

Guest Lecture in Prodi Teknik Sipil

Introduction to Finite Element Method

Wong Foek Tjong, Ph.D.

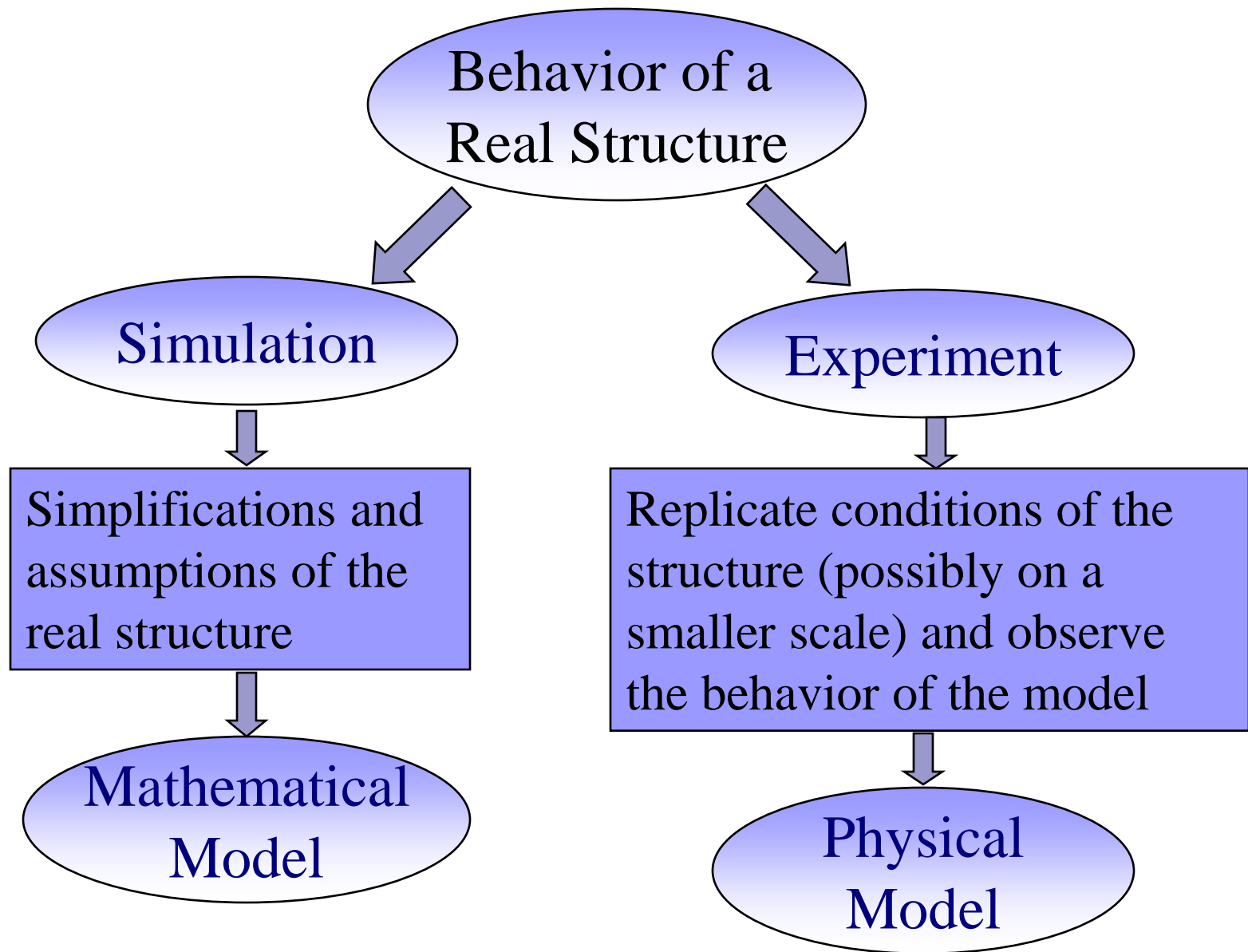
Petra Christian University
Surabaya





Lecture Outline

1. Overview of the FEM
2. Computational steps of the FEM
3. Learning the direct stiffness method using ta29.petra.ac.id



An example of the FEM applications

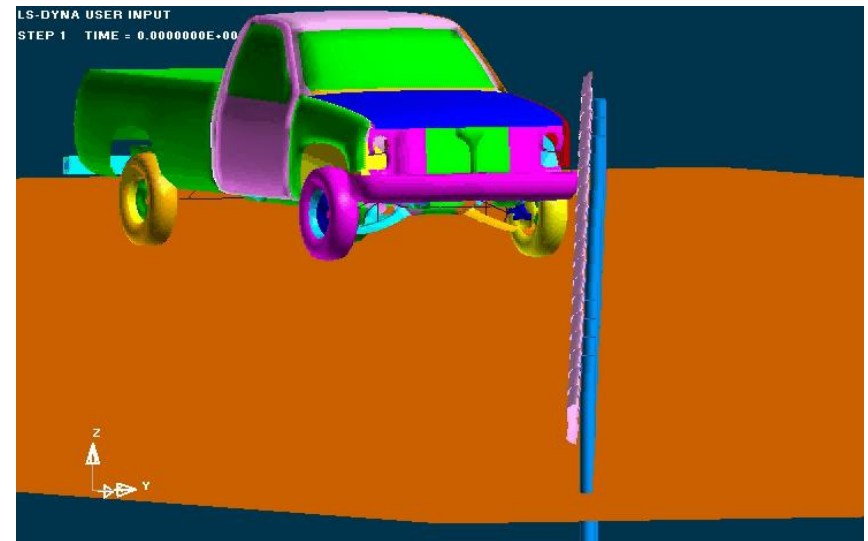
Real experiment

It is often expensive or dangerous

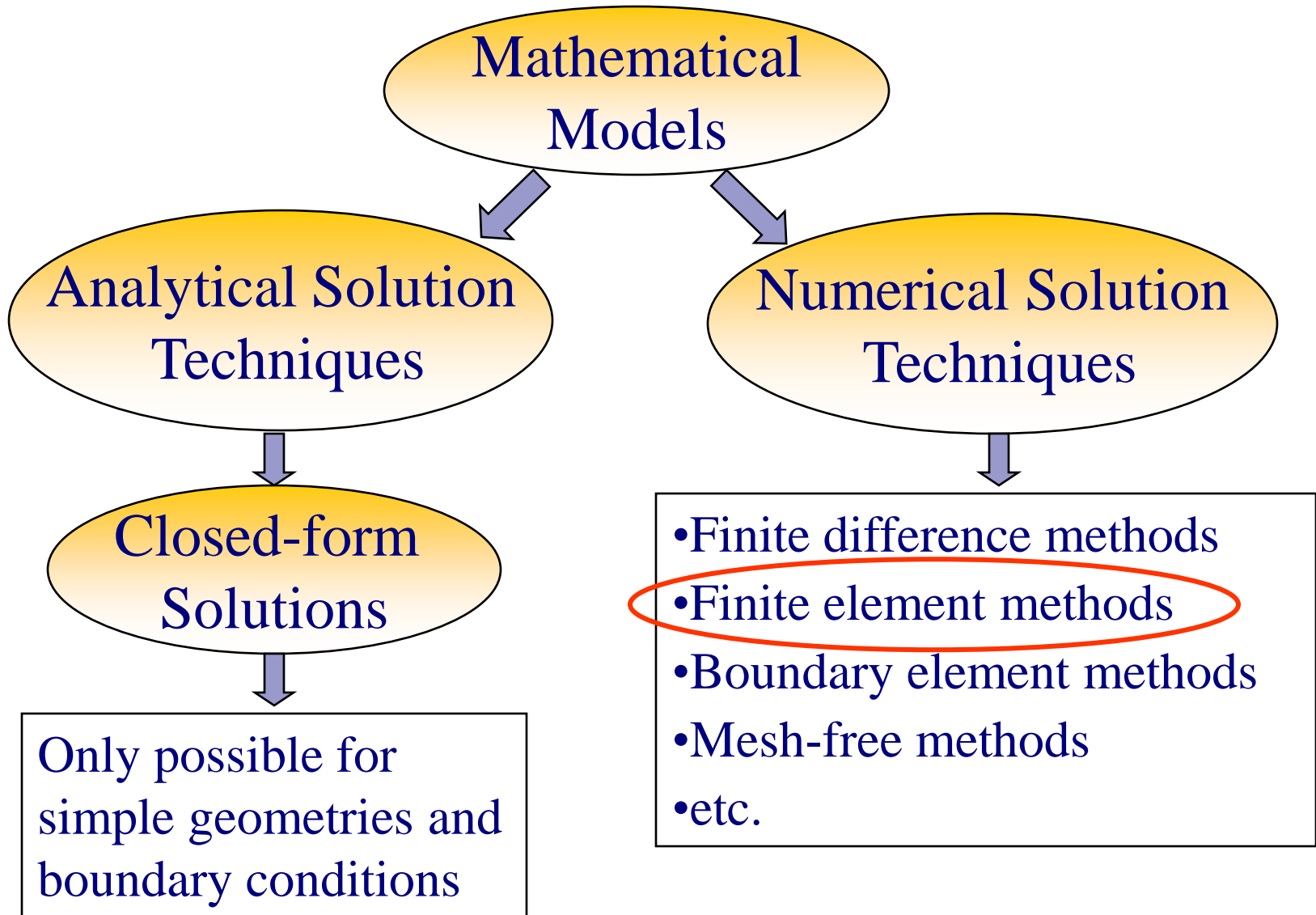


FE simulation

It replicates conditions of the real experiment



Source: W.J. Barry (2003), "FEM Lecture Slides", AIT Thailand

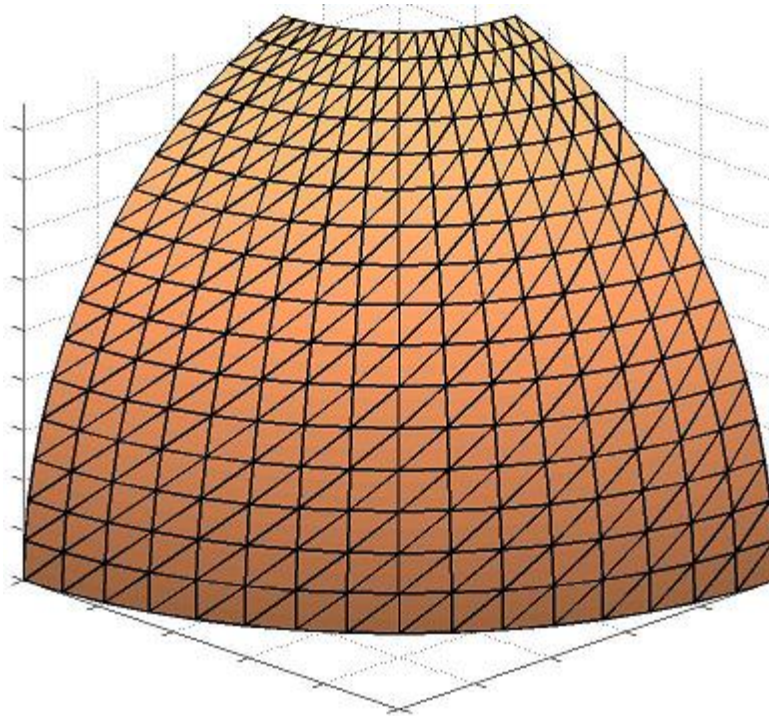


What is FEM?

- It is a computational technique used to obtain **approximate solutions** of engineering problems.
 - The results are generally **not exact**.
 - However, the accuracy of the results can be improved either using finer mesh (**h -refinement**) or higher degree elements (**p -refinement**)

Solution refinements in FEM

h-refinement



$h=1$



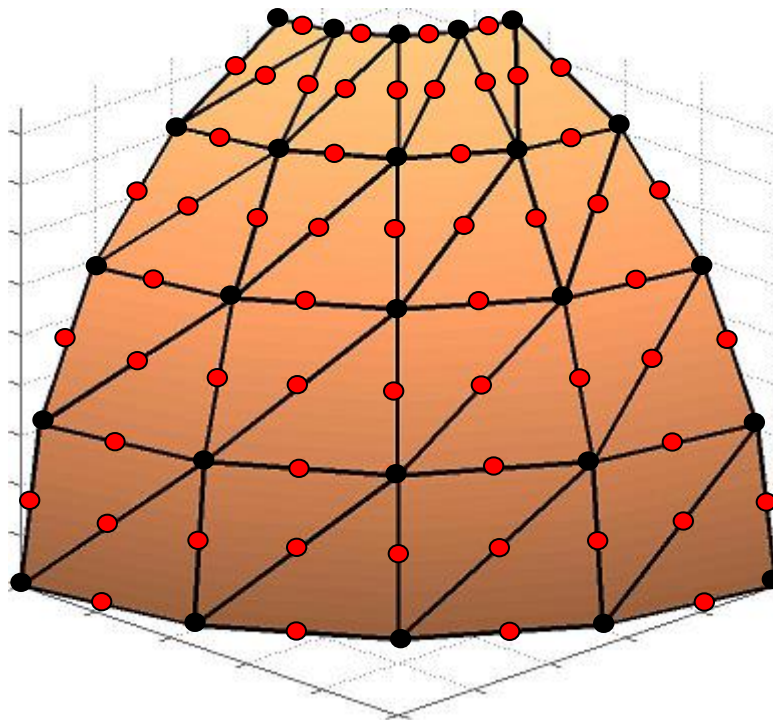
$h=1/2$



$h=1/4$

Solution refinements in FEM (cont'd)

p-refinement



$$u = a + bx + cy$$



$$u = a + bx + cy + dx^2 + exy + fy^2$$

Finite element method (1)

