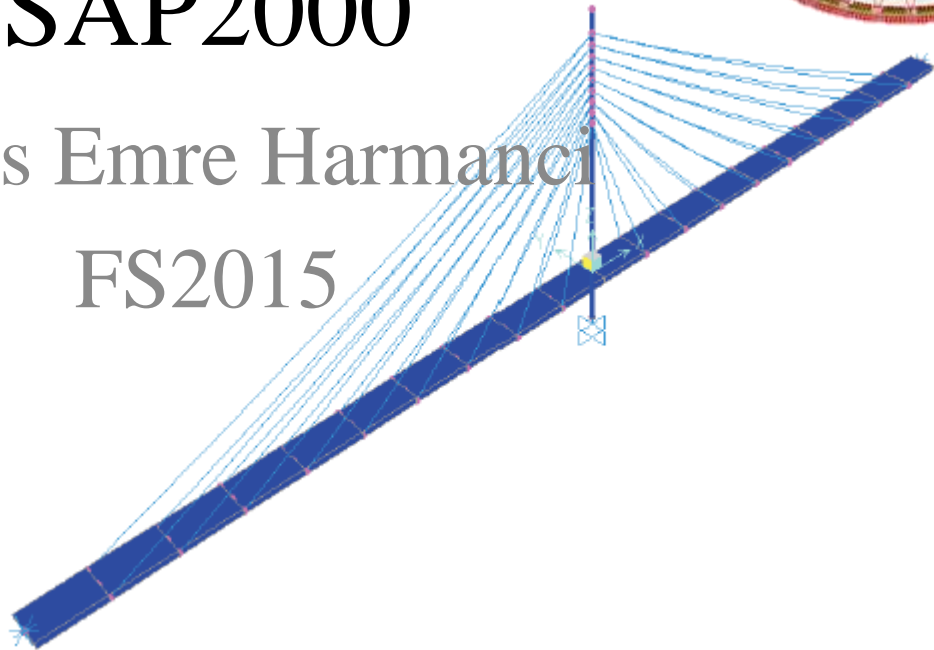
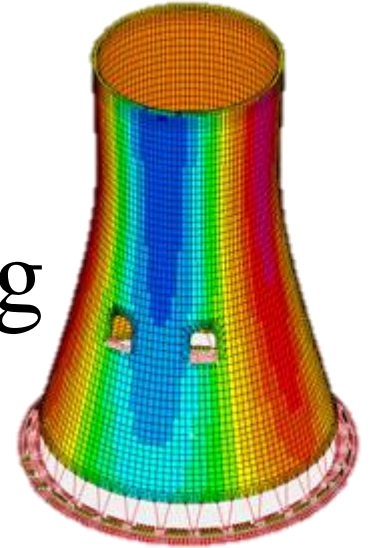


Introduction to Modelling with SAP2000

Yunus Emre Harmanci

FS2015

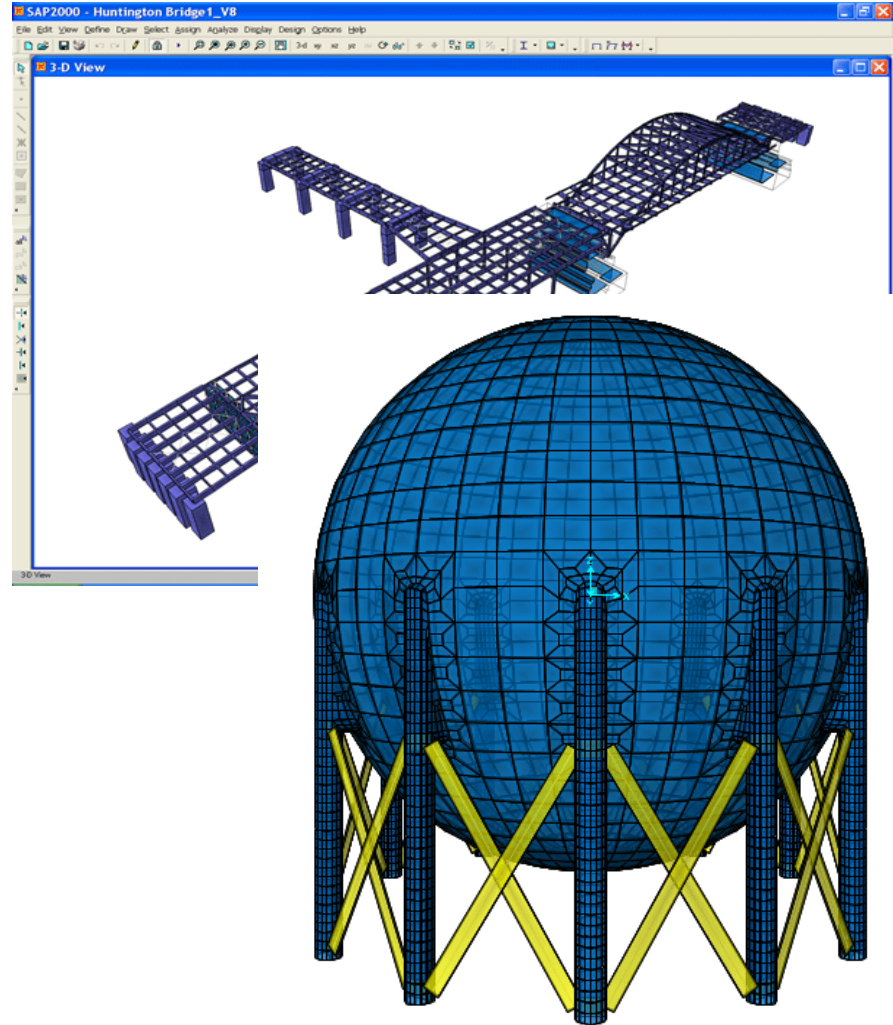


Overview

- Key Features of SAP2000
- Modelling of Elements
- Analysis Capabilities
- Viewing Results
- Tutorial Example: Static and Dynamic Analysis of a 3D Truss Frame
- Further Reading/Tutorials
- Q&A

