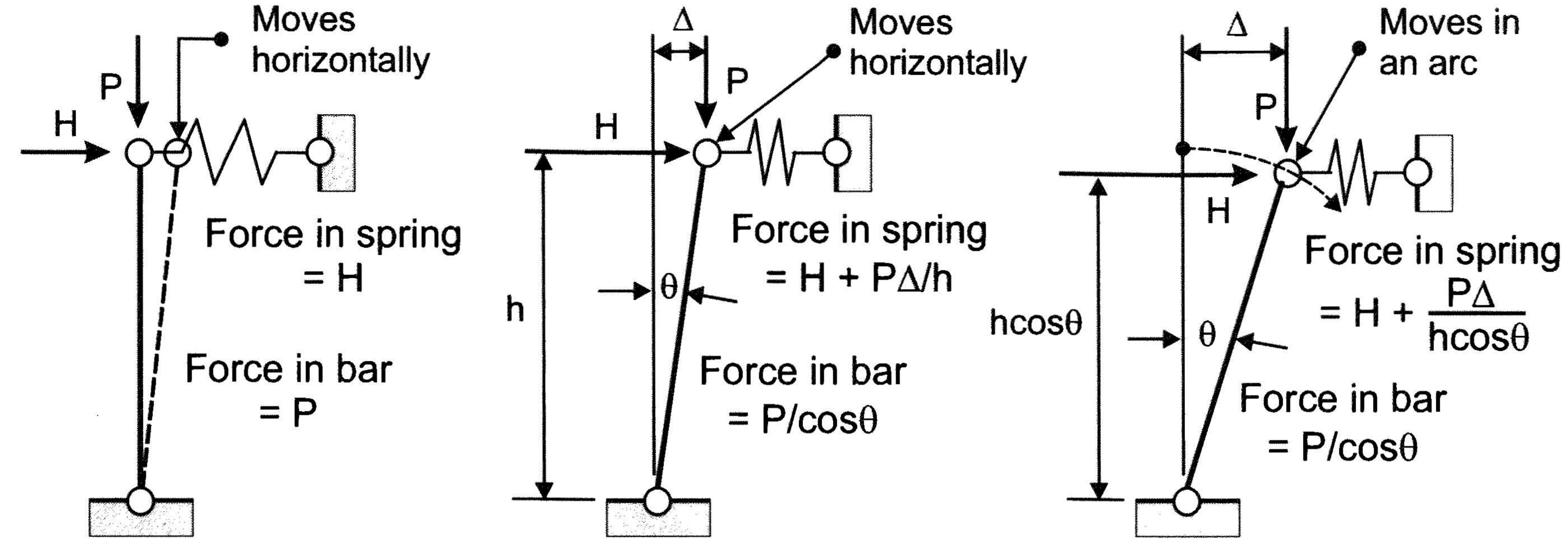
Imposed Displacement Δ

Compatibility Relationship

Analysis Types to Account for Geometric Nonlinearity

- For analysis, the effects of large displacements on the equilibrium and compatibility relationships can be treated separately.
- Consequently, there are three different types of analysis that can be carried out, as follows.
  - **(1) Small displacements analysis**. This is one extreme. Equilibrium is considered in the undeformed position, and for compatibility the displacements are assumed to be vanishingly small.
  - **(2) True large displacements analysis**. This is the other extreme. Equilibrium is considered in the deformed position, and for compatibility the displacements are assumed to be finite. The compatibility relationships are nonlinear. In this case, geometric nonlinearity is considered with no approximations.
  - **(3) P-Δ analysis**. This is in the middle. Equilibrium is considered in the deformed position (with some minor approximations), and the compatibility relationships are assumed to be linear. In this case, geometric nonlinearity is considered approximately.
- There is a fourth type (deformed position for equilibrium, small displacements for compatibility), but this is never used.

Analysis Types to Account for Geometric Nonlinearity



(a) Small Displacements (b) P-Δ (c) Large Displacements

Assume that the bar is stiff axially, so that it has negligible axial deformation.

Analysis Types to Account for Geometric Nonlinearity

- The differences among the three methods depend on the relative values of the loads P and H, and on the displacement Δ. Consider two example cases as follows.
  - (1) P = 0, and Δ/h = 0.1 (i.e., 10% drift ratio, which is a very large drift for most structures). For all three methods the force in the spring is H and the force in the bar is zero. The only difference is that the vertical displacement is negligible for small displacements and P-Δ. analysis, and equal to a small value (0.005h) for large displacements analysis.
  - 2) P/H = 5, Δ/h = 0.10. For the small displacements case the forces in the spring and bar are respectively H and P. For the P-Δ case the forces are 1.5H and 0.995P. For the large displacements case the forces are 1.503H and 0.995P. The vertical displacements are essentially the same as for P = 0.
- These examples show that small displacements analysis can be in error when there are substantial gravity loads and large drifts, but only in the force in the spring (in the second example above there is an error of 50% in the spring force).

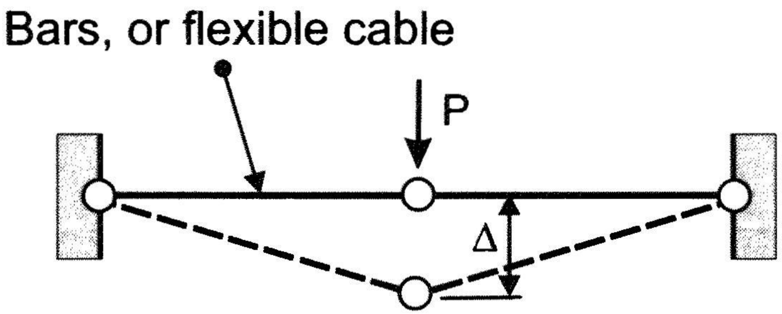
Analysis Types to Account for Geometric Nonlinearity

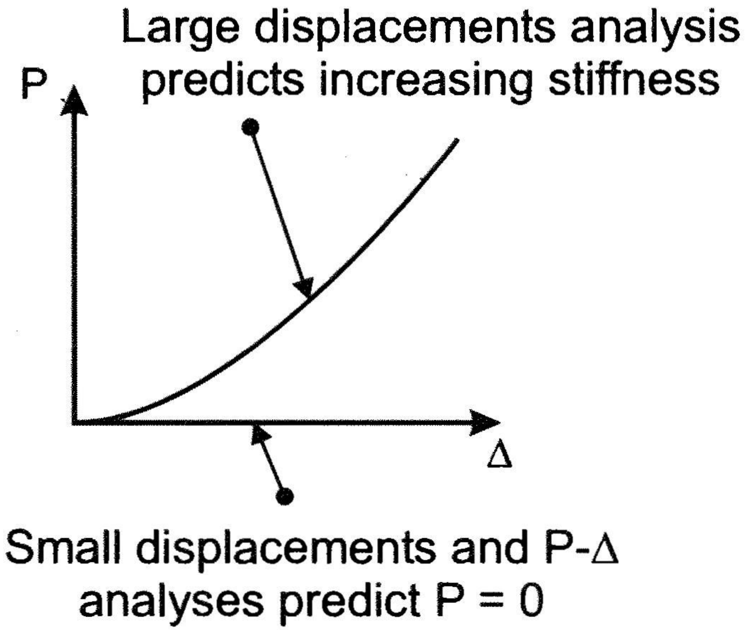
- For all three analyses the axial force in the bar is very close to P (because case is very close to 1.0).
- When P-Δ and large displacement analyses are compared there is very little difference in the spring forces.
- The only significant difference is that the calculated vertical deflection is zero for P-Δ. analysis and a small value for large displacements analysis.
- These examples roughly represent a single-story building structure (the spring models the horizontal stiffness). They indicate that it can be important to consider P-Δ effects, but that it is not necessary to consider true large displacements. This is important because P-Δ analysis can be much more efficient computationally than large displacements analysis.
- For building structures under gravity plus lateral loads, it is often important to consider P-Δ effects, but it is rarely necessary to consider true large displacements.

Practical Guideline to Account for Geometric Nonlinearity

- If geometric nonlinearity must be considered, **it is almost always accurate enough to use P-Δ analysis**.
- Only for very flexible structures, such as cable structures, is it necessary to use large displacements analysis.
- P-Δ analysis is more efficient computationally than large displacements analysis.
- For most structures, it is a waste of computer time to account for true large displacements.

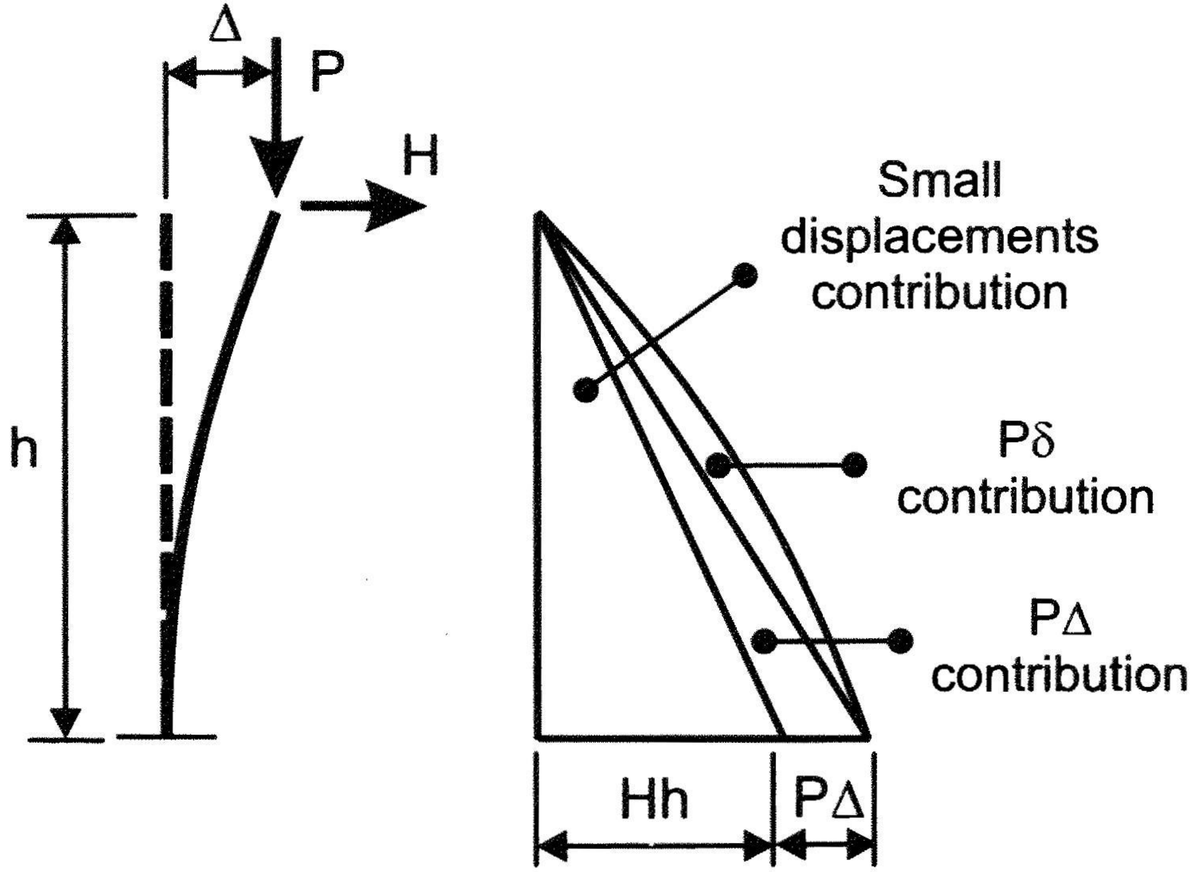
Catenary Effect





- Figure shows behavior of a different type. In this example, as the structure deflects it gets progressively stiffer. This is usually referred to as the **"catenary" effect**.
- Only a large displacements analysis accounts for this effect. The reason is that the small displacements and P-Δ analyses assume a linear compatibility relationship between the structure displacement and the bar extension.
- In this example, the linear compatibility relationship gives zero bar extension, even for large values of Δ. Consequently the bar force is zero, and hence P = 0 in both the undeformed and deformed positions.
- The large displacements analysis uses a nonlinear compatibility relationship, as in Figure, and hence accounts for the catenary effect.