- In the context of structural analyzes, it may be regarded as a **generalized matrix method of structural analysis**.
- It is originated as a method of **structural analysis** but is now widely used in various disciplines such as heat transfer, fluid flow, seepage, electricity and magnetism, and others.

Finite element method (2)

- Modern FEM were first developed and applied by aeronautical engineers, i.e. [M.J. Turner et al.](#), at Boeing company in the period 1950s.
 - 1956: The first *engineering* FEM paper

JOURNAL OF THE AERONAUTICAL SCIENCES

VOLUME 23

SEPTEMBER, 1956

NUMBER 9

Stiffness and Deflection Analysis of Complex Structures

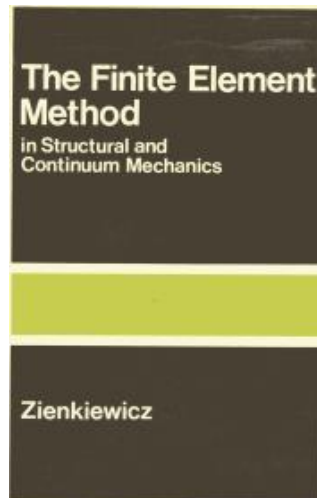
M. J. TURNER,* R. W. CLOUGH,† H. C. MARTIN,‡ AND L. J. TOPP**

Finite element method (3)

- The name “finite element method” was coined by R.W. Clough in 1960. It is called “finite” in order to distinguish with “infinitesimal element” in Calculus.
- 1967: First FEM book by O.C. Zienkiewicz



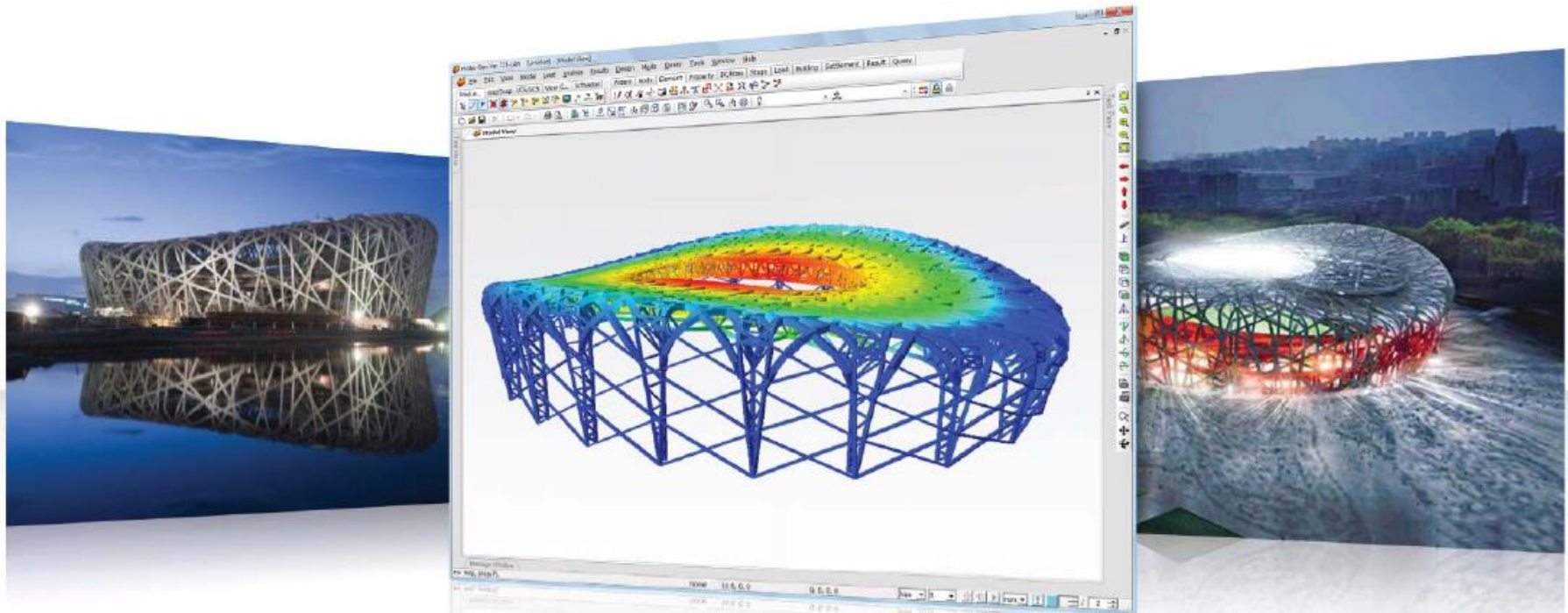
R.W. Clough (1956)



Examples of FEM software

- For General purposes:
NASTRAN, ANSYS, ADINA, ABAQUS, etc.
- For structural analysis, particularly in Civil Engineering:
SANS, SAP2000, STAAD, GT STRUDL, MIDAS, DIANA, STRAND 7, etc.
- For building structures:
ETABS, BATS etc.