Key Features

- Powerful and Integrated Structural Analysis and Design Software
- Fully Interactive Graphical Interface for quickly creating models
- Frame, Shell, Solid and Non-linear Link Elements

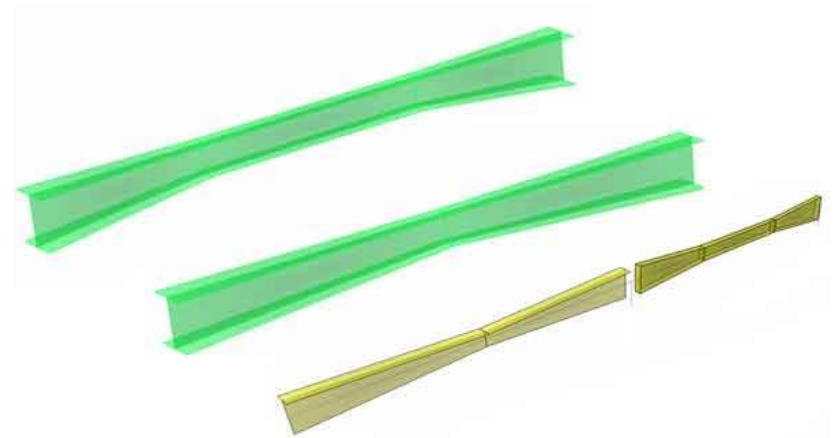
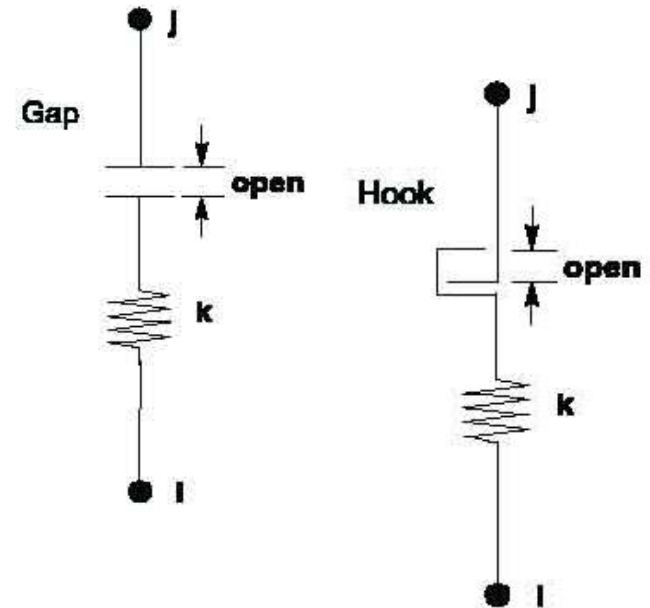


Key Features

- Extensive Analysis Options including Linear-Static, Linear-Dynamic, Nonlinear-Dynamic
- Automated design of Concrete and Steel Members to various International Codes
- Fully formatted, customized reporting
- DXF Import and Export Link to AutoCAD