P-Δ vs. P-$\boldsymbol{\delta}$



(a) Column (b) Bending Moments

Geometric Nonlinearity

- The large-stress and large-displacement effects are both termed geometric (or **kinematic**) nonlinearity, as distinguished from material nonlinearity. Kinematic nonlinearity may also be referred to as **second-order geometric effects**.
- For each nonlinear static and nonlinear direct integration time-history analysis, you may choose to consider:
  - No geometric nonlinearity
  - P-delta effects only
  - Large displacement and P-delta effects

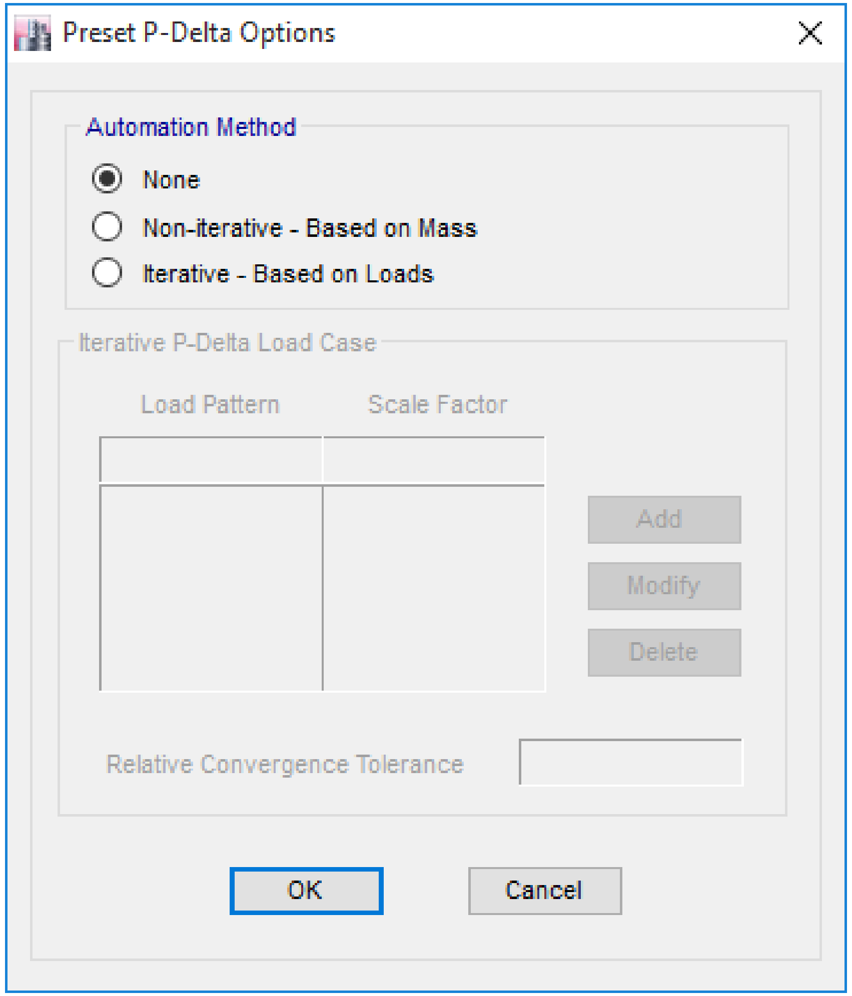
Material Nonlinearity

- Material nonlinearity has a wide range of causes, many of which are poorly understood, and it is not governed by any single theory.
- Our knowledge of material nonlinearity depends almost entirely on what we observe in experiments on actual structures and structural components.
- Material nonlinearity is subject to judgment and interpretation.

Geometric Nonlinearity

- Geometrical nonlinearity has clear causes and is governed by a well-defined mathematical theory.
- Geometrical nonlinearity has two well-defined causes (equilibrium and compatibility), both of which are governed by clear mathematical rules
- Geometrical nonlinearity is not subject to judgment and interpretation.
- This does not mean, however, that geometrical nonlinearity is easy to account for in an analysis. Its effects can be complex and subtle, and they can be difficult to capture in an analysis model.

P-Delta Options in SAP2000/ETABS



Preset P-Delta Options

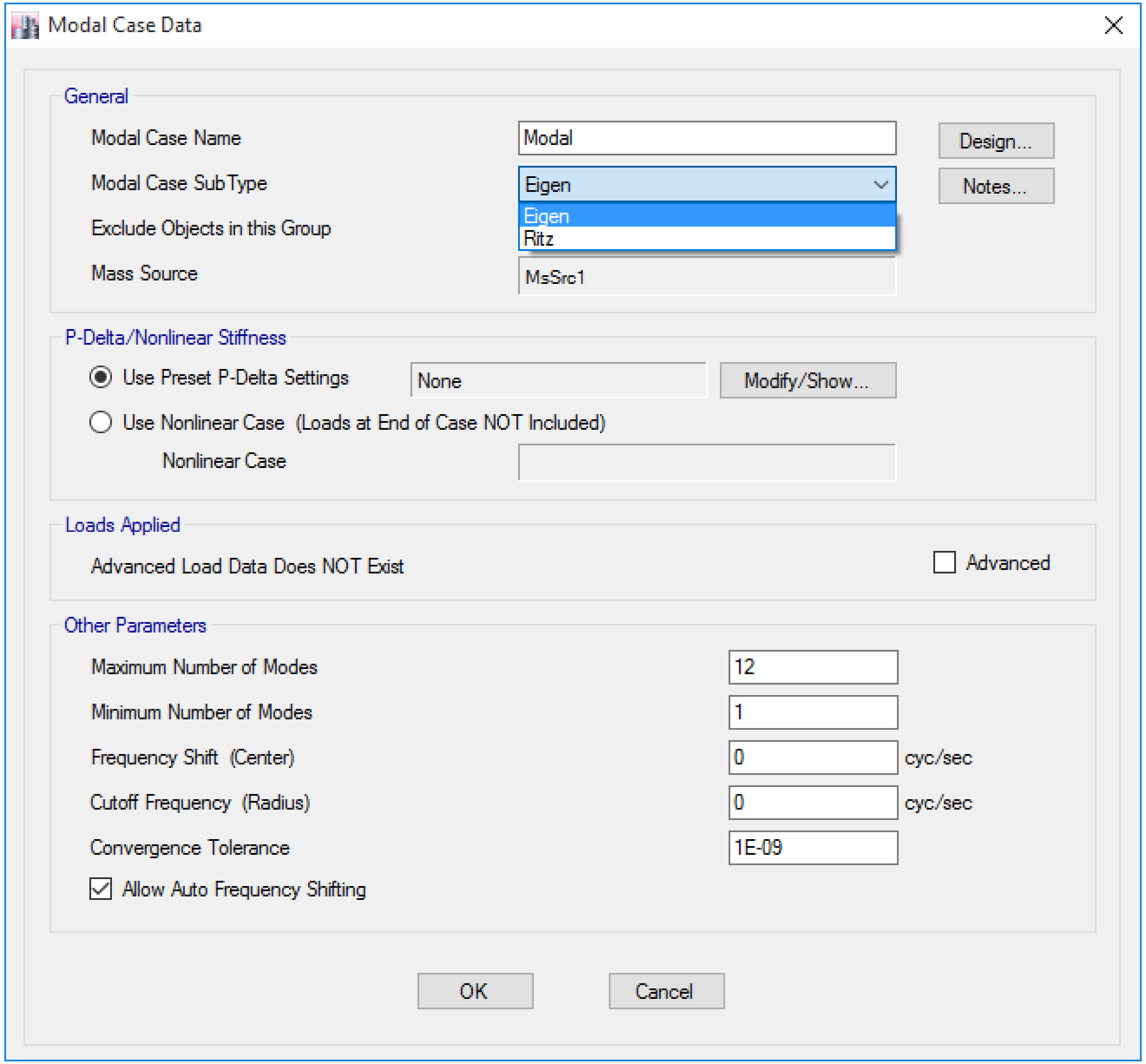
Automation Method

● None

○ Non-iterative - Based on Mass

○ Iterative - Based on Loads

Modal Case



Modal Case Data

General

Modal Case Name: Modal

Modal Case SubType: Eigen

Exclude Objects in this Group

Mass Source: MsSrc1

P-Delta/Nonlinear Stiffness

● Use Preset P-Delta Settings: None

○ Use Nonlinear Case (Loads at End of Case NOT Included)