Example of applications in structural engineering



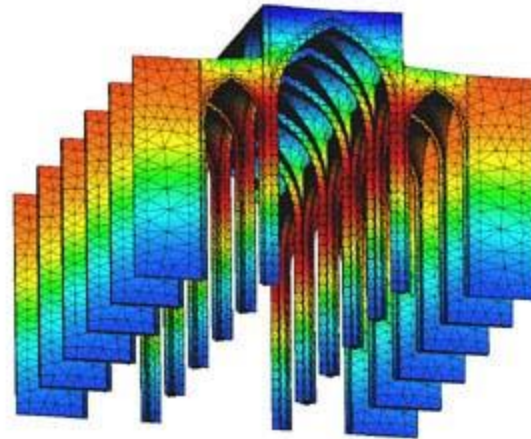
Beijing National Stadium in China
(Bird's Nest Stadium)

Source: MidasGen project applications
<http://en.midasuser.com/products/gallery.asp>

Example of applications in structural engineering (cont'd)



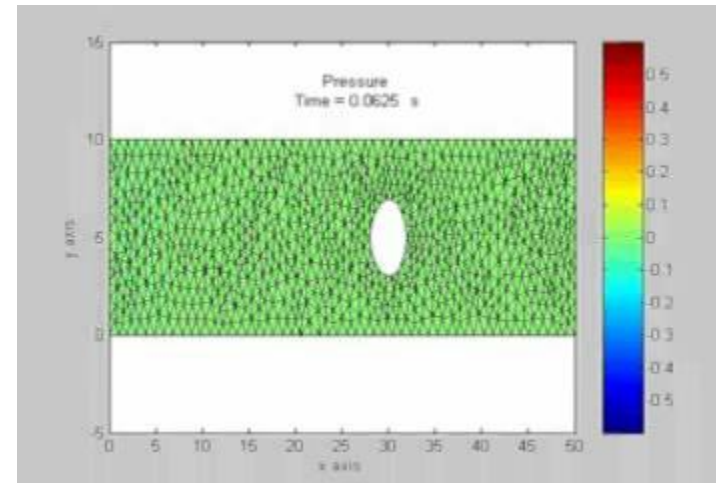
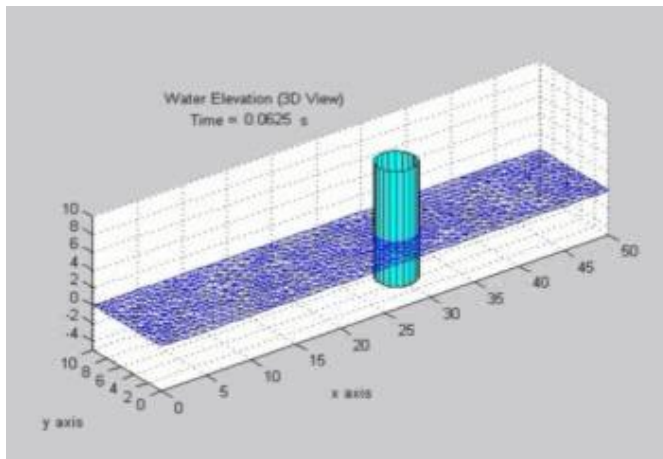
The structural analysis of an F-16 aircraft



The analysis of the Cathedral of Barcelona using 3D solid elements. (courtesy of Barcelona Cathedral)

Source: <http://gid.cimne.upc.es/gidinpractice/gp01.html>

Simulation of Wave Propagation Passing a Solid Barrier



Source: G. B. Wijaya (2005), a Petra-ALT Thesis

Lecture Outline

1. Overview of the FEM
2. Computational steps of the FEM
3. Learning the direct stiffness method using ta29.petra.ac.id

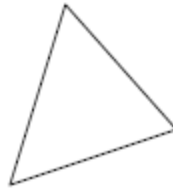
Discretization

- Fundamental concept is discretization, i.e. **dividing** a continuum (continuous body, structural system) into **a finite number of smaller and simple elements** whose union approximates the geometry of the continuum.

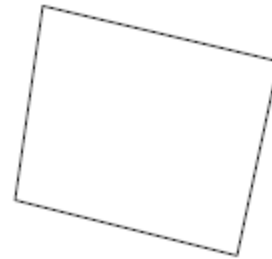
Some basic element shapes



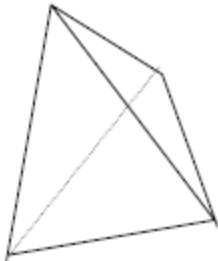
1D Line



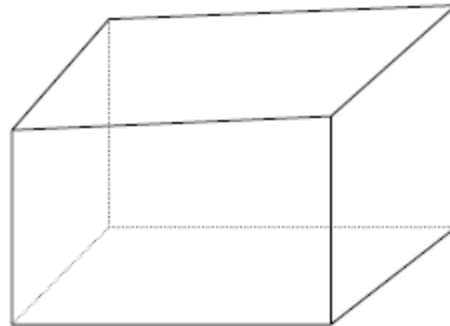
2D Triangle



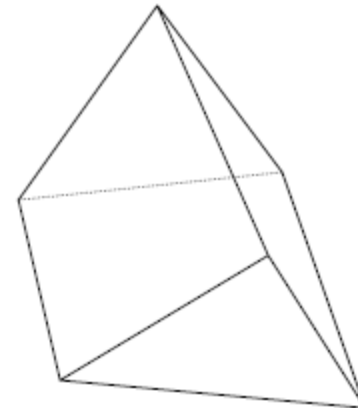
2D Quadrilateral



3D Tetrahedron



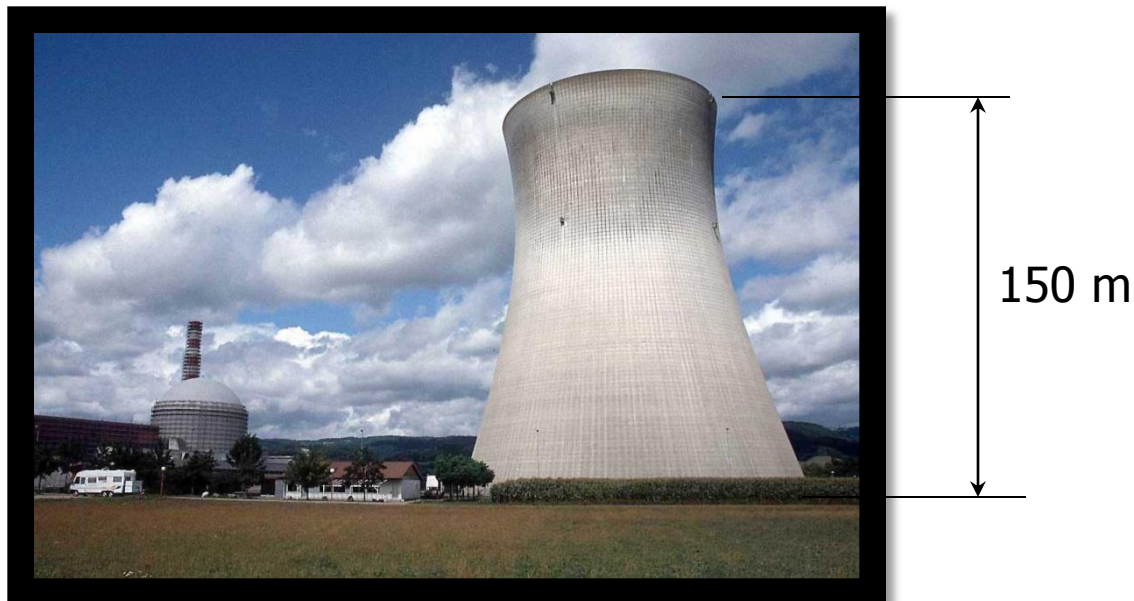
3D Hexahedron



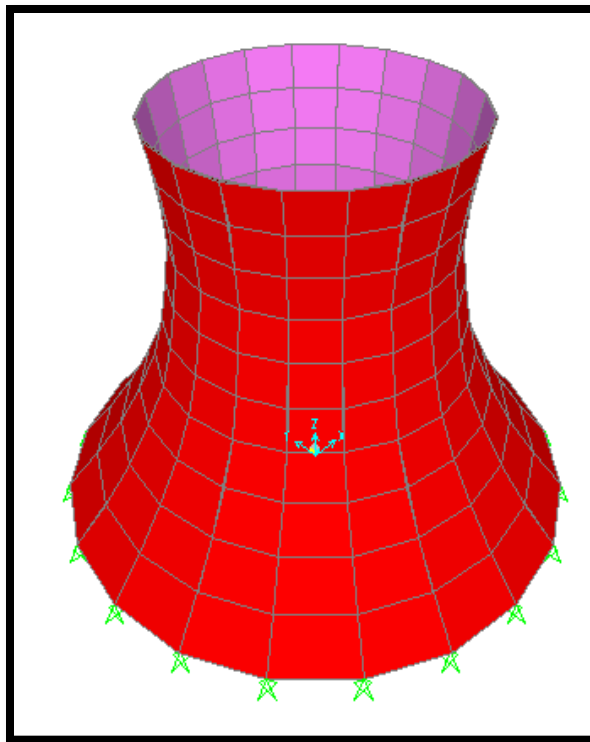
3D Wedge

Examples of discretization (3)

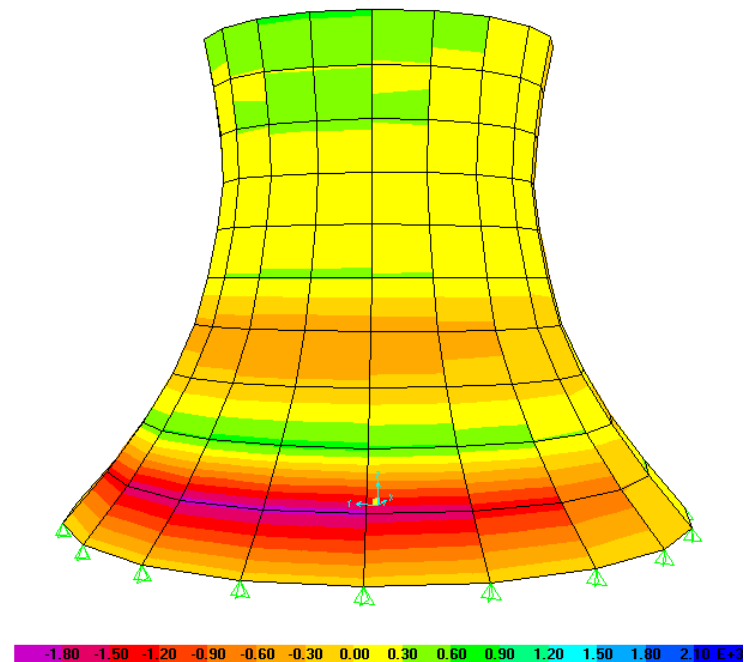
Cooling Tower– Nuclear Power Plant (taken from a FEM Course Project of Doddy and Andre, Dec 2008)