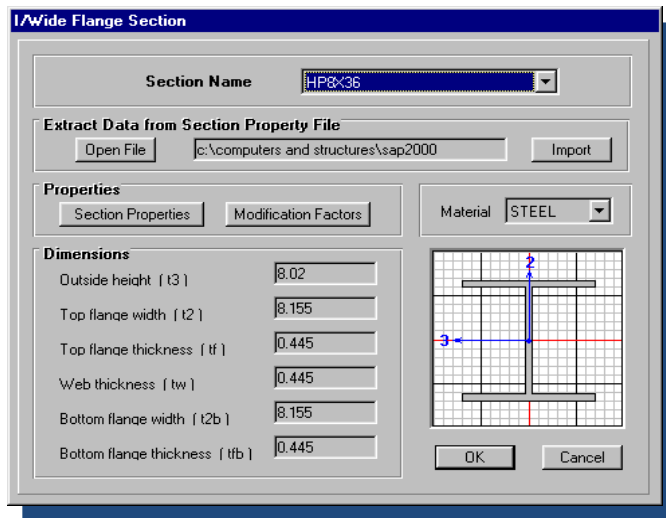
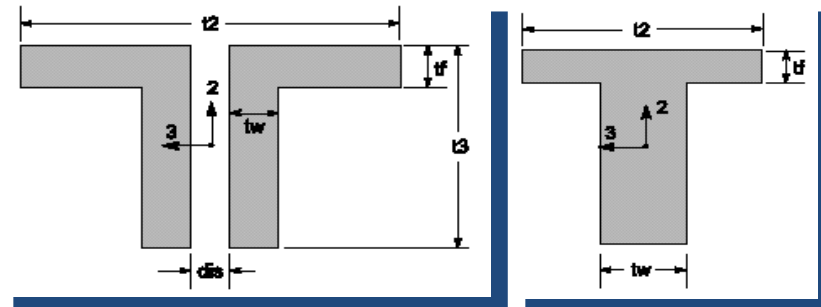
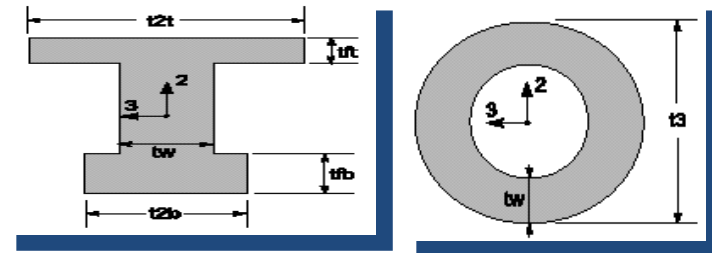
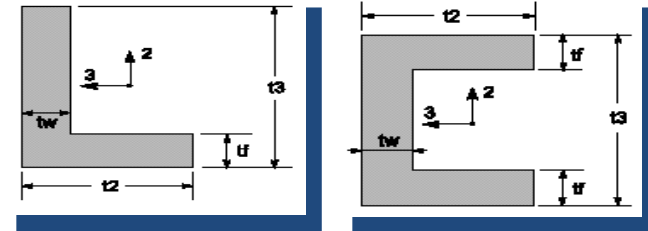
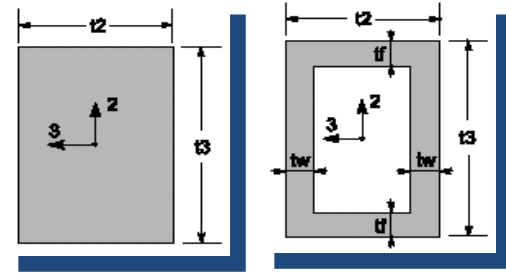
Modelling of Elements

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



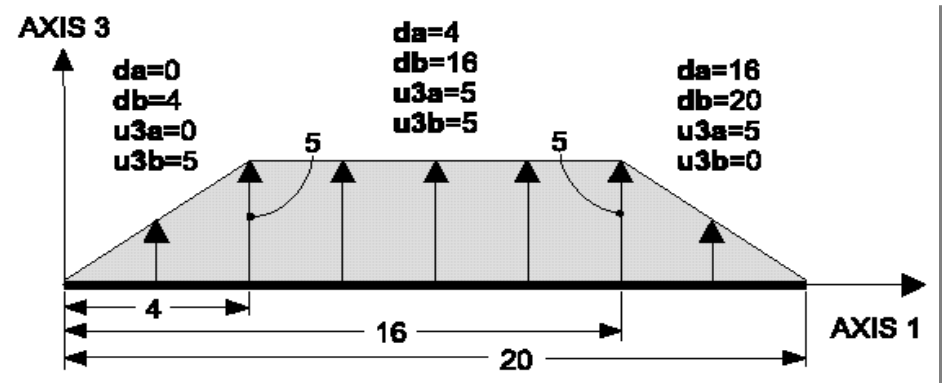
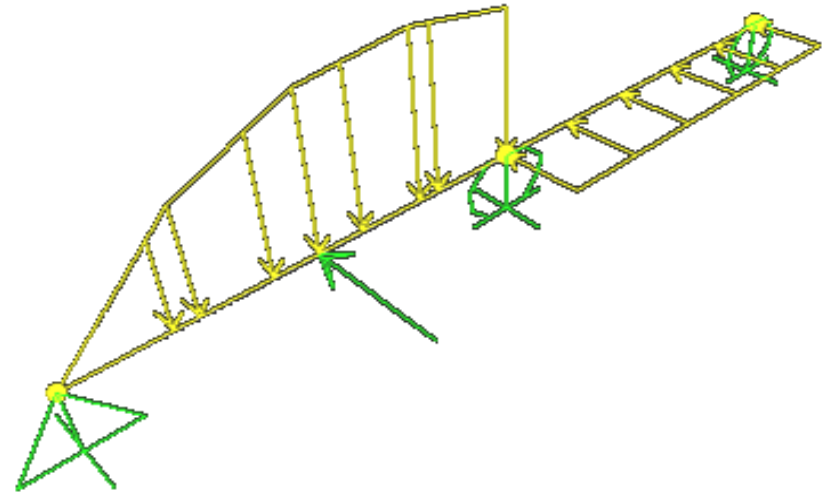
Modelling of Elements

- Automatic section property calculation for standard shapes
- Built-in steel sections for several Standards including AISC, CISC, EN, BS etc.