Nonlinear Case

Loads Applied

Advanced Load Data Does NOT Exist

☐ Advanced

Other Parameters

Maximum Number of Modes: 12

Minimum Number of Modes: 1

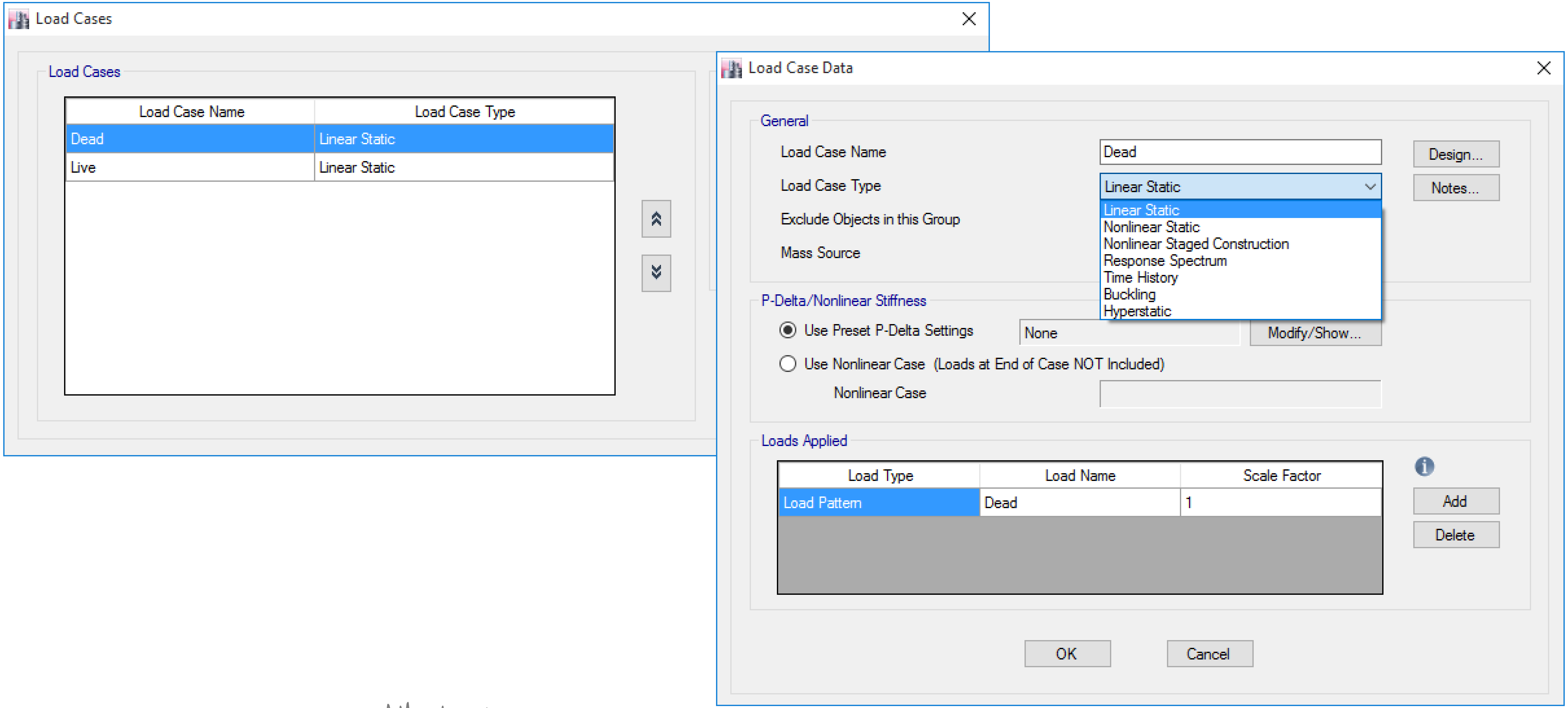
Frequency Shift (Center): 0 cyc/sec

Cutoff Frequency (Radius): 0 cyc/sec

Convergence Tolerance: 1E-09

☑ Allow Auto Frequency Shifting

Load Cases



Load Cases

Load Case Name	Load Case Type
Dead	Linear Static
Live	Linear Static

Load Case Data

General

Load Case Name: Dead

Load Case Type: Linear Static

Exclude Objects in this Group

Mass Source

P-Delta/Nonlinear Stiffness

● Use Preset P-Delta Settings: None

○ Use Nonlinear Case (Loads at End of Case NOT Included)

Nonlinear Case

Loads Applied

Load Type	Load Name	Scale Factor
Load Pattern	Dead	1

Load Cases

- Each different analysis performed is called a Load Case. For each Load Case you define, you need to supply the following type of information:
  - **Case name:** This name must be unique across all Load Cases of all types. The case name is used to request analysis results (displacements, stresses, etc.), for creating Load Combinations, and sometimes for use by other dependent Load Cases.
  - **Analysis type:** This indicate the type of analysis (static, response-spectrum, buckling, etc.), as well as available options for that type (linear, nonlinear, etc.).
  - **Prerequisite load cases:** Some load cases may continue from a previous load case, use the stiffness from a previous load case, and/or use the modes from a previous load case.
  - **Loads applied:** For most types of analysis, you specify the Load Patterns that are to be applied to the structure.
- Additional data may be required, depending upon the type of analysis being defined.

Linear and Nonlinear Load Cases

- Every Load Case is considered to be either **linear** or **nonlinear**.
- **Structural properties**
  - Linear: Structural properties (stiffness, damping, etc.) are constant during the analysis.
  - Nonlinear: Structural properties may vary with time, deformation, and loading. How much nonlinearity actually occurs depends upon the properties you have defined, the magnitude of the loading, and the parameters you have specified for the analysis.
- **Initial conditions**
  - Linear: The analysis starts with zero stress. It does not contain loads from any previous analysis, even if it uses the stiffness from a previous nonlinear analysis.
  - Nonlinear: The analysis may continue from a previous nonlinear analysis, in which case it contains all loads, deformations, stresses, etc., from that previous case.
- **Structural response and superposition**
  - Linear: All displacements, stresses, reactions, etc., are directly proportional to the magnitude of the applied loads. The results of different linear analyses may be superposed.
  - Nonlinear: Because the structural properties may vary, and because of the possibility of non-zero initial conditions, the response may not be proportional to the loading. Therefore, the results of different nonlinear analyses should not usually be superposed.

Sequence of Analysis

- A Load Case may be **dependent** upon other Load Cases in the following situations:
  - A modal-superposition type of Load Case (response-spectrum or modal time-history) uses the modes calculated in a modal Load Case.
  - A nonlinear Load Case may continue from the state at the end of another nonlinear case.
- Sequence of Analysis: A linear Load Cases may use the stiffness of the structure as computed at the end of a nonlinear case.
- A Load Case which depends upon an other case is called **dependent**. The case upon which it depends is called a **prerequisite** case.

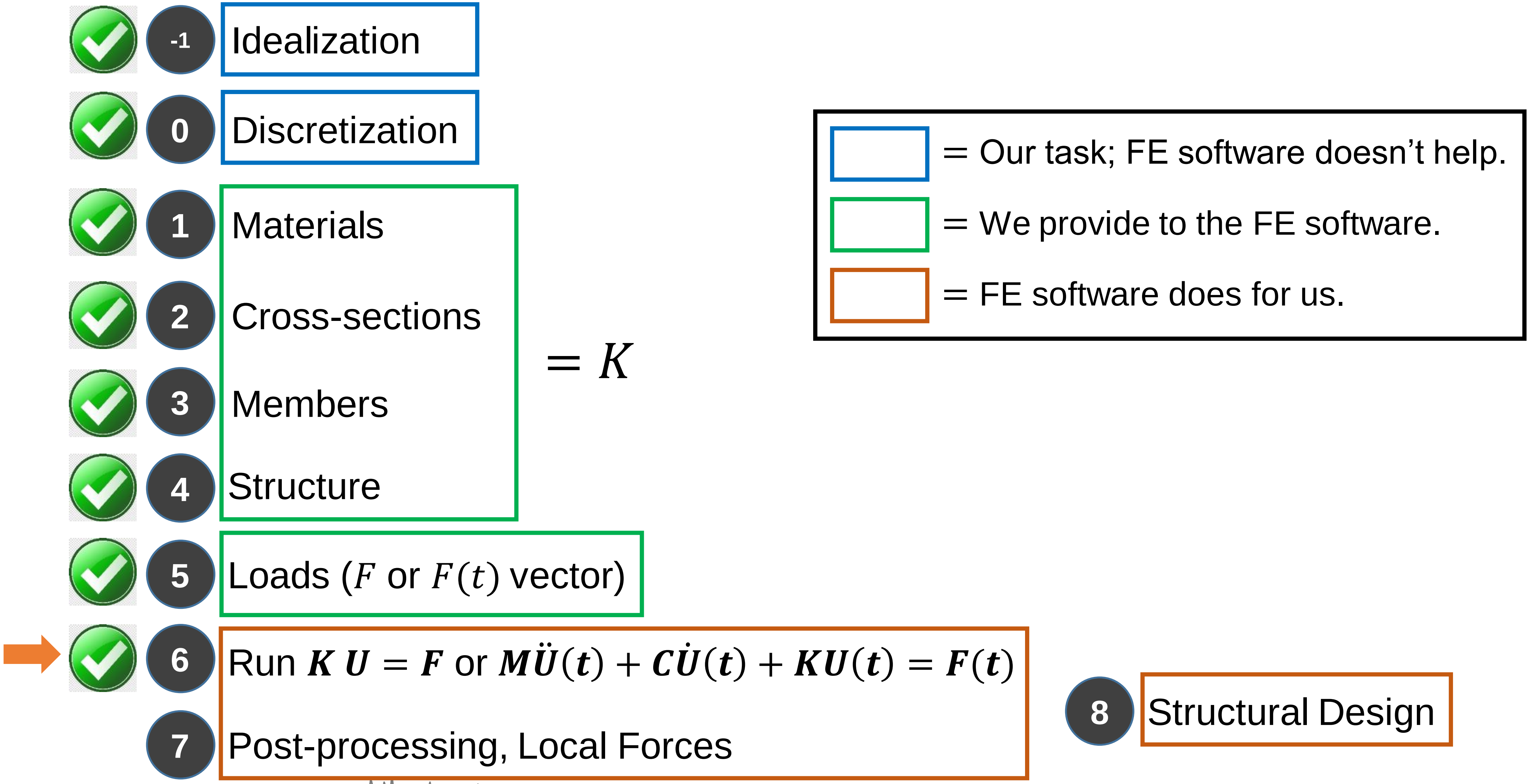
Load Functions

- A **Function** is a series of digitized abscissa- ordinate pairs that may represent:
  - Pseudo- spectral acceleration vs. period for response-spectrum analysis, or
  - Load vs. time for time-history analysis
  - Load vs. frequency for steady-state analysis
  - Power density (load squared per frequency) vs. frequency for power-spectral-density analysis

Load Combinations

- Five types of Combos are available. For each individual response quantity (force, stress, or displacement component) the two Combo values are calculated as follows:
  - **Additive type:** The Combo maximum is an algebraic linear combination of the maximum values for each of the contributing cases. Similarly, Combo minimum is an algebraic linear combination of the minimum values for each of the contributing cases.
  - **Absolute type:** The Combo maximum is the sum of the larger absolute values for each of the contributing cases. The Combo minimum is the negative of the Combo maximum.
  - **SRSS type:** The Combo maximum is the square root of the sum of the squares of the larger absolute values for each of the contributing cases. The Combo minimum is the negative of the Combo maximum.
  - **Range type:** The Combo maximum is the sum of the positive maximum values for each of the contributing cases (a case with a negative maximum does not contribute.) Similarly, the Combo minimum is the sum of the negative minimum values for each of the contributing cases (a case with a positive minimum does not contribute.)
  - **Envelope type:** The Combo maximum is the maximum of all of the maximum values for each of the contributing cases. Similarly, Combo minimum is the minimum of all of the minimum values for each of the contributing cases.

Finite Element Modeling, Analysis and Design Process



✓ -1 Idealization

✓ 0 Discretization

✓ 1 Materials

✓ 2 Cross-sections

✓ 3 Members

✓ 4 Structure

$= K$

✓ 5 Loads (F or $F(t)$ vector)

✓ 6 Run $\boldsymbol{K\ U = F}$ or $\boldsymbol{M\ddot{U}(t) + C\dot{U}(t) + KU(t) = F(t)}$

7 Post-processing, Local Forces

8 Structural Design

🟦 = Our task; FE software doesn’t help.