Structural Model and Its Example of the Analysis Results

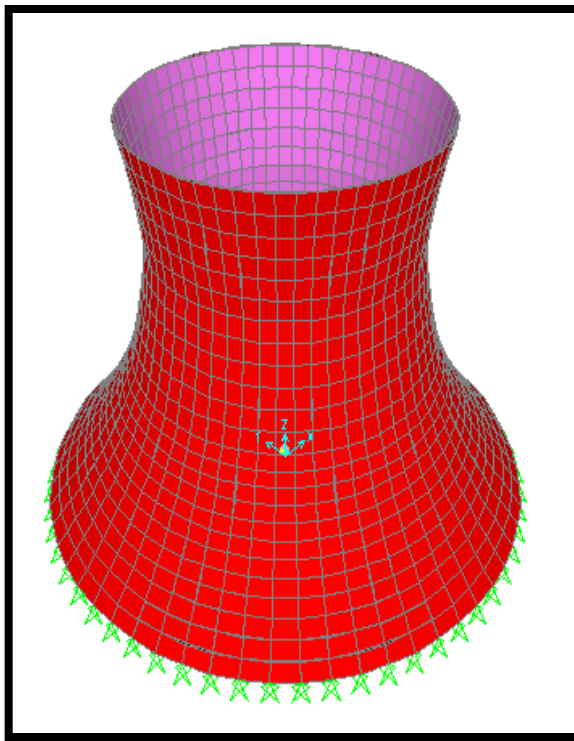


The structure is divided into smaller parts called "element"

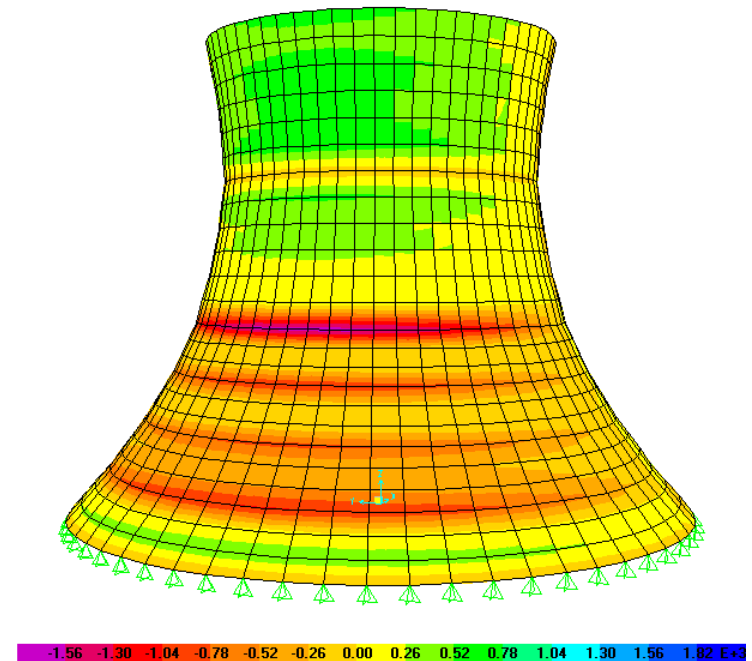


Membrane force contour in the circumferential direction

The FE Model with a Finer Mesh



The structure is modeled with a finer mesh



The result is now better

Computational steps of the FEM- the direct stiffness method

- Discretize the structure (problem domain)
 - Divide the structure or continuum into finite elements
- Once the structure has been discretized, the computational steps faithfully follow the steps in **the direct stiffness method**.
- The direct stiffness method:
 - The global stiffness matrix of the discrete structure are obtained by superimposing (assembling) the stiffness matrices of the element in a direct manner.

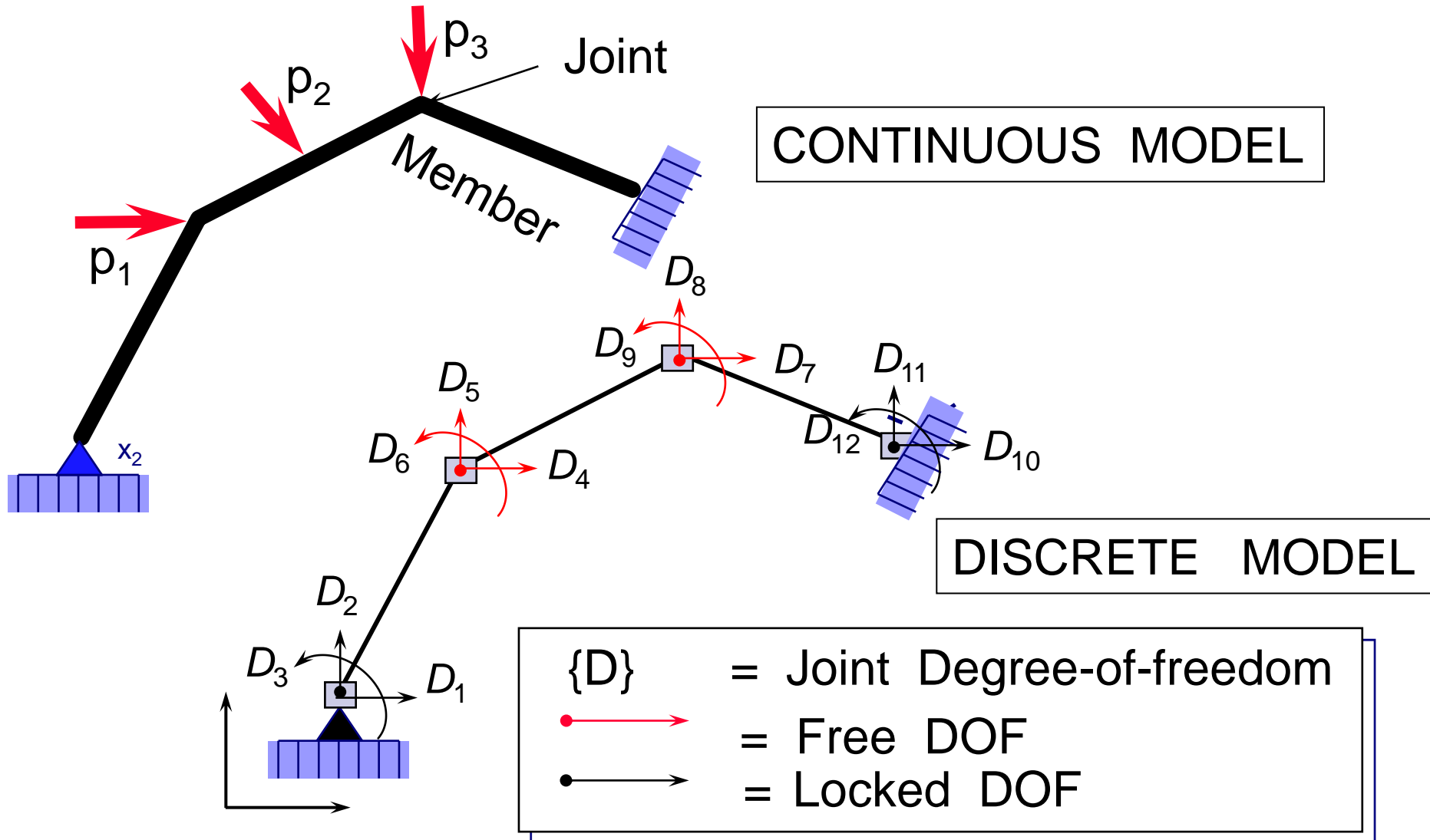
Computational steps... (cont'd)

- Generate **element stiffness matrix** and **element force matrix** for each element.
- **Assemble** the element matrices to obtain the global stiffness equation of the structure.
- Apply the known **nodal loads**.
- Specify how the structure is **supported**:
 - Set several nodal displacements to known values.

General steps of the FEM (cont'd)

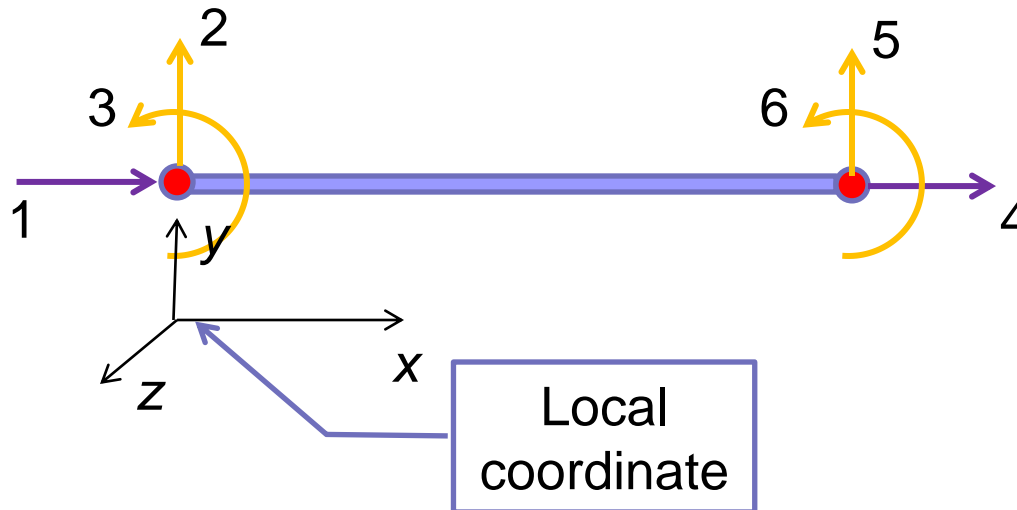
- Solve simultaneous linear algebraic equation.
 - The **nodal parameters** (displacements) are obtained.
- Calculate element stresses or stress resultants (internal forces).