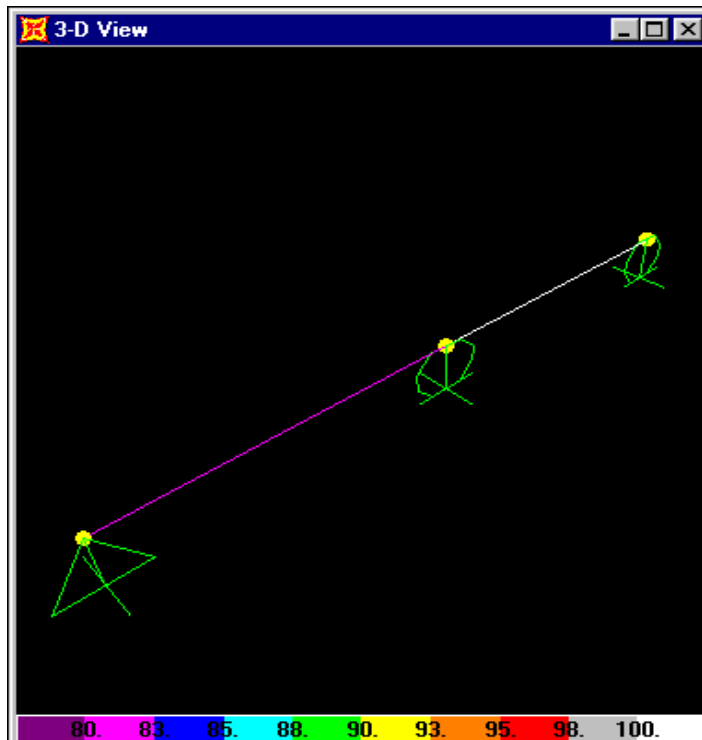
Modelling of Elements

- Gravity Load
- Point Load
- Uniform Load
- Trapezoidal Load
- Prestress
- Temperature Variation



Modelling of Elements

- Applied to Element Section in any direction
- Applied to Nodes and Groups



The figure shows a dialog box titled "Frame Temperature Loading". It has a "Load Case Name" dropdown menu set to "TEMP". Under the "Type" section, "Temperature" is selected with a radio button. Under the "Options" section, "Add to existing loads" is selected with a radio button. Under the "Temperature" section, "By Element" is selected with a radio button, and the "Temperature" field contains the value "100.". There are also fields for "Pattern" and "Multiplier" which are currently empty. "OK" and "Cancel" buttons are at the bottom right.

Modelling of Elements

- Load due to Prestress can be applied to Frame Elements as Cables in Patterns and Load Cases

The diagram illustrates the modeling of a frame element with a prestressing cable. On the left, a vertical frame element is shown with a horizontal axis labeled **AXIS 1** and a vertical axis labeled **AXIS 2**. The cable starts at **End I** at the bottom of the element. The vertical distance from the centerline to the cable at the start is labeled **dl**. The cable curves to the right, and its vertical distance from the centerline at the end is labeled **dc**. The centerline is labeled **Center**. A tension force **t** is shown at the end of the cable. On the right, a software dialog box titled **Frame Prestressing Patterns** is shown. It contains the following fields and options:

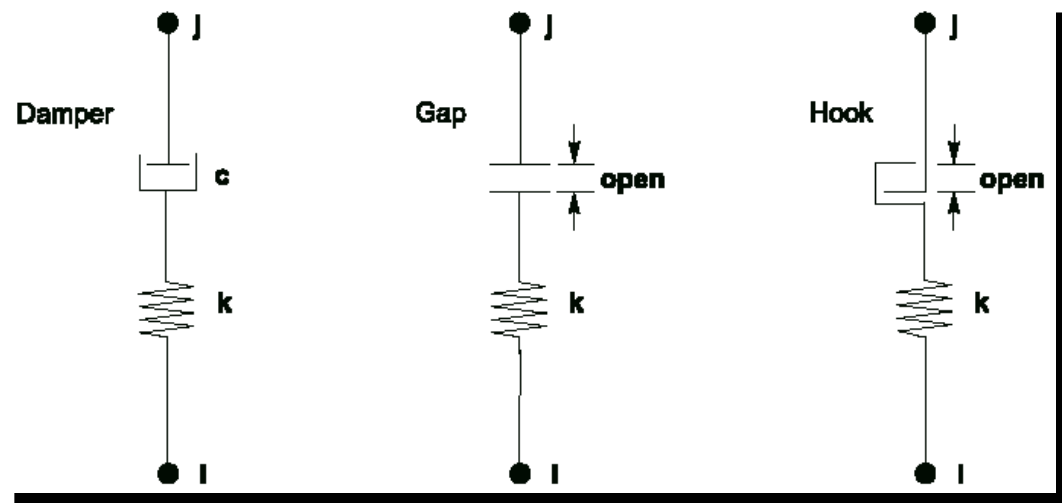
- Cable Tension**: 200.
- Cable Eccentricities**:
 - Start (+2 direction): 8.
 - Middle (-2 direction): 12.
 - End (+2 direction): 3.
- Options**:
 - Add to existing pattern
 - Replace existing pattern
 - Delete existing pattern

A second diagram on the right shows a cable with tension **t** and a vertical distance **l** from the centerline to the cable, with **AXIS 1** and **AXIS 2** axes.