🟩 = We provide to the FE software.

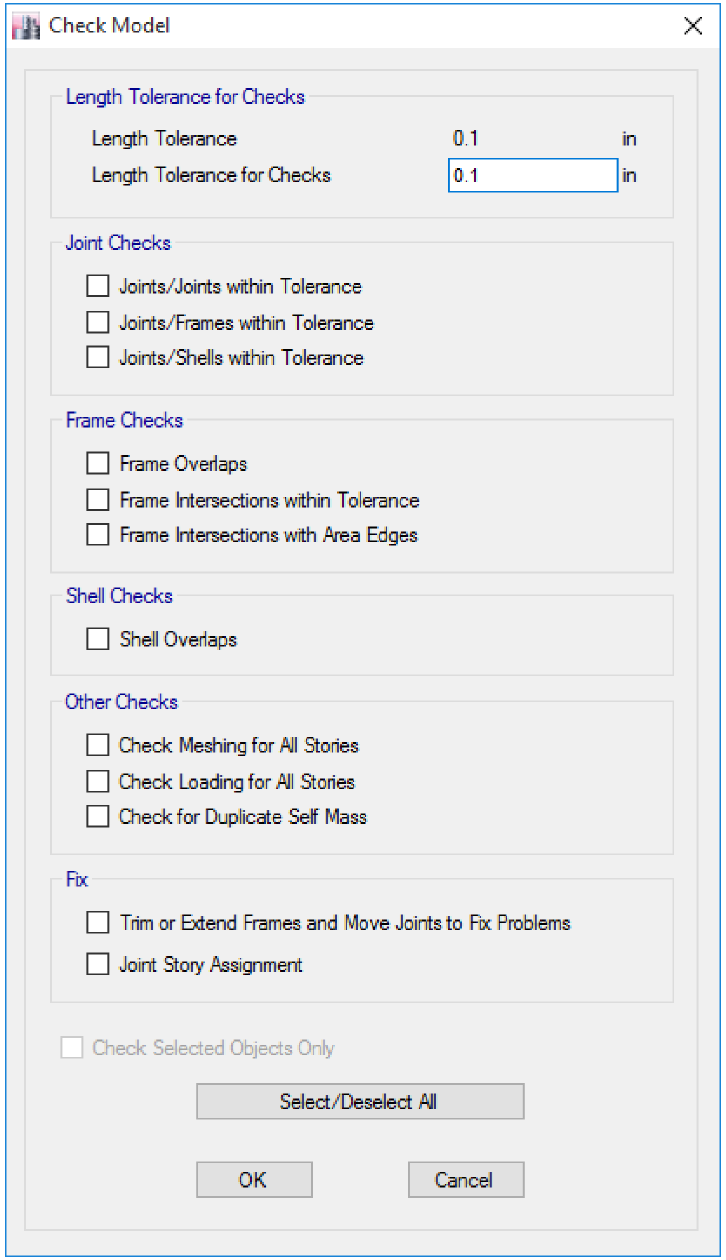
🟧 = FE software does for us.

Linear Elastic Load Cases

- The results of linear analyses may be **superposed**, i.e., added together after analysis.
- The available types of linear analysis are:
  - Static analysis
  - Modal analysis for vibration modes, using eigen vectors or Ritz vectors
  - Response-spectrum analysis for seismic response
  - Time- history dynamic response analysis
  - Buckling-mode analysis
  - Moving-load analysis for bridge and other vehicle live loads
  - Steady-state analysis
  - Power-spectral-density analysis

Nonlinear Load Cases

- The results of nonlinear analyses **should not normally be superposed**.
- Instead, all loads acting together on the structure should be combined directly within the Load Cases.
- Nonlinear Load Cases may be **chained together** to represent complex loading sequences.
- The available types of nonlinear analyses are:
  - Nonlinear static analysis
  - Nonlinear time-history analysis



Check Model

Length Tolerance for Checks

Length Tolerance 0.1 in

Length Tolerance for Checks 0.1 in

Joint Checks

☐ Joints/Joints within Tolerance
☐ Joints/Frames within Tolerance
☐ Joints/Shells within Tolerance

Frame Checks

☐ Frame Overlaps
☐ Frame Intersections within Tolerance
☐ Frame Intersections with Area Edges

Shell Checks

☐ Shell Overlaps

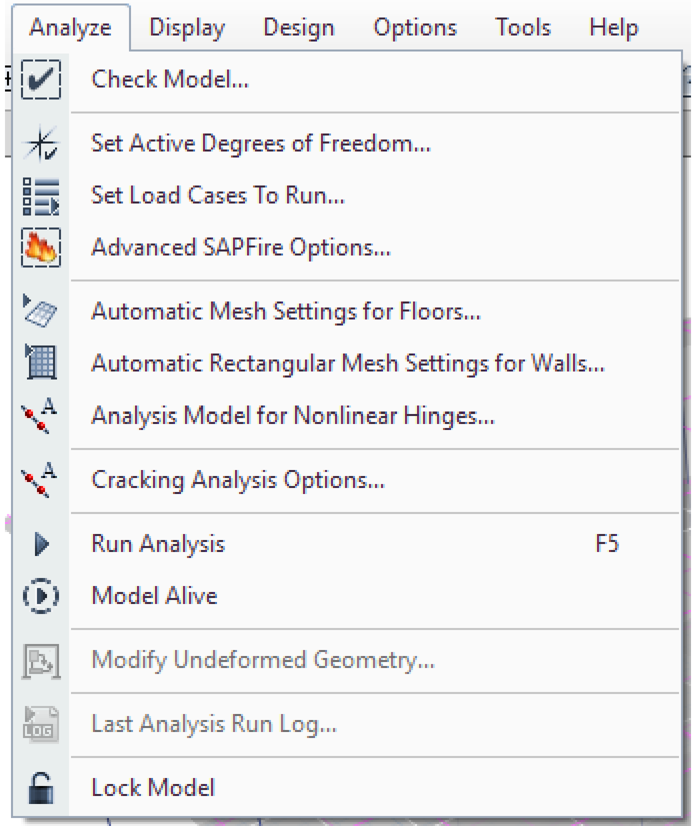
Other Checks

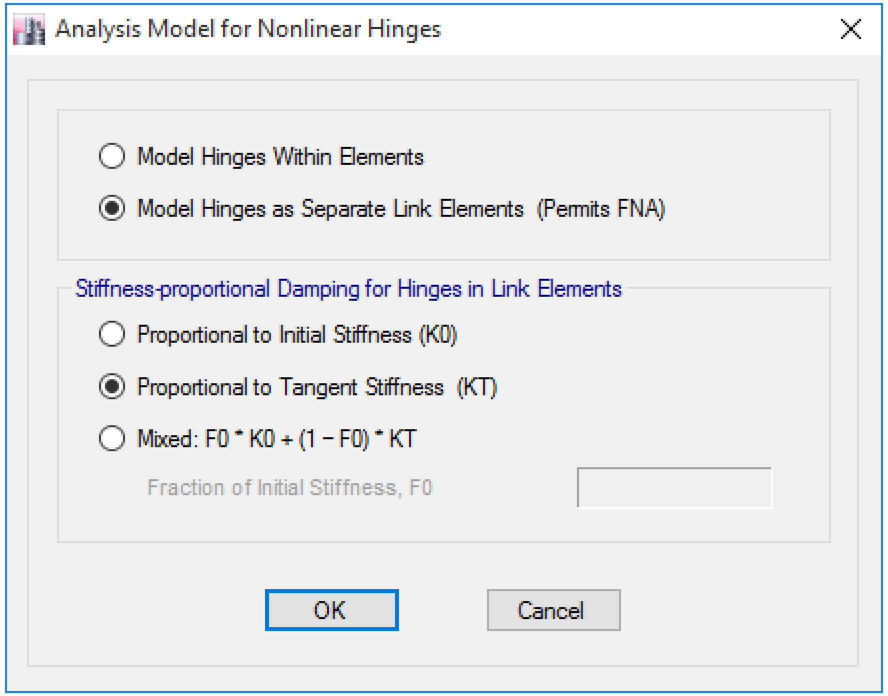
☐ Check Meshing for All Stories
☐ Check Loading for All Stories
☐ Check for Duplicate Self Mass

Fix

☐ Trim or Extend Frames and Move Joints to Fix Problems
☐ Joint Story Assignment

☐ Check Selected Objects Only





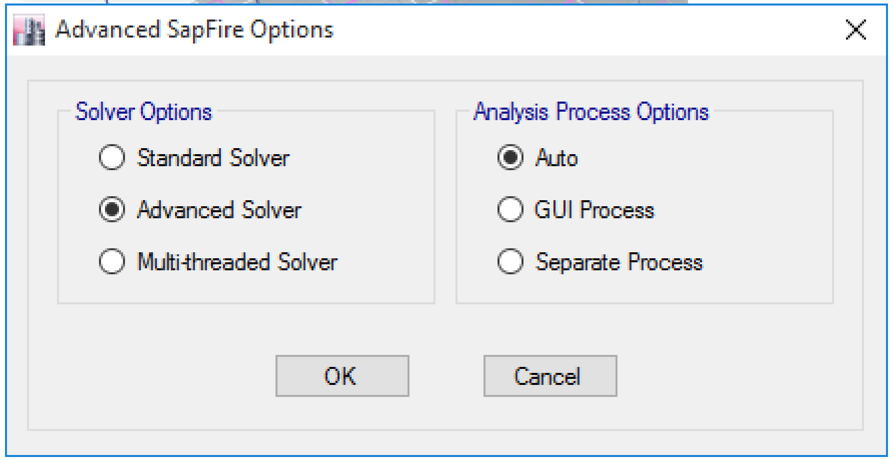
Analysis Model for Nonlinear Hinges

○ Model Hinges Within Elements
● Model Hinges as Separate Link Elements (Permits FNA)

Stiffness-proportional Damping for Hinges in Link Elements

○ Proportional to Initial Stiffness (K0)
● Proportional to Tangent Stiffness (KT)
○ Mixed: F0 * K0 + (1 – F0) * KT