Discretization of Plane Frame Structures



Plane Frame Element

- The stiffness equation for the plane frame element can be obtained by superposing the bar and beam equations.





- The discretized equation:

$$\mathbf{k}\mathbf{d}=\mathbf{f}$$

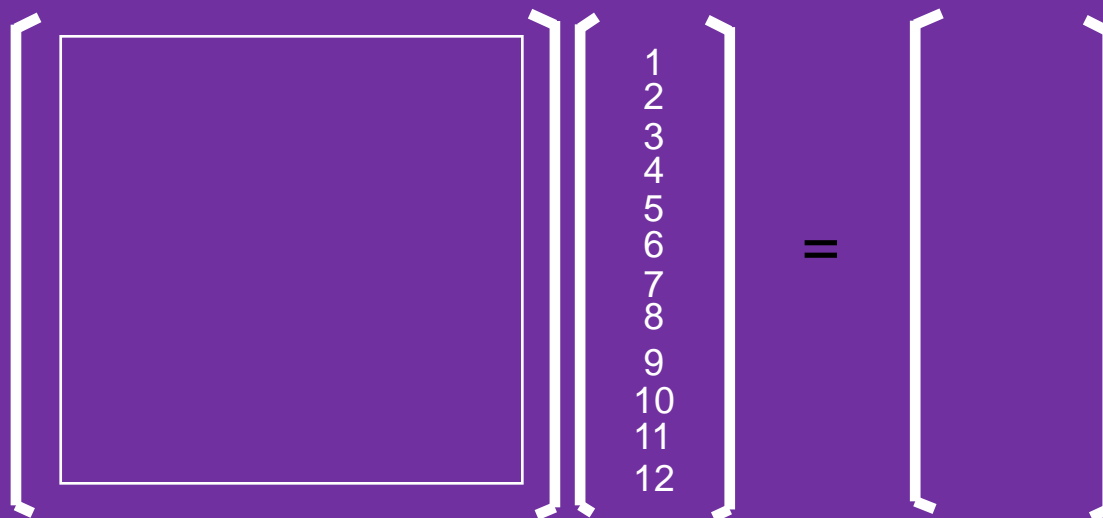
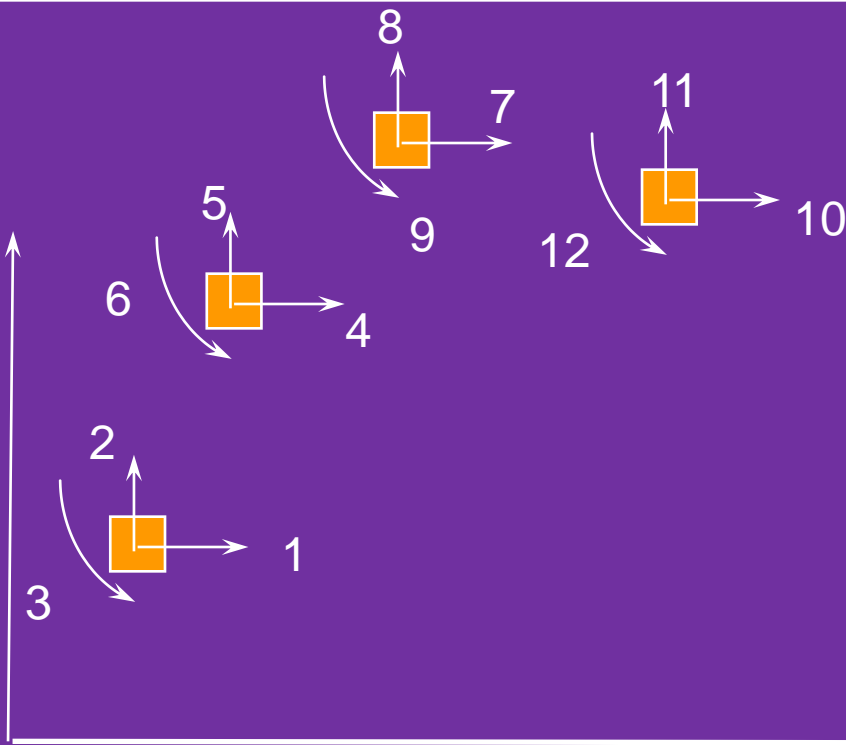
$$\mathbf{d} = \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \end{Bmatrix} = \begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix}; \quad \mathbf{f} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix} = \begin{Bmatrix} N_1 \\ V_1 \\ M_1 \\ N_2 \\ V_2 \\ M_2 \end{Bmatrix}$$

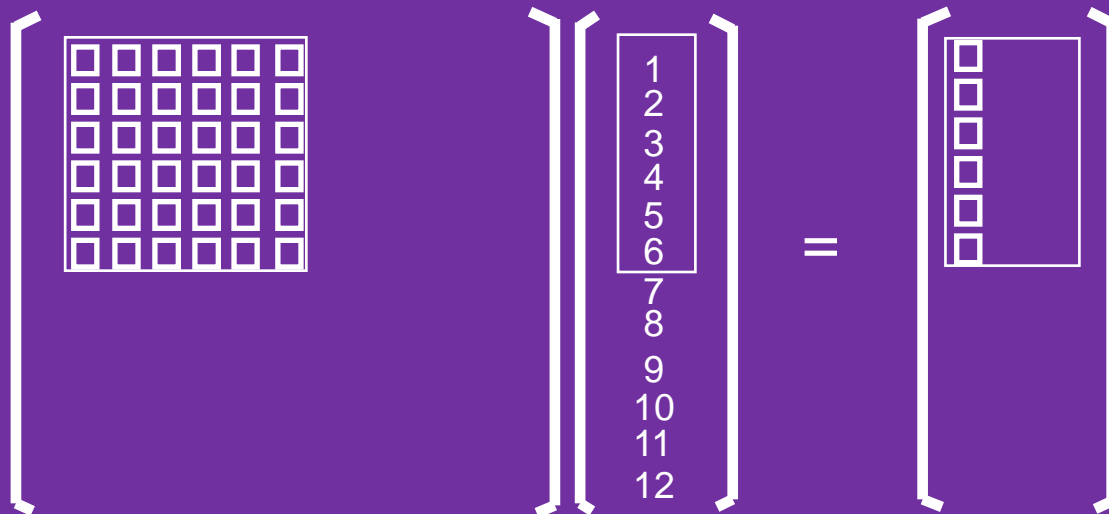
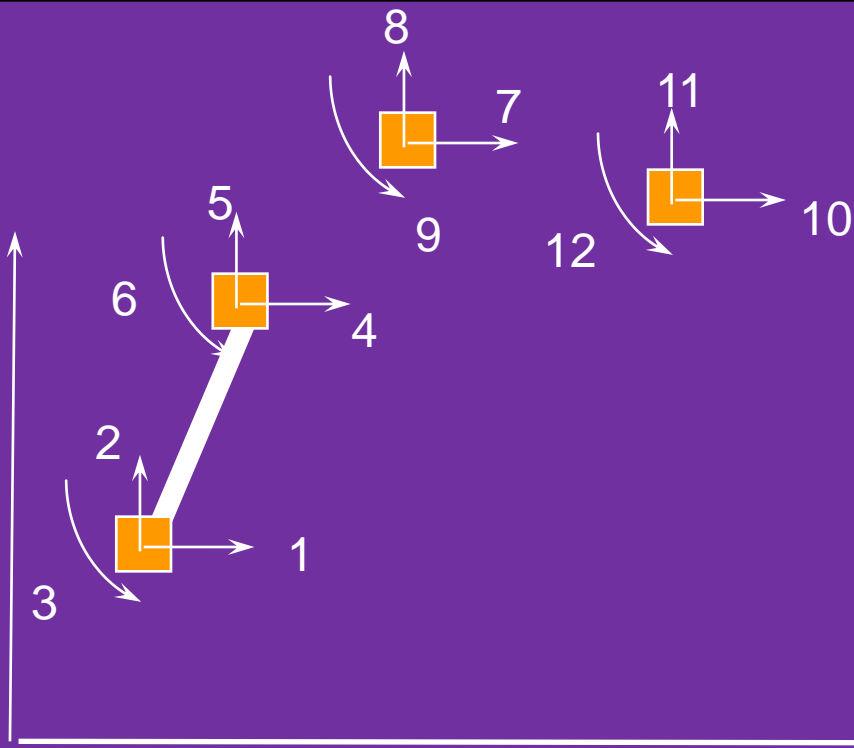