Modelling of Elements

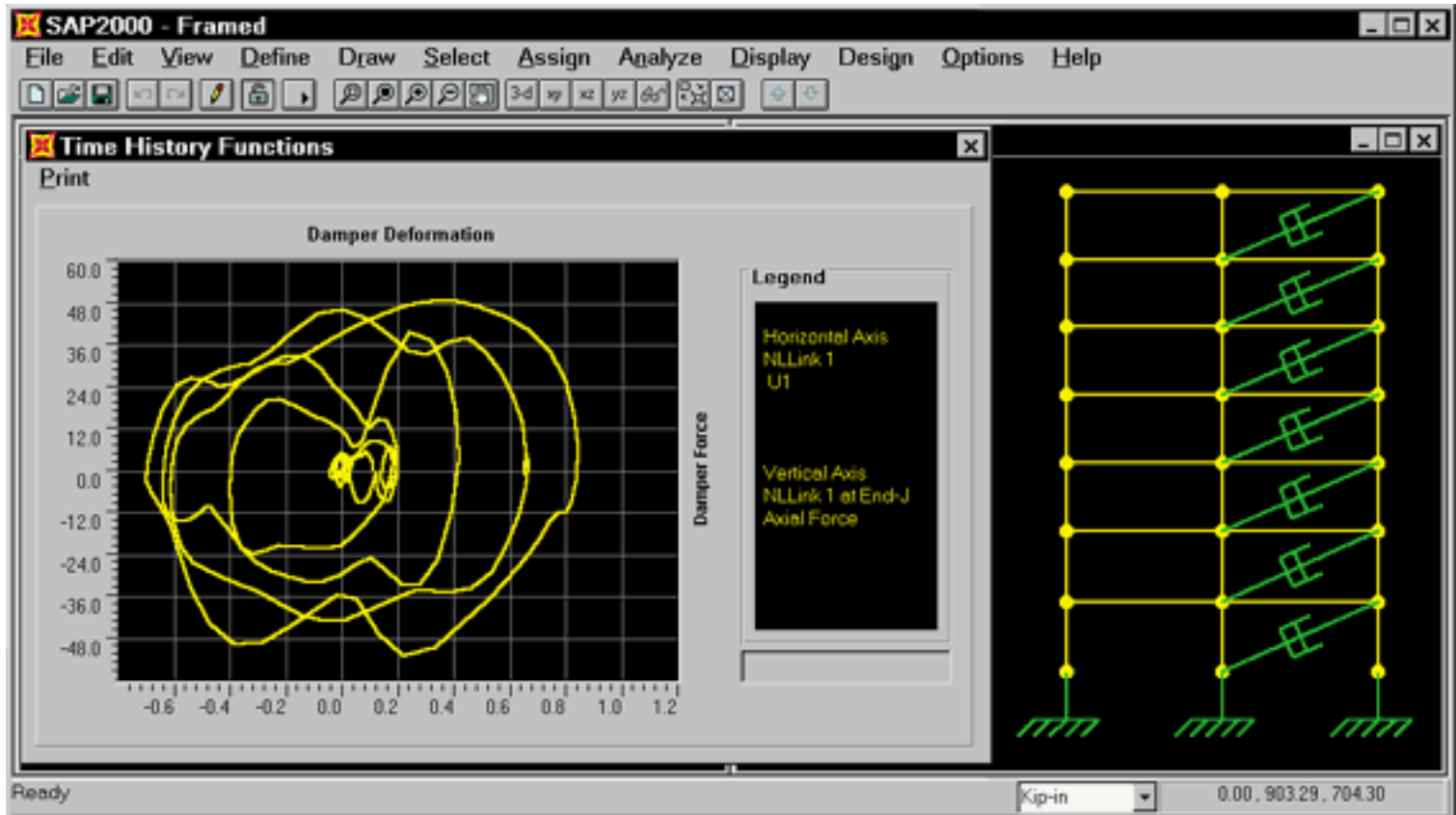
Non-Linear Link Element

- For use with the dynamic time history analysis option
- Link may be placed between any two joints or from joint to ground
- Viscous damper with nonlinear exponent on velocity
- Gap (compression only)
- Hook (tension only)