Fraction of Initial Stiffness, F0



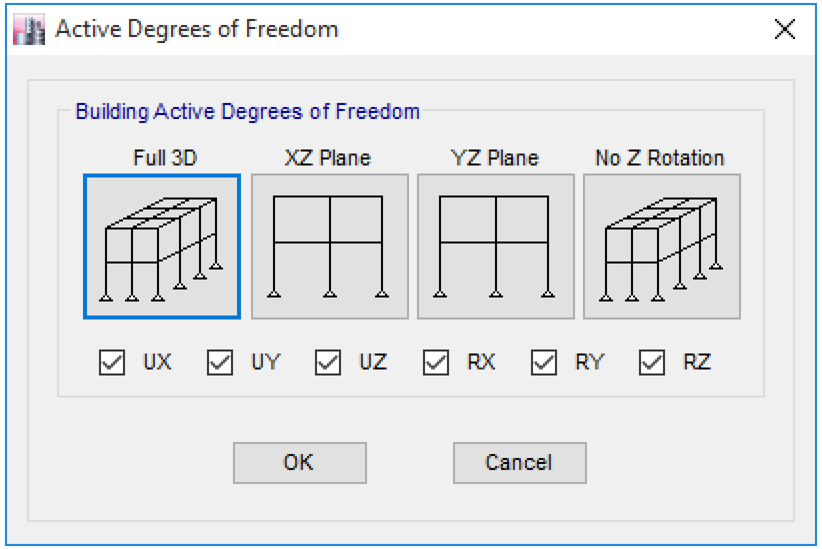
Advanced SapFire Options

Solver Options

○ Standard Solver
● Advanced Solver
○ Multi-threaded Solver

Analysis Process Options

● Auto
○ GUI Process
○ Separate Process

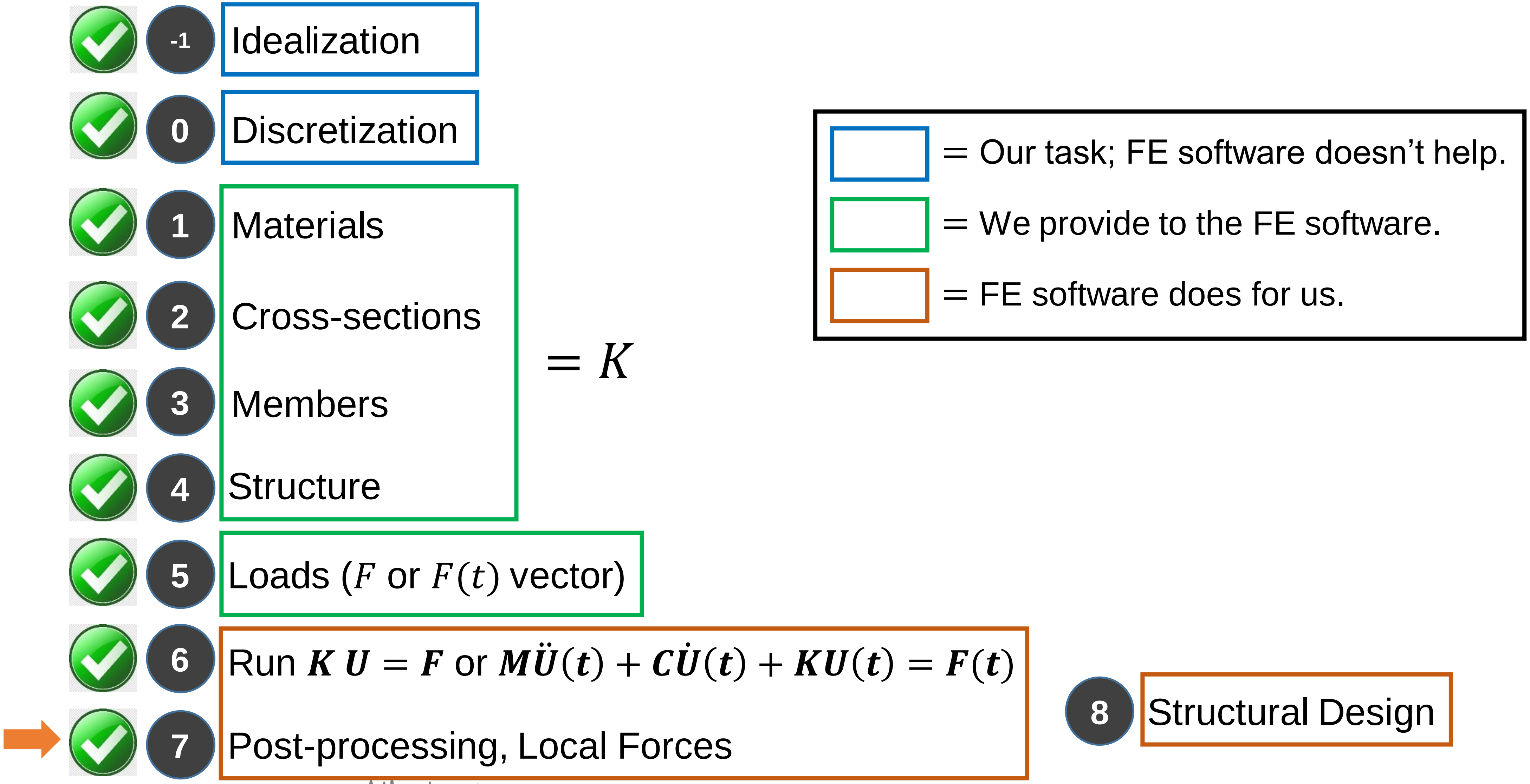


Active Degrees of Freedom

Building Active Degrees of Freedom

☑ UX ☑ UY ☑ UZ ☑ RX ☑ RY ☑ RZ

Finite Element Modeling, Analysis and Design Process



- ✓ -1 Idealization
- ✓ 0 Discretization
- ✓ 1 Materials
- ✓ 2 Cross-sections
- ✓ 3 Members
- ✓ 4 Structure

$= K$

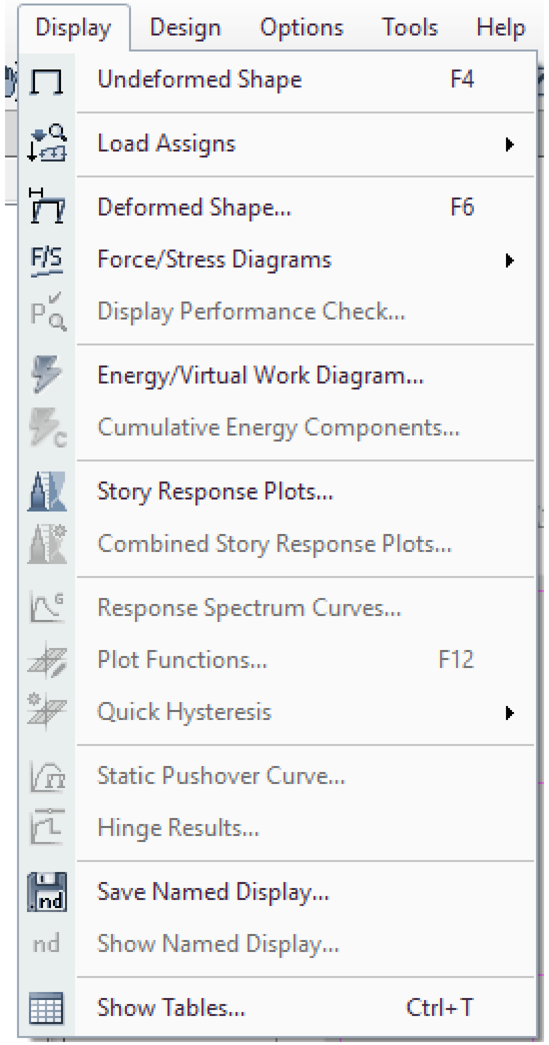
- ✓ 5 Loads (F or $F(t)$ vector)
- ✓ 6 Run $\boldsymbol{K\,U = F}$ or $\boldsymbol{M\ddot{U}(t) + C\dot{U}(t) + KU(t) = F(t)}$
- → ✓ 7 Post-processing, Local Forces
- 8 Structural Design

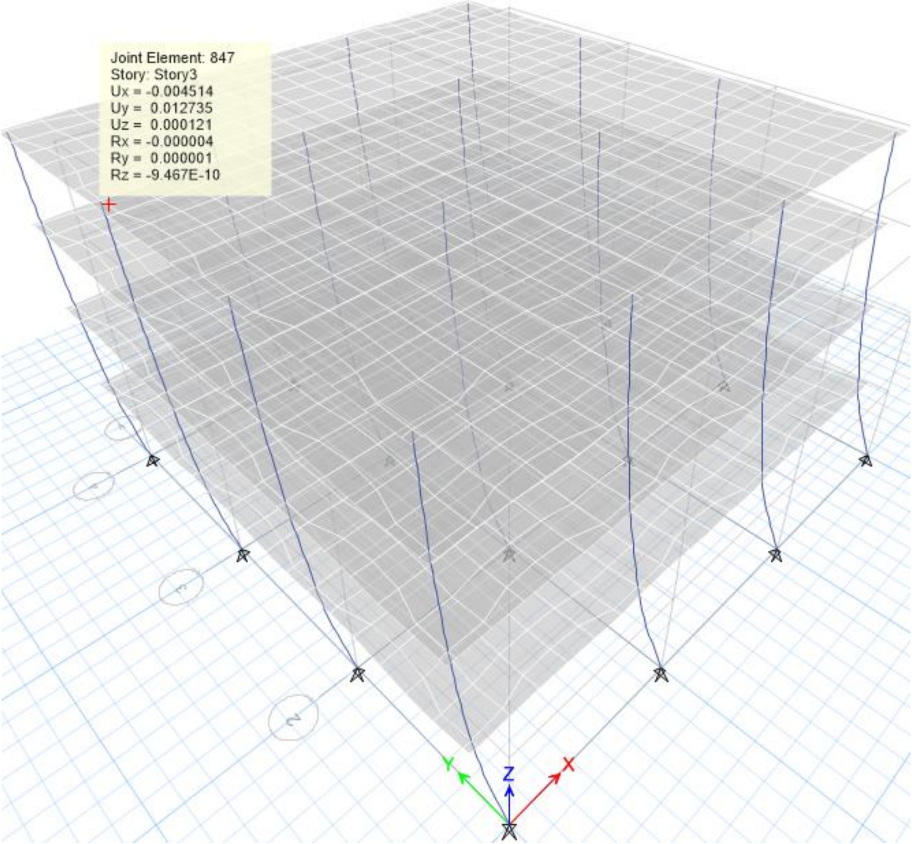
🟦 = Our task; FE software doesn't help.

🟩 = We provide to the FE software.