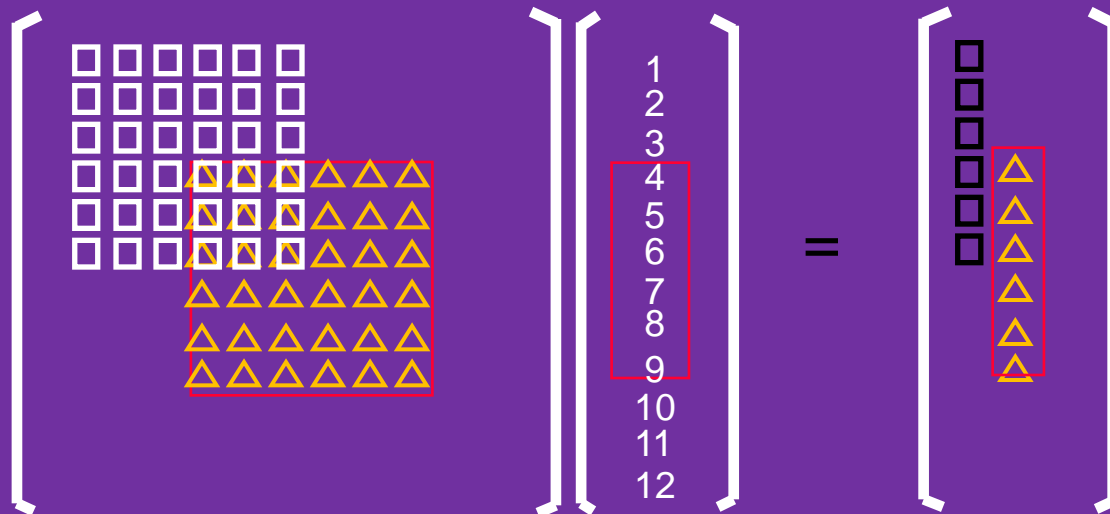
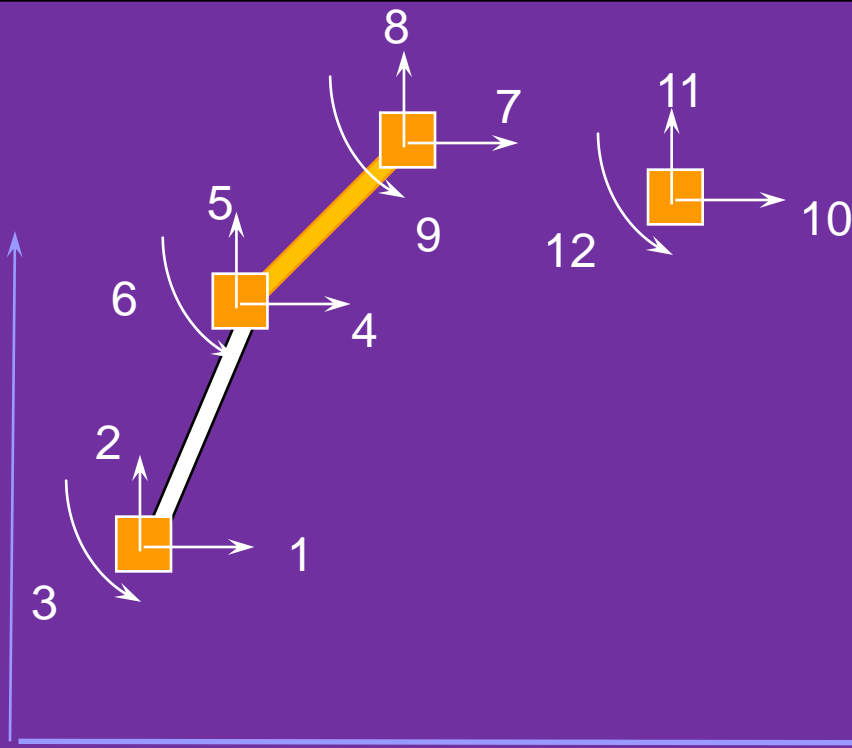
■ The stiffness matrix:

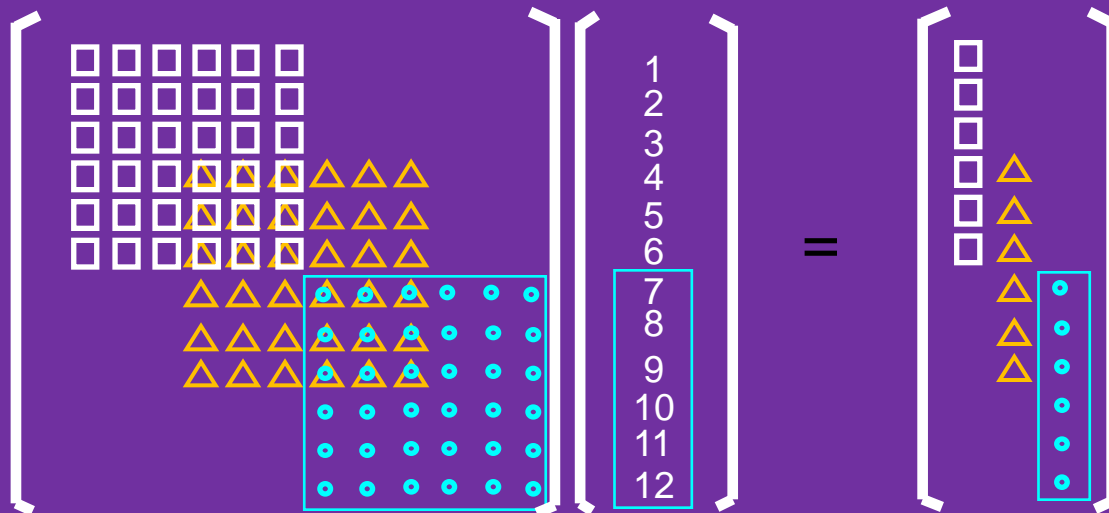
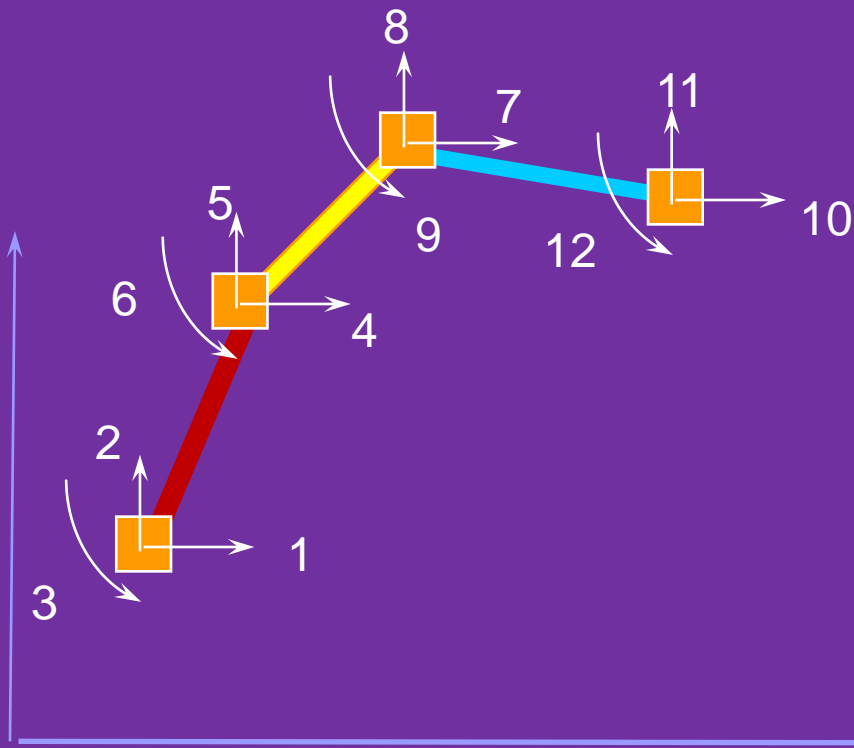
$$\mathbf{k} = \begin{bmatrix} C_1 & 0 & 0 & -C_1 & 0 & 0 \\ 0 & 12C_2 & 6C_2L & 0 & -12C_2 & 6C_2L \\ 0 & 6C_2L & (4 + \varphi)C_2L^2 & 0 & -6C_2L & (2 - \varphi)C_2L^2 \\ -C_1 & 0 & 0 & C_1 & 0 & 0 \\ 0 & -12C_2 & -6C_2L & 0 & 12C_2 & -6C_2L \\ 0 & 6C_2L & (2 - \varphi)C_2L^2 & 0 & -6C_2L & (4 + \varphi)C_2L^2 \end{bmatrix}$$