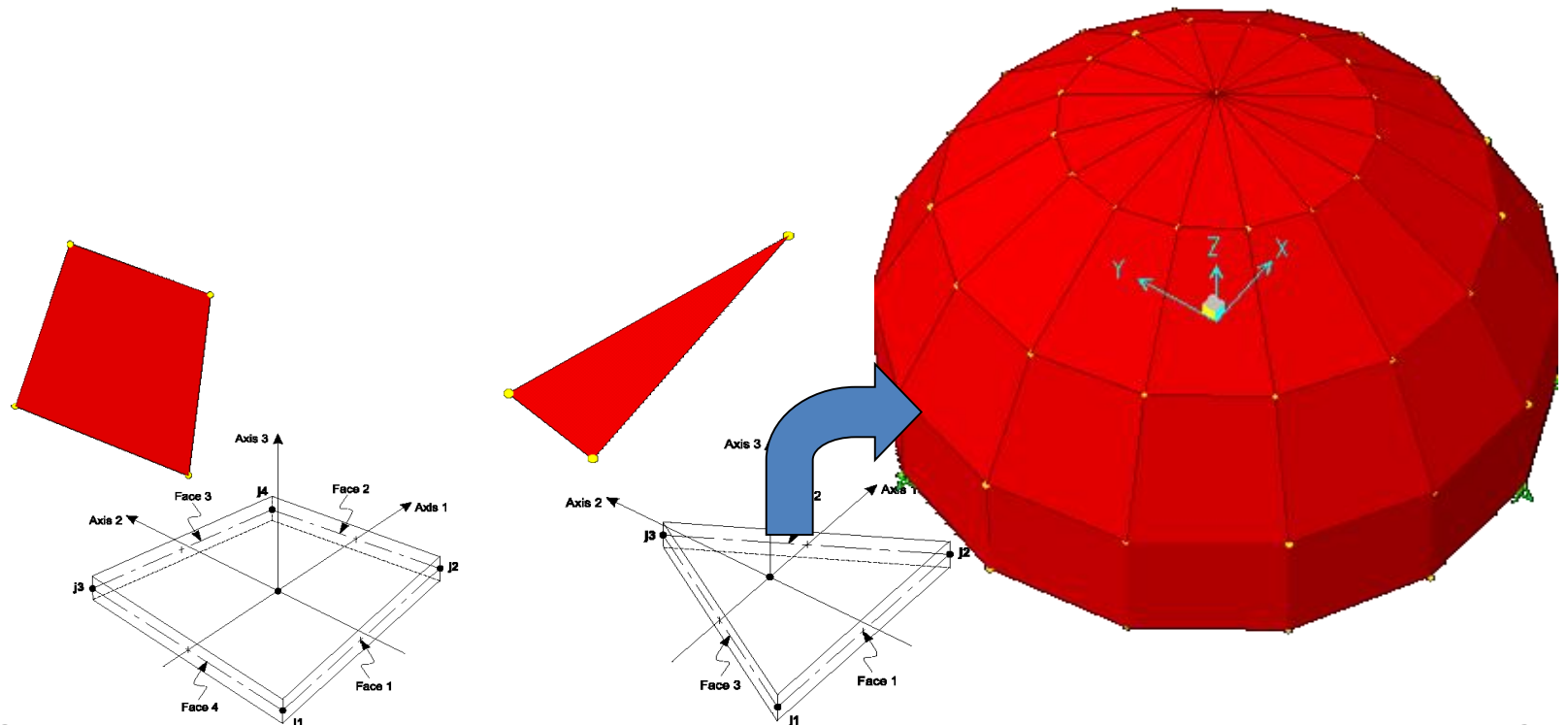
Modelling of Elements

- Force versus deformation plots of nonlinear systems for energy dissipation studies



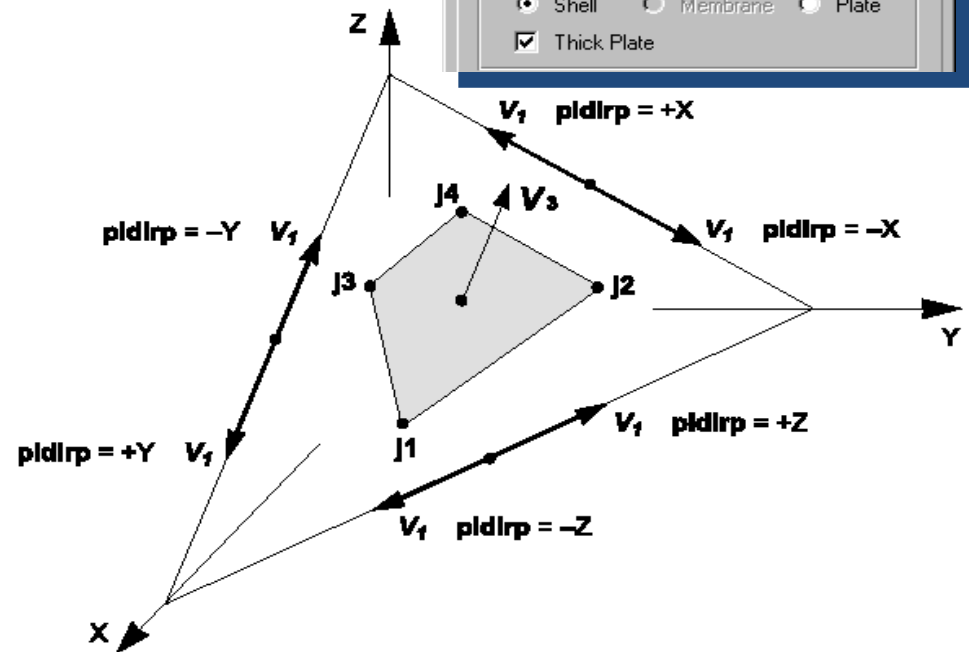
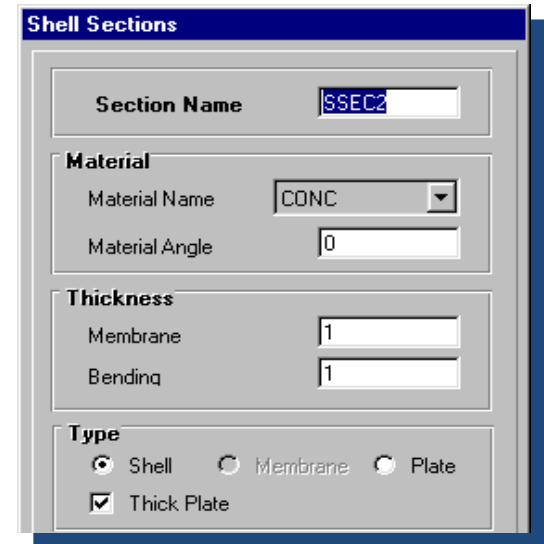
Modelling of Elements

- General quadrilateral or triangular element
- Isotropic, Orthotropic and Anisotropy material