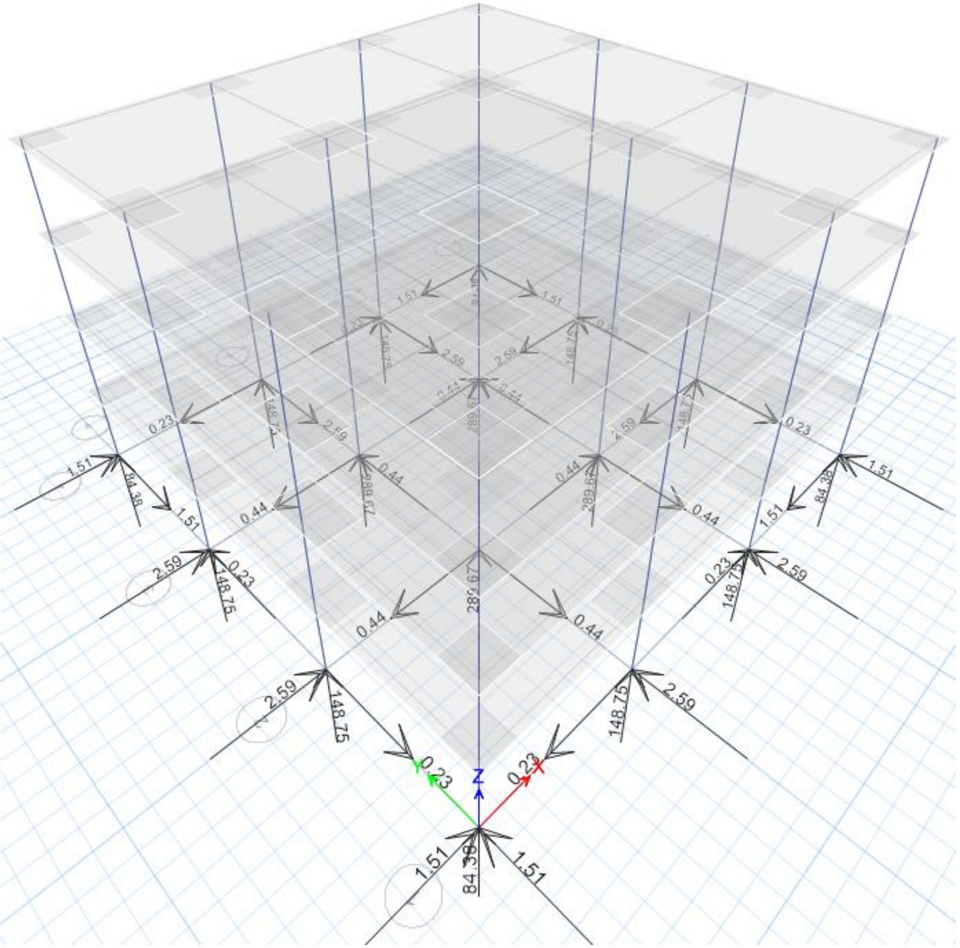
🟧 = FE software does for us.

Analysis Results

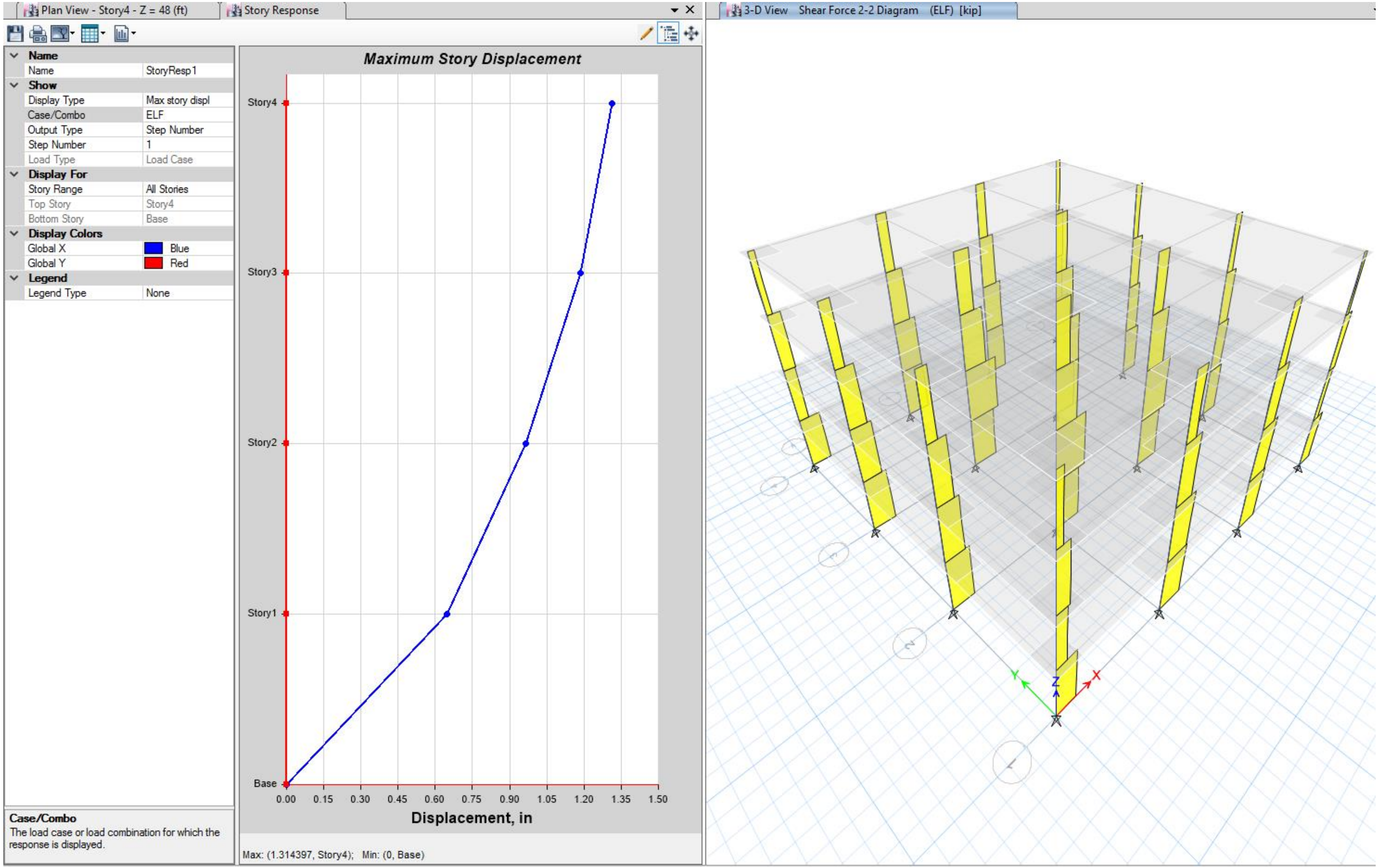




```
Joint Element: 847
Story: Story3
Ux = -0.004514
Uy =  0.012735
Uz =  0.000121
Rx = -0.000004
Ry =  0.000001
Rz = -9.467E-10
```



Results

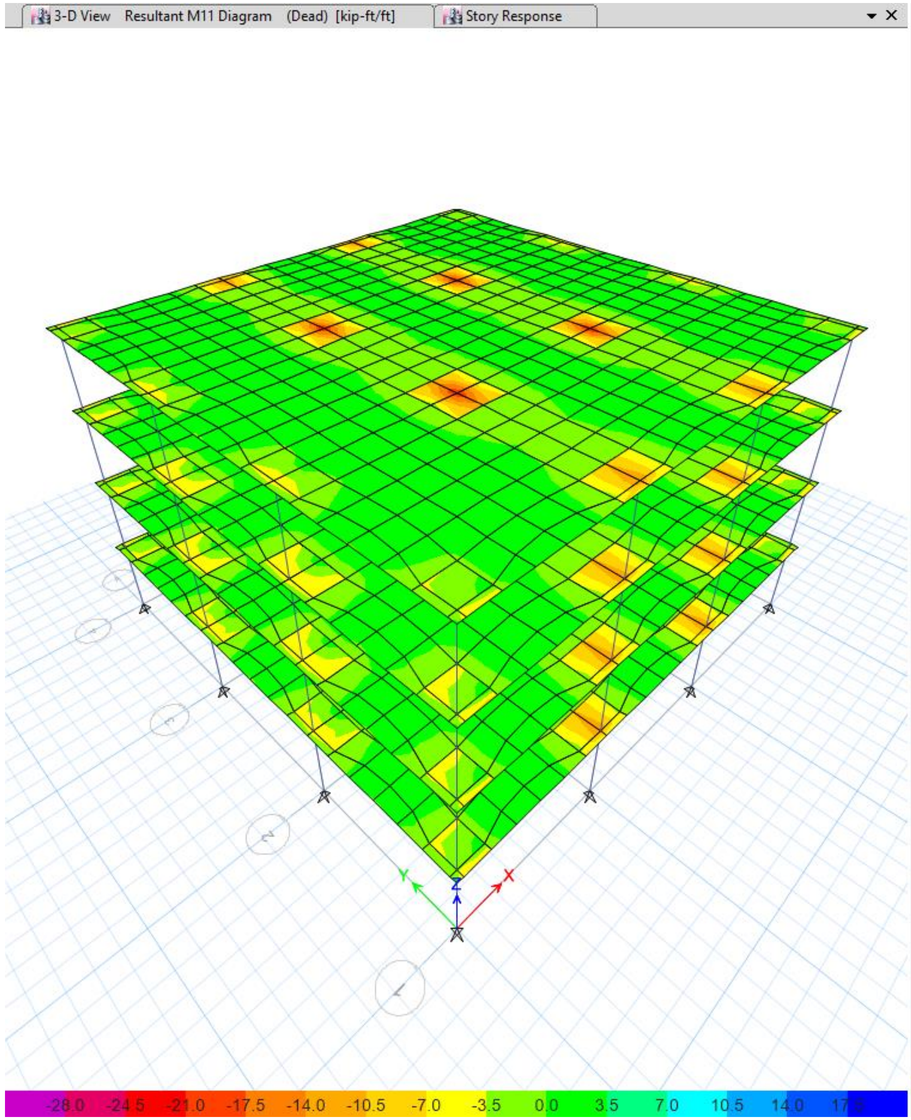


Name	
Name	StoryResp1
Show	
Display Type	Max story displ
Case/Combo	ELF
Output Type	Step Number
Step Number	1
Load Type	Load Case
Display For	
Story Range	All Stories
Top Story	Story4
Bottom Story	Base
Display Colors	
Global X	🟦 Blue
Global Y	🟥 Red
Legend	
Legend Type	None

Case/Combo
The load case or load combination for which the response is displayed.

Max: (1.314397, Story4); Min: (0, Base)