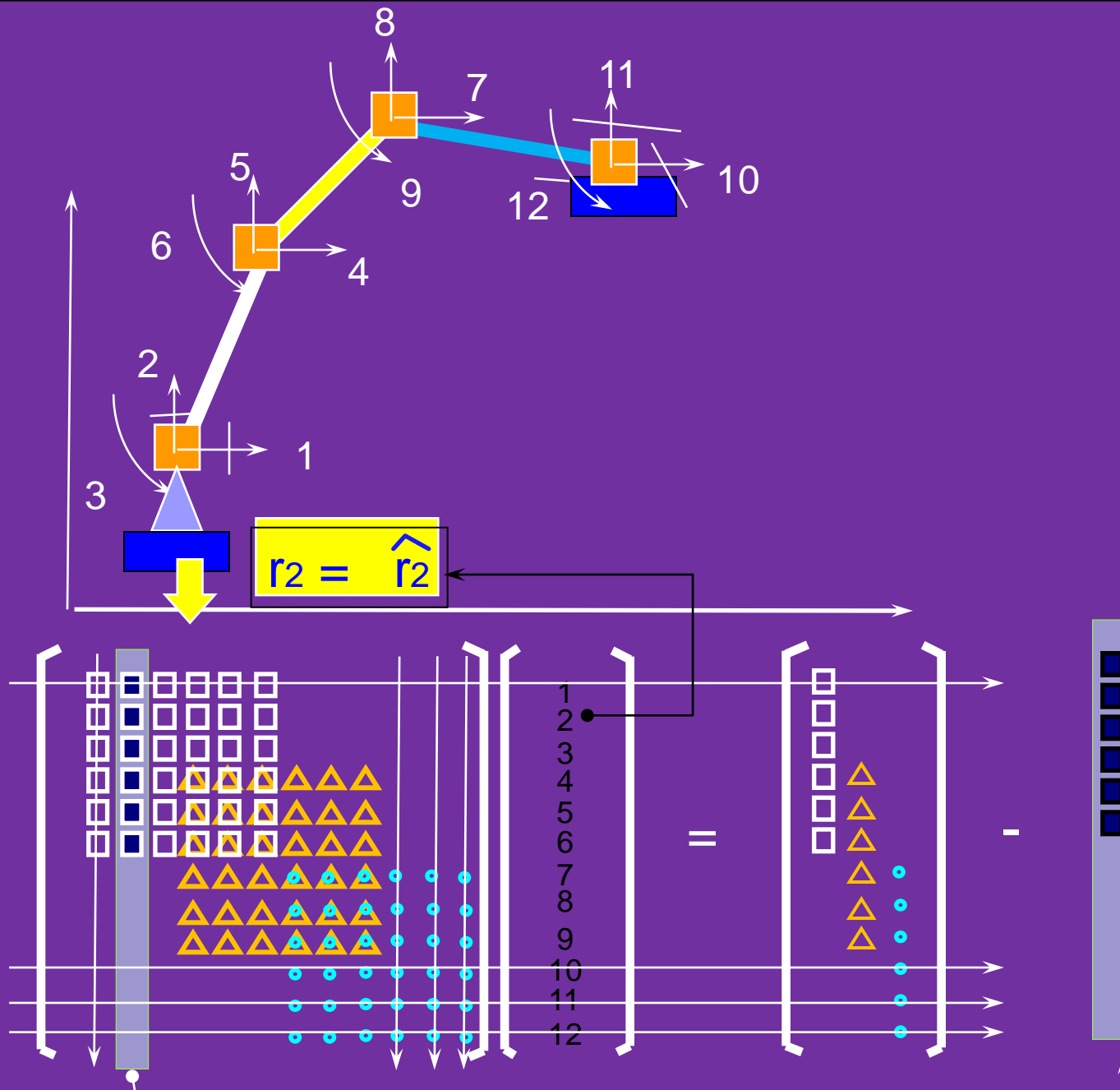
$$C_1 = \frac{EA}{L} ; \quad C_2 = \frac{EI_z}{L^3 (1 + \varphi_y)} ; \quad \varphi_y = \frac{12EI_z}{A_s GL^2}$$

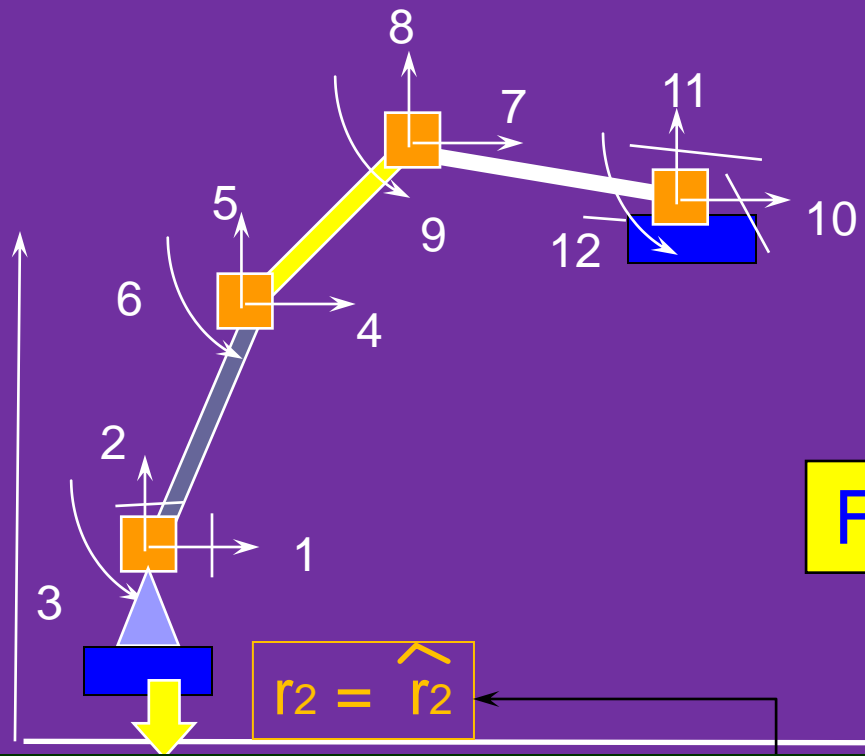




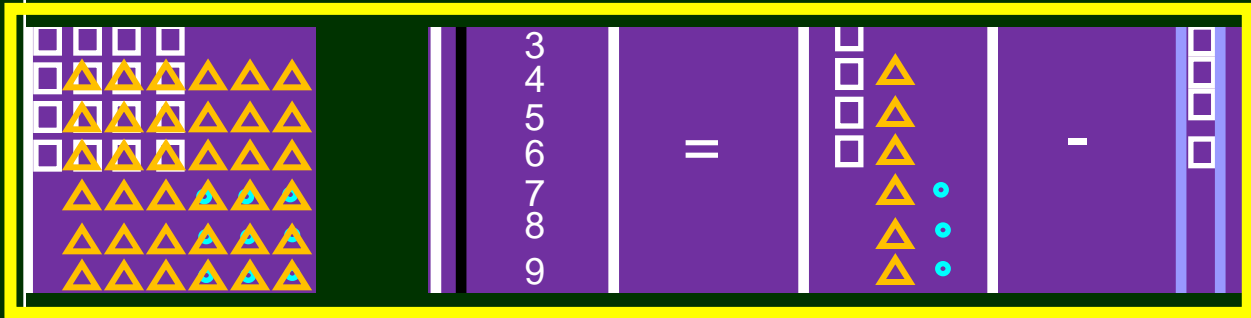








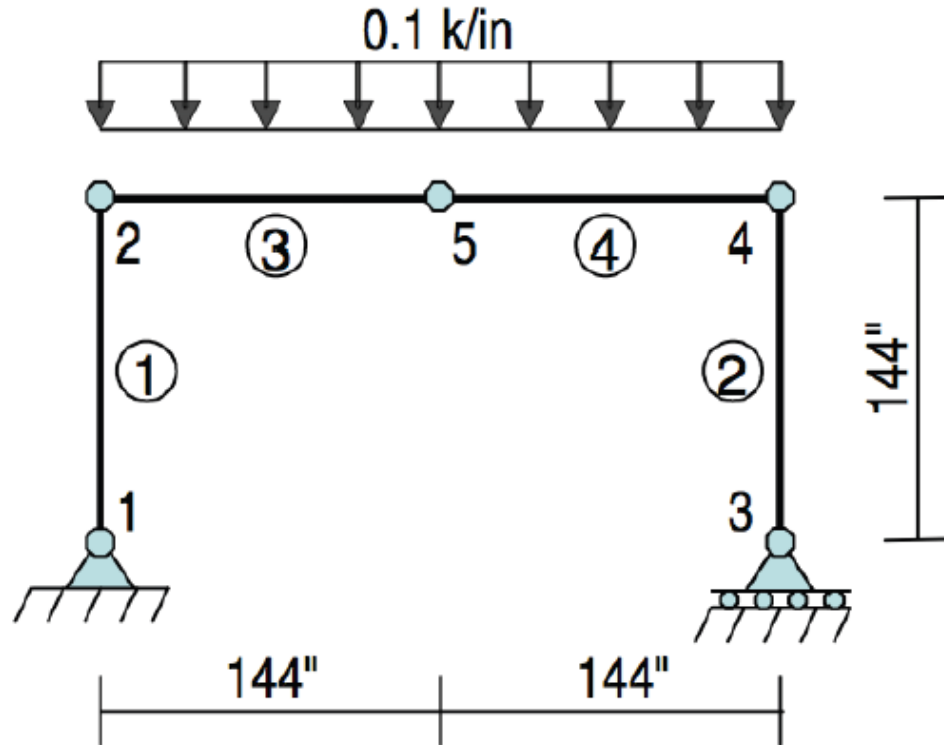
Final System



Lecture Outline

1. Overview of the FEM
2. Computational steps of the FEM
3. Learning the direct stiffness method using ta29.petra.ac.id

Example (SAP2000 Verification Manual, 2007)



Material Properties :

$$E = 29,900 \text{ k/in}^2$$

$$\nu = 0.3$$

$$G = 11,500 \text{ k/in}^2$$

Section Properties :










$$A = 9.12 \text{ in}^2$$

$$I = 110 \text{ in}^4$$

$$A_v = 2.28 \text{ in}^2$$

Find the internal forces, support reactions and the deflection at the middle of the beam (node 5)

- Nodes *(click to expand)*

ID	X Coordinate	Y Coordinate	Boundary Condition		
1	0	0	Hinge	 EDIT	 DELETE
2	0	144	None	 EDIT	 DELETE
3	288	0	Roll1	 EDIT	 DELETE
4	288	144	None	 EDIT	 DELETE
5	144	144	None	 EDIT	 DELETE





Add New Node

- Properties *(click to expand)*

ID	E	ν	A	I	ks		
1	29900	0.3	9.12	110	0.25	 EDIT	 DELETE

Add New Property

- Elements *(click to expand)*

ID	First Node	Second Node	Property		
1	Node #1	Node #2	Property #1	 EDIT	 DELETE
2	Node #2	Node #5	Property #1	 EDIT	 DELETE
3	Node #5	Node #4	Property #1	 EDIT	 DELETE
4	Node #3	Node #4	Property #1	 EDIT	 DELETE

Add New Element

ID	Target Element	Global WX	Global WY	Local WX	Local WY
1	Element #2	0	-0.1	0	0
2	Element #3	0	-0.1	0	0