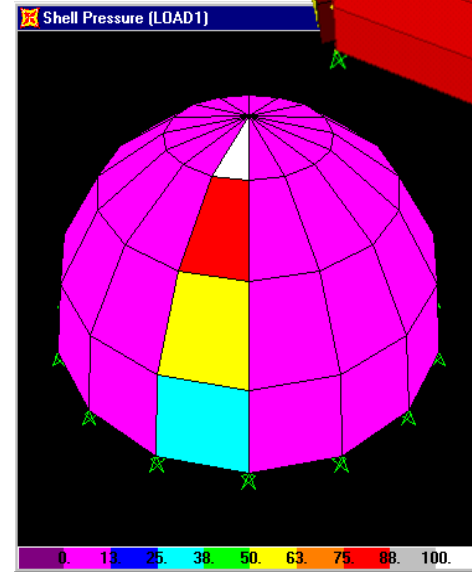
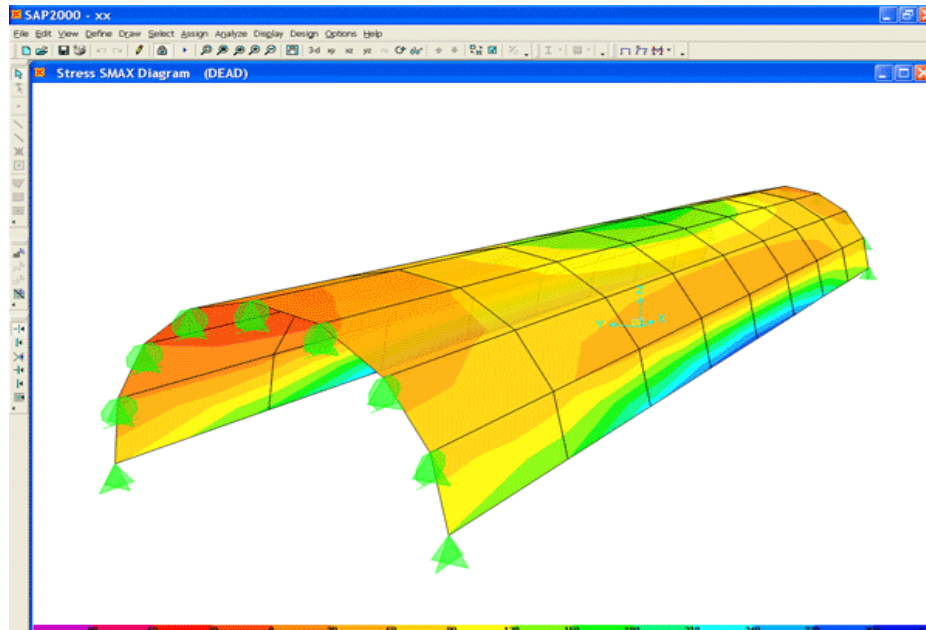
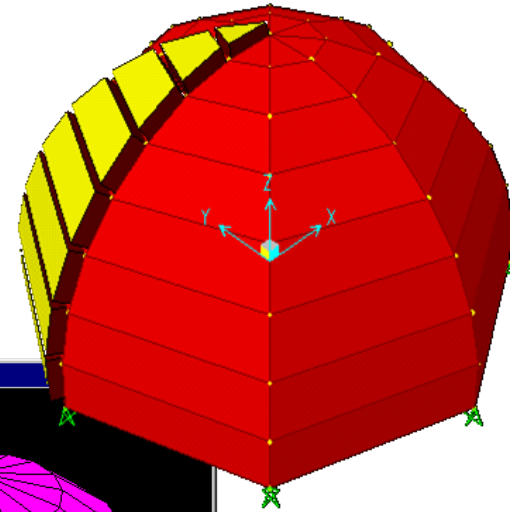
Modelling of Elements

- Six degrees of freedom per joint
- Shell, plate or membrane action
- Thick shell option



Modelling of Elements

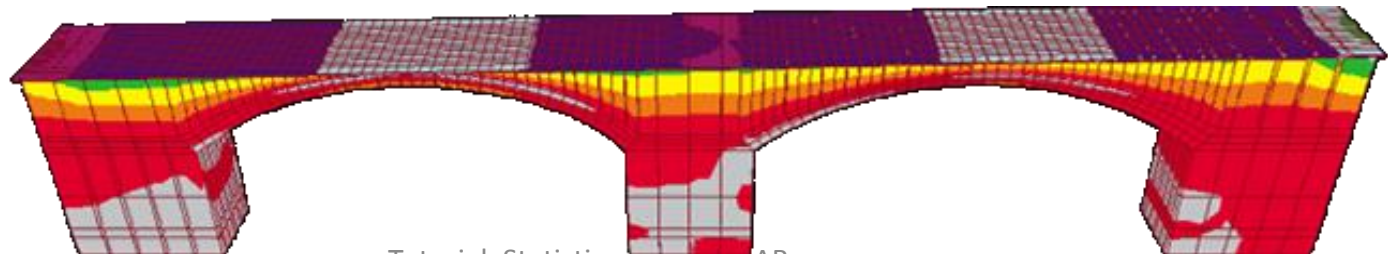
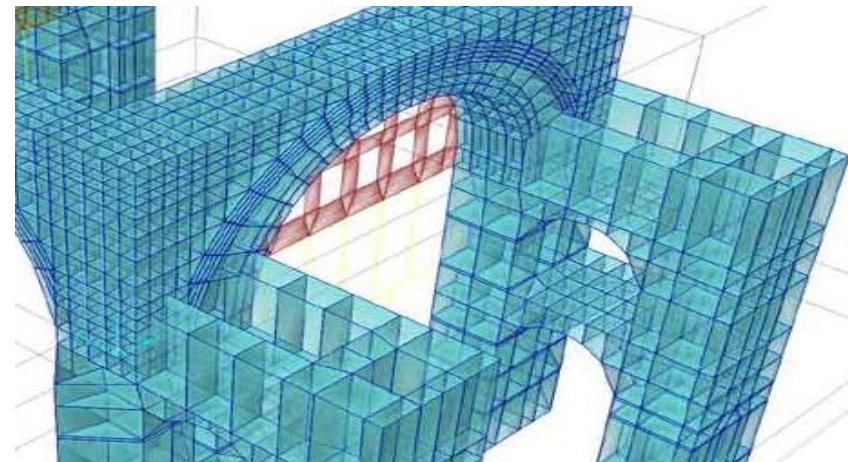
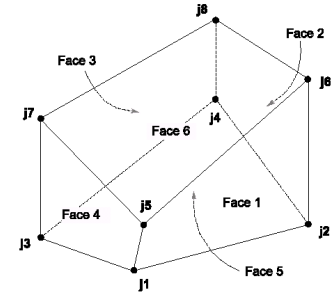
- Gravity and uniform loading
- Pressure loading
- Temperature and thermal gradient loading