Analysis Results



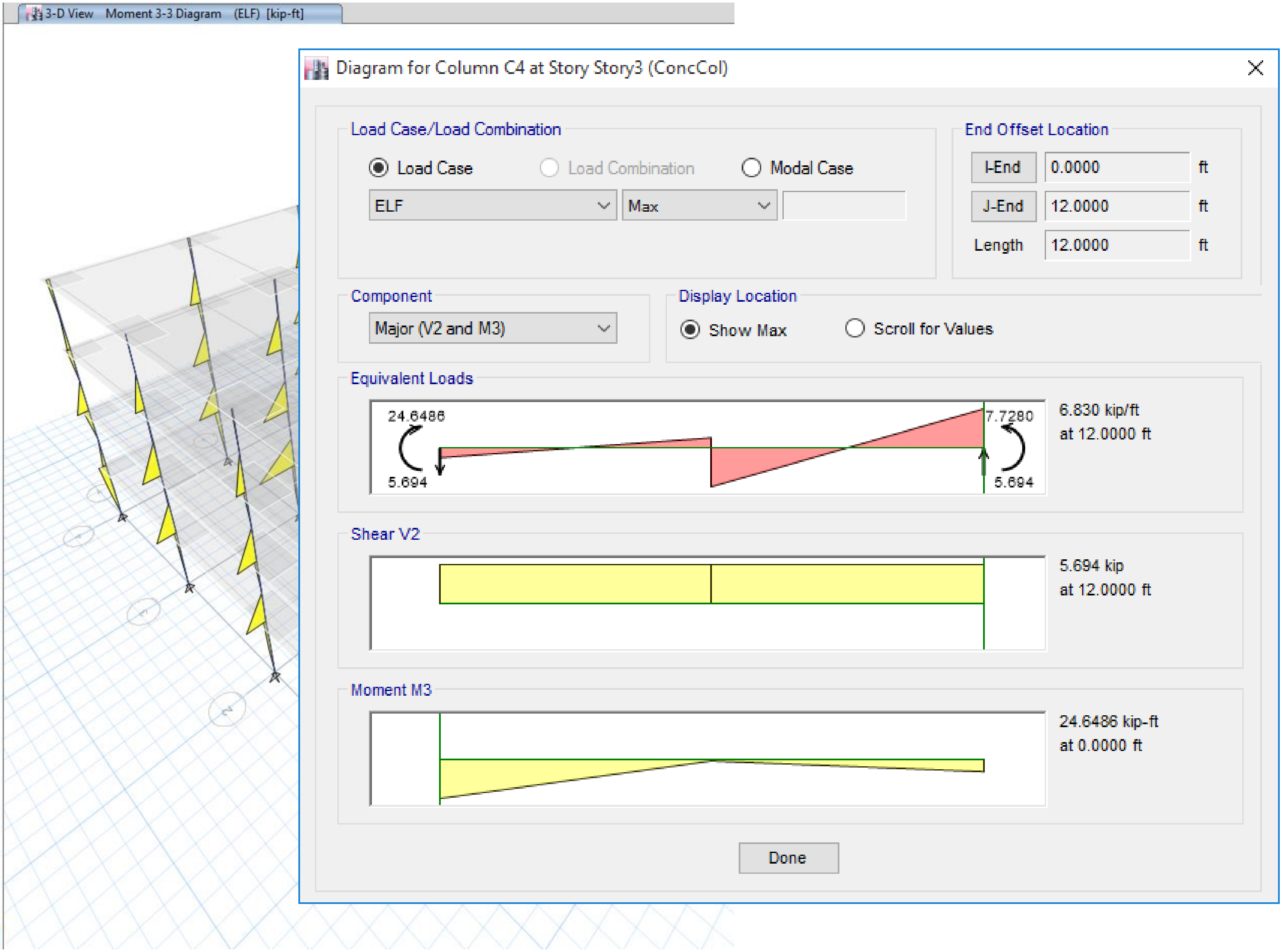


Diagram for Column C4 at Story Story3 (ConcCol)

Load Case/Load Combination

● Load Case ○ Load Combination ○ Modal Case

ELF Max

End Offset Location

I-End 0.0000 ft

J-End 12.0000 ft

Length 12.0000 ft

Component: Major (V2 and M3)

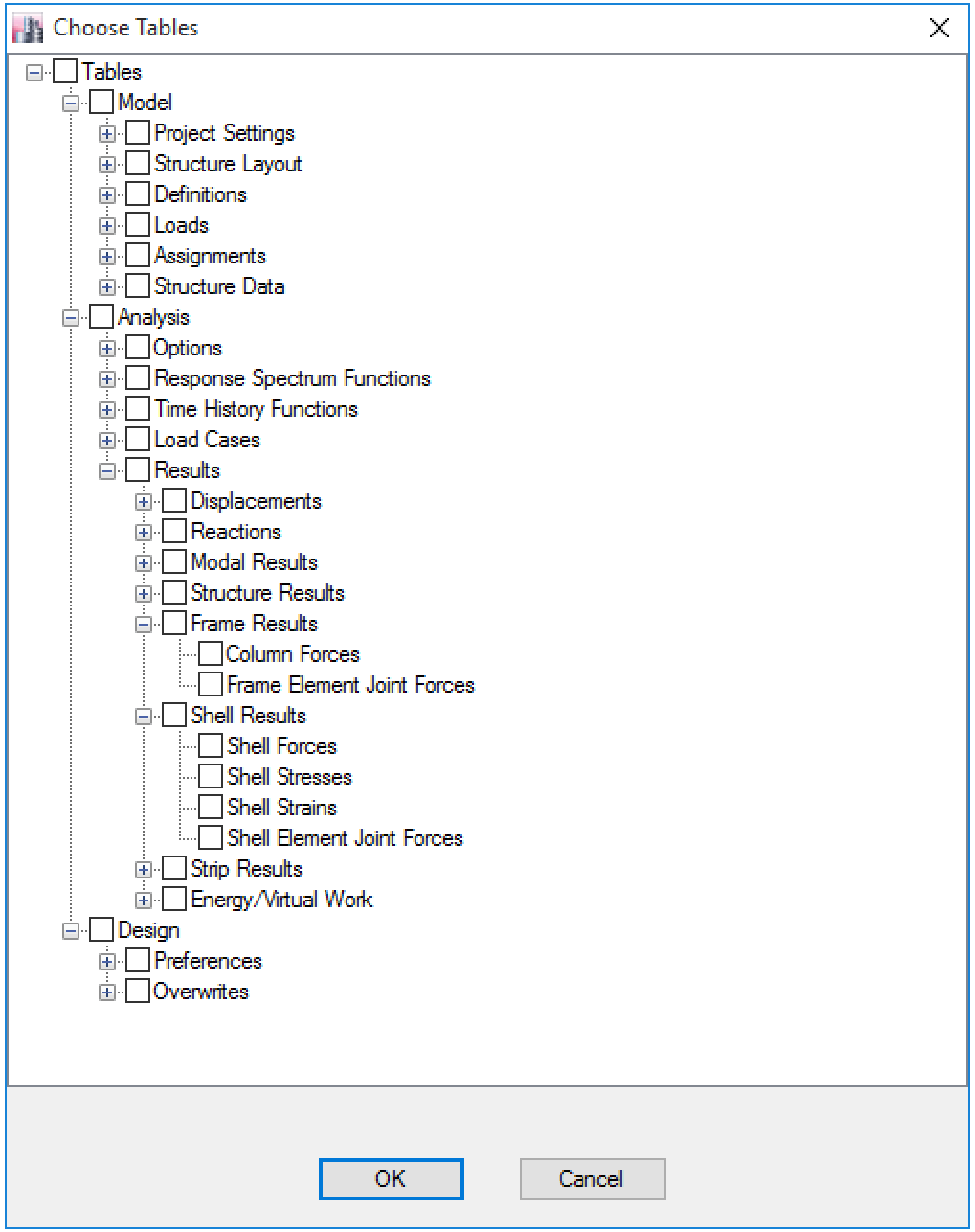
Display Location: ● Show Max ○ Scroll for Values