EDIT



DELETE



EDIT

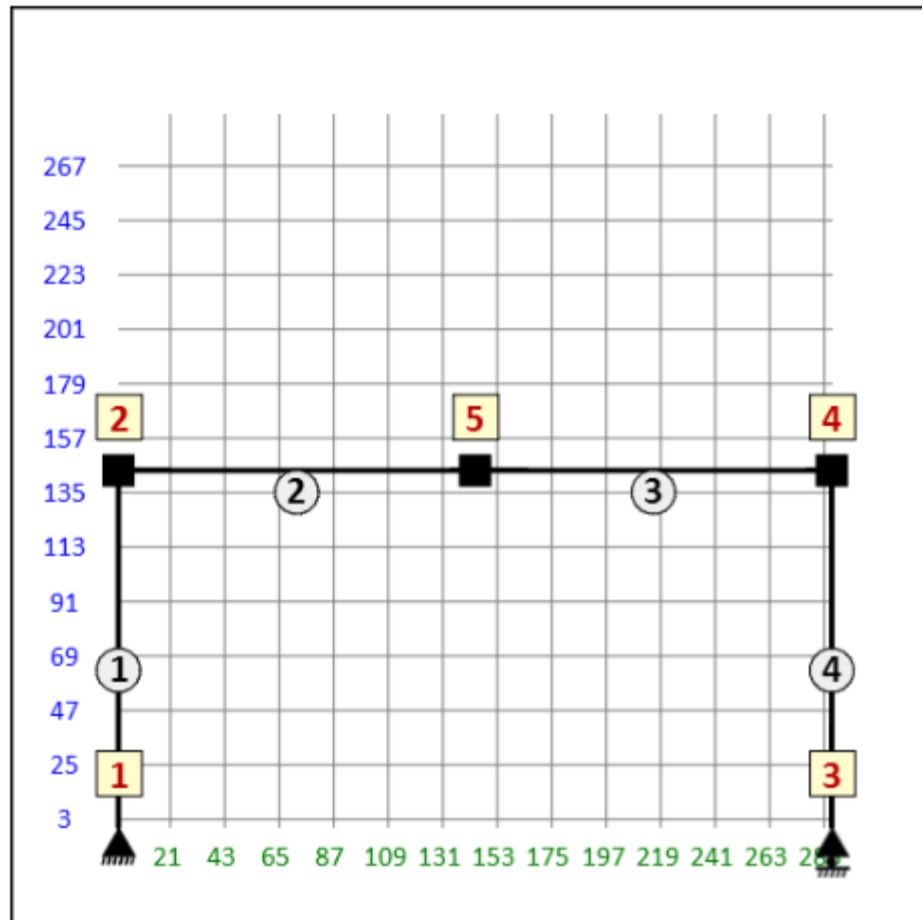


DELETE

Add New Distributed Load

+ Prescribed Displacements *(click to expand)*

deformed Shape



k^A Matrix :

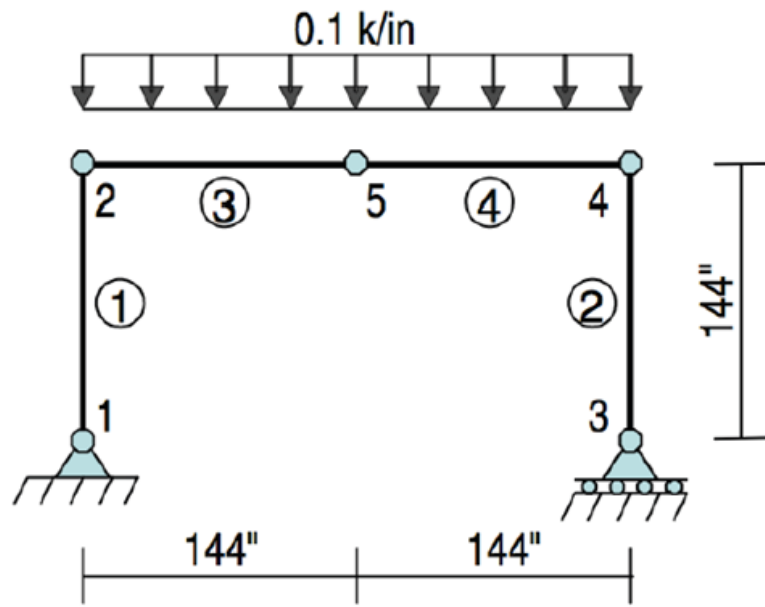
	7	8	9	10	11	12
7	1893.66667	0	0	-1893.66667	0	0
8	0	12.32319	887.26975	0	-12.32319	887.26975
9	0	887.26975	86723.69977	0	-887.26975	41043.14422
10	-1893.66667	0	0	1893.66667	0	0
11	0	-12.32319	-887.26975	0	12.32319	-887.26975
12	0	887.26975	41043.14422	0	-887.26975	86723.69977

T Matrix :

	7	8	9	10	11	12
7	0	1	0	0	0	0
8	-1	0	0	0	0	0
9	0	0	1	0	0	0
10	0	0	0	0	1	0
11	0	0	0	-1	0	0
12	0	0	0	0	0	1

k Matrix :

	7	8	9	10	11	12
7	12.32319	0	-887.26975	-12.32319	0	-887.26975
8	0	1893.66667	0	0	-1893.66667	0
9	-887.26975	0	86723.69977	887.26975	0	41043.14422
10	-12.32319	0	887.26975	12.32319	0	887.26975
11	0	-1893.66667	0	0	1893.66667	0
12	-887.26975	0	41043.14422	887.26975	0	86723.69977



- Solution Phase

F (solved)

F1x	=	0
F1y	=	14.4
M1	=	0
F2x	=	0
F2y	=	-7.2
M2	=	-172.8
F3x	=	0
F3y	=	14.4
M3	=	0
F4x	=	0
F4y	=	-7.2
M4	=	172.8
F5x	=	0
F5y	=	-14.4
M5	=	0

d (solved)

d1x	=	0
d1y	=	0
Ø1	=	-0.03026
d2x	=	4.35778
d2y	=	-0.0076
Ø2	=	-0.03026
d3x	=	8.71555
d3y	=	0
Ø3	=	0.03026
d4x	=	4.35778
d4y	=	-0.0076
Ø4	=	0.03026
d5x	=	4.35778
d5y	=	-2.77076
Ø5	=	0

+ Shear Force Diagrams

- Bending Moment Diagrams

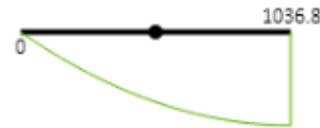
Scale for Bending Moment Diagrams

Set Scale

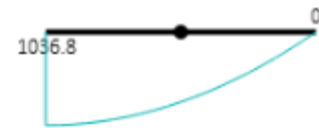
Element #1



Element #2



Element #3



Element #4



- Shear Force Diagrams

Scale for Shear Force Diagrams

3

Set Scale

Element #1



Element #2



Element #3



Element #4



- Axial Force Diagrams

Scale for Axial Force Diagrams

3 [Set Scale](#)

Element #1



Element #2



Element #3



Element #4



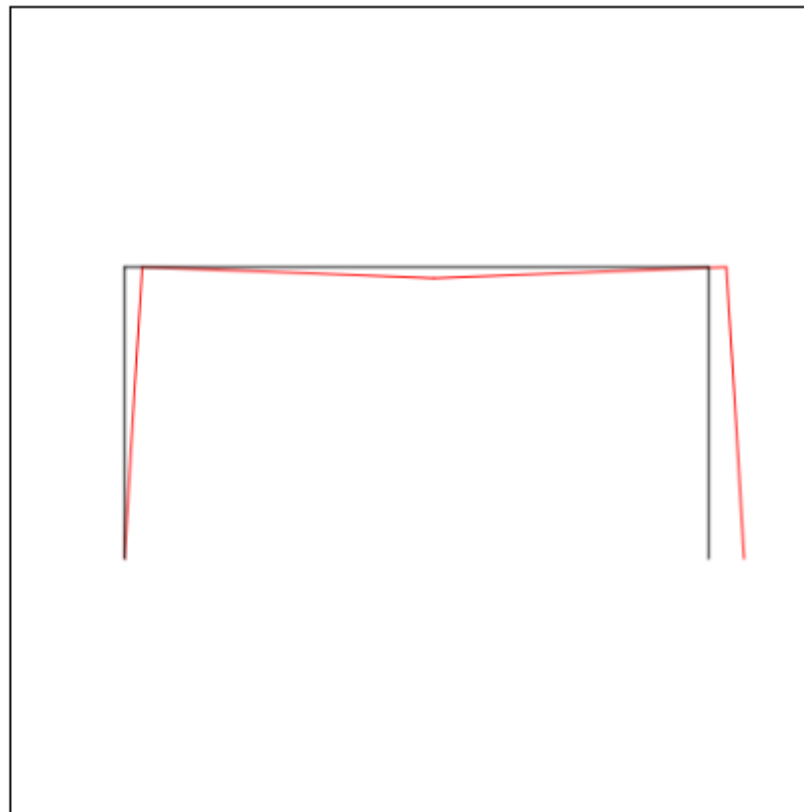
- Deformed shape

Scale for Deformed Shape

2

Set Scale

#	x	y	dx	dy	x'	y'
Node #1	0	0	0	0	0	0
Node #2	0	144	4.35778	-0.0076	4.35778	143.9924
Node #3	288	0	8.71555	0	296.71555	0
Node #4	288	144	4.35778	-0.0076	292.35778	143.9924
Node #5	144	144	4.35778	-2.77076	148.35778	141.22924





Thank you for
your attention.

*Let take the challenge
of the future!*