Equivalent Loads: 6.830 kip/ft at 12.0000 ft

Shear V2: 5.694 kip at 12.0000 ft

Moment M3: 24.6486 kip-ft at 0.0000 ft

Analysis Results

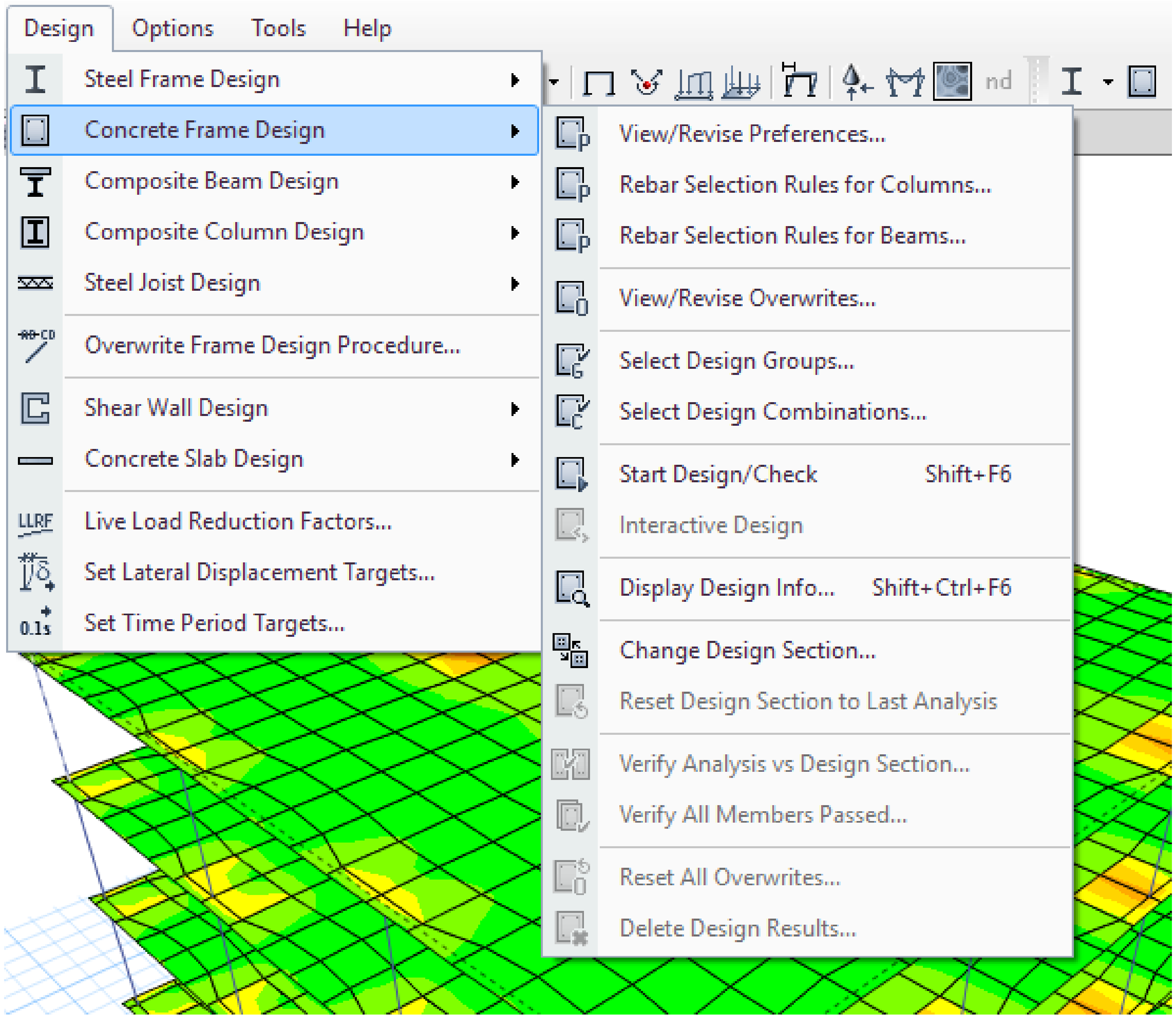


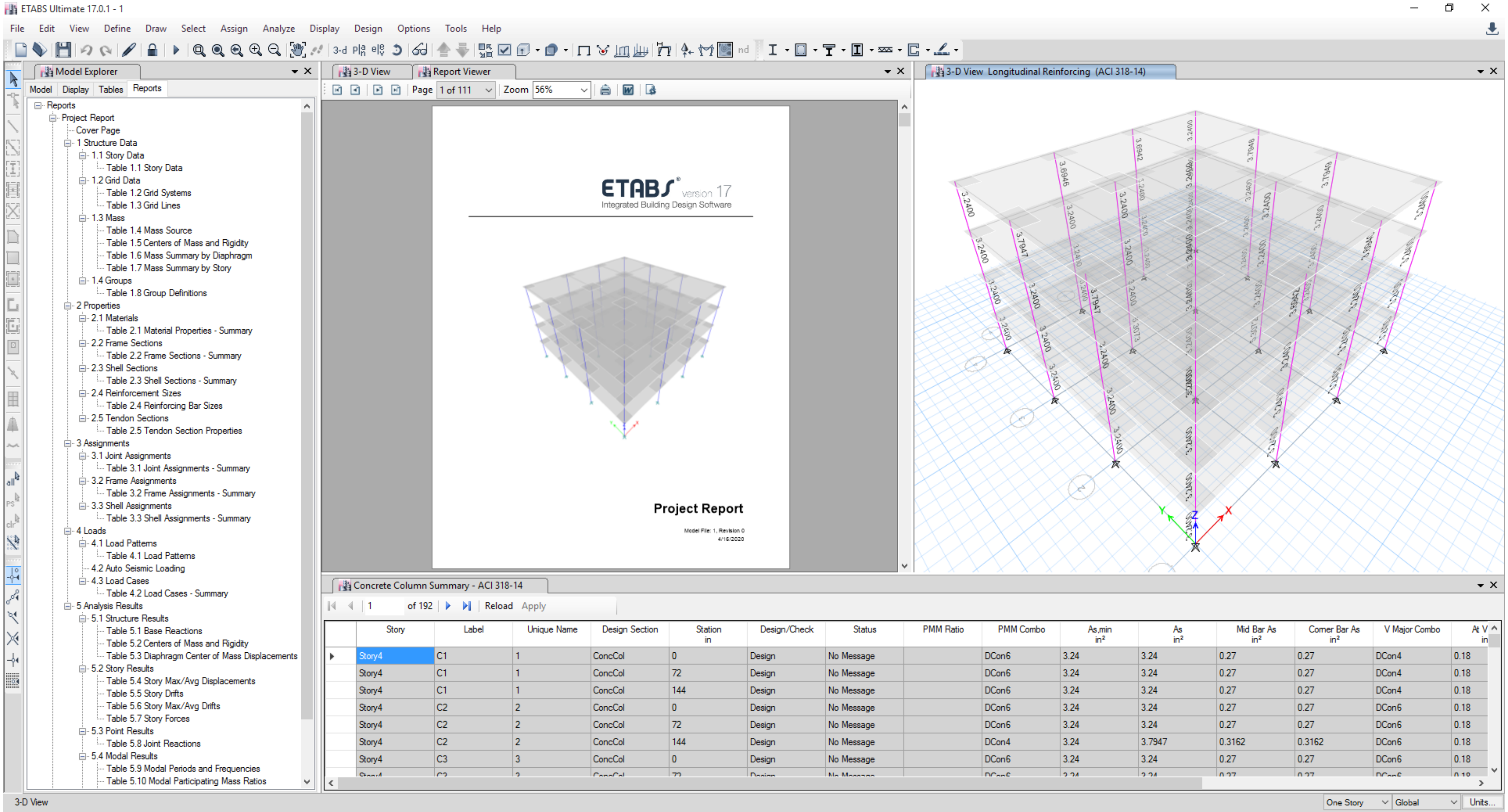
Choose Tables

- ☐ Tables
  - ☐ Model
    - ☐ Project Settings
    - ☐ Structure Layout
    - ☐ Definitions
    - ☐ Loads
    - ☐ Assignments
    - ☐ Structure Data
  - ☐ Analysis
    - ☐ Options
    - ☐ Response Spectrum Functions
    - ☐ Time History Functions
    - ☐ Load Cases
    - ☐ Results
      - ☐ Displacements
      - ☐ Reactions
      - ☐ Modal Results
      - ☐ Structure Results
      - ☐ Frame Results
        - ☐ Column Forces
        - ☐ Frame Element Joint Forces
      - ☐ Shell Results
        - ☐ Shell Forces
        - ☐ Shell Stresses
        - ☐ Shell Strains
        - ☐ Shell Element Joint Forces
      - ☐ Strip Results
      - ☐ Energy/Virtual Work
  - ☐ Design
    - ☐ Preferences
    - ☐ Overwrites

OK | Cancel

Structural Design





- Reports
  - Project Report
    - Cover Page
    - 1 Structure Data
      - 1.1 Story Data
        - Table 1.1 Story Data
      - 1.2 Grid Data
        - Table 1.2 Grid Systems
        - Table 1.3 Grid Lines
      - 1.3 Mass
        - Table 1.4 Mass Source
        - Table 1.5 Centers of Mass and Rigidity
        - Table 1.6 Mass Summary by Diaphragm
        - Table 1.7 Mass Summary by Story
      - 1.4 Groups
        - Table 1.8 Group Definitions
    - 2 Properties
      - 2.1 Materials
        - Table 2.1 Material Properties - Summary
      - 2.2 Frame Sections
        - Table 2.2 Frame Sections - Summary
      - 2.3 Shell Sections
        - Table 2.3 Shell Sections - Summary
      - 2.4 Reinforcement Sizes
        - Table 2.4 Reinforcing Bar Sizes
      - 2.5 Tendon Sections
        - Table 2.5 Tendon Section Properties
    - 3 Assignments
      - 3.1 Joint Assignments
        - Table 3.1 Joint Assignments - Summary
      - 3.2 Frame Assignments
        - Table 3.2 Frame Assignments - Summary
      - 3.3 Shell Assignments
        - Table 3.3 Shell Assignments - Summary
    - 4 Loads
      - 4.1 Load Patterns
        - Table 4.1 Load Patterns
      - 4.2 Auto Seismic Loading
      - 4.3 Load Cases
        - Table 4.2 Load Cases - Summary
    - 5 Analysis Results
      - 5.1 Structure Results
        - Table 5.1 Base Reactions
        - Table 5.2 Centers of Mass and Rigidity
        - Table 5.3 Diaphragm Center of Mass Displacements
      - 5.2 Story Results
        - Table 5.4 Story Max/Avg Displacements
        - Table 5.5 Story Drifts
        - Table 5.6 Story Max/Avg Drifts
        - Table 5.7 Story Forces
      - 5.3 Point Results
        - Table 5.8 Joint Reactions
      - 5.4 Modal Results
        - Table 5.9 Modal Periods and Frequencies
        - Table 5.10 Modal Participating Mass Ratios

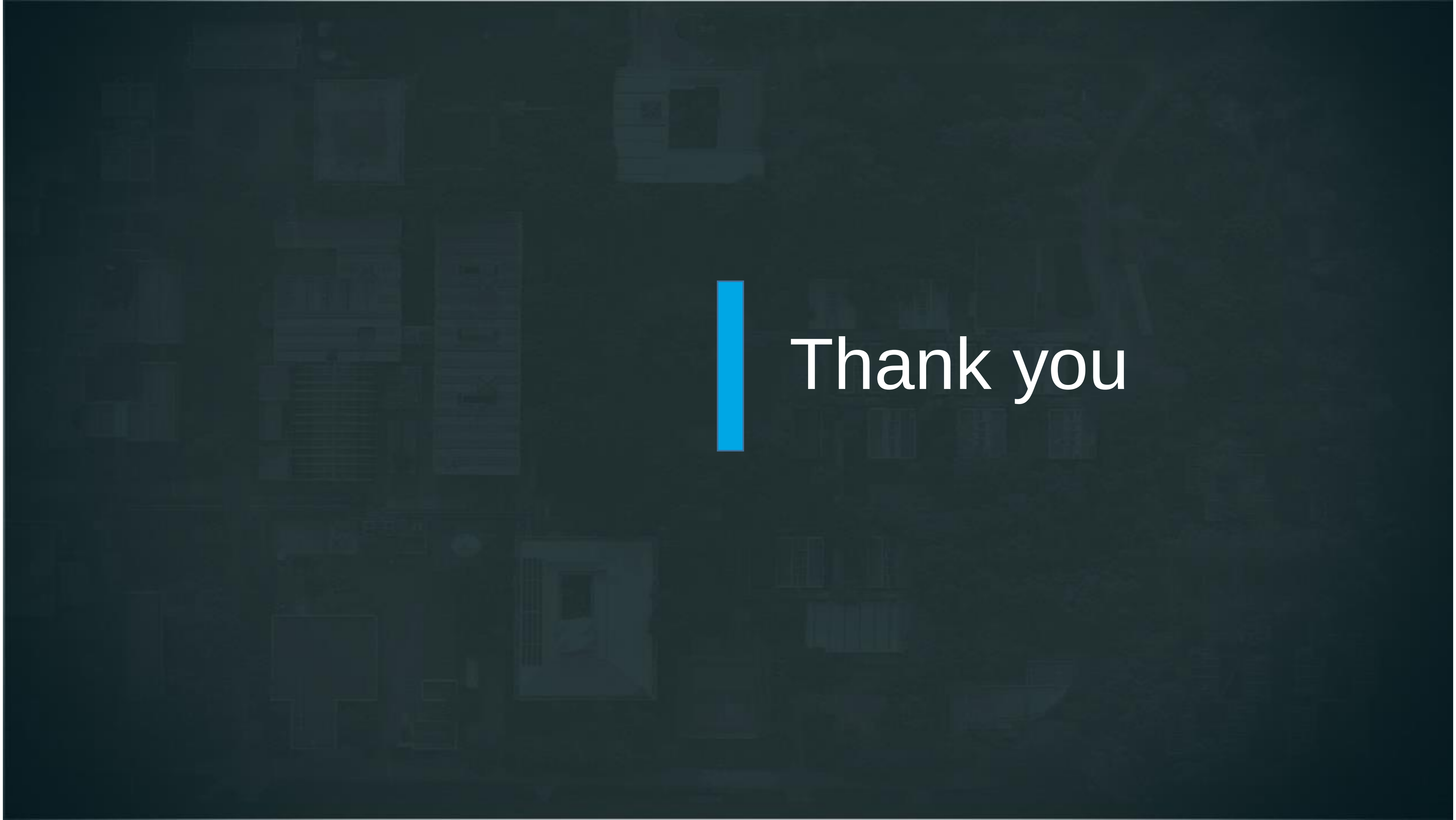
Project Report

Model File: 1, Revision 0
4/16/2020

Concrete Column Summary - ACI 318-14

1 of 192 Reload Apply

Story	Label	Unique Name	Design Section	Station in	Design/Check	Status	PMM Ratio	PMM Combo	As,min in²	As in²	Mid Bar As in²	Corner Bar As in²	V Major Combo	At V in
Story4	C1	1	ConcCol	0	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon4	0.18
Story4	C1	1	ConcCol	72	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon4	0.18
Story4	C1	1	ConcCol	144	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon4	0.18
Story4	C2	2	ConcCol	0	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon6	0.18
Story4	C2	2	ConcCol	72	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon6	0.18
Story4	C2	2	ConcCol	144	Design	No Message		DCon4	3.24	3.7947	0.3162	0.3162	DCon6	0.18
Story4	C3	3	ConcCol	0	Design	No Message		DCon6	3.24	3.24	0.27	0.27	DCon6	0.18



Thank you