Statistics: Overview

Solid Elements

- Three dimensional 8 node brick element
- Anisotropic material
- Gravity, thermal, surface pressure and pressure gradient loading



Modelling of Elements

Joint restraints

- General Spring Connection
- Global and skewed springs
- Coupled 6x6 user-defined spring stiffness option
- modeling

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.

Options

Add to existing springs
 Replace existing springs
 Delete existing springs

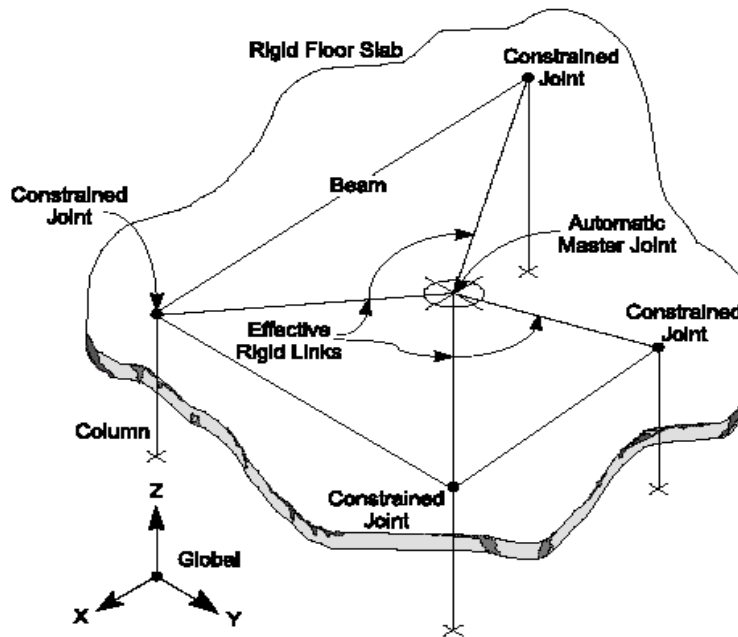
Upper Stiffness Matrix

	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.

Modelling of Elements

Joint constraints

- Generalized joint constraint optical rigid bodies, diaphragms, rods and welds



Weld Constraint

Constraint Name

Weld

Weld Tolerance

Rod Constraint

Constraint Name

Constraint Axis

X Axis Auto

Y Axis

Z Axis

Rod

Diaphragm Constraint

Constraint Name

Constraint Axis

X Axis Auto

Y Axis

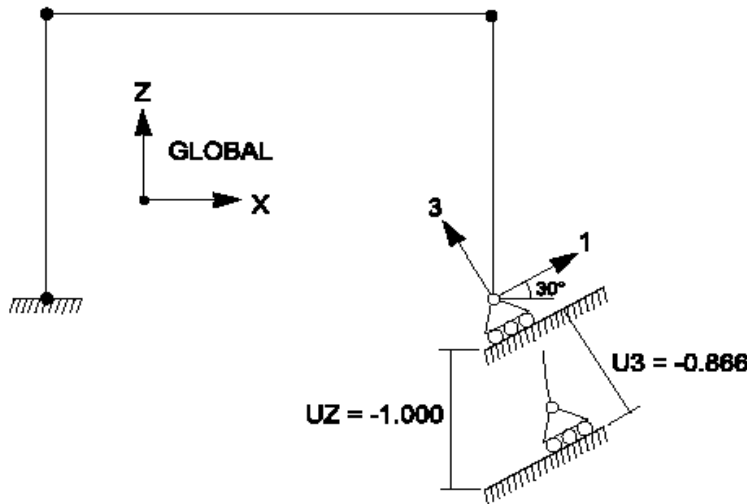
Z Axis

Diaphragm

Modelling of Elements

Joint Loads and Displacements

- Applied force and applied displacement
- Inclined Supports and joint location



Joint Forces

Load Case Name: **LOAD1**

Loads

Force Global X: 0.

Force Global Y: 0.

Force Global Z: 0.

Moment Global XX: 0.

Moment Global YY: 0.

Moment Global ZZ: 0.

Options

Add to existing loads

Replace existing loads

Delete existing loads

OK

Cancel

Ground Displacements

Load Case Name: **DL**

Displacements

Translation X: 0.

Translation Y: 0.

Translation Z: -5.

Rotation about XX: 0.

Rotation about YY: 0.

Rotation about ZZ: 0.

Options

Add to existing loads

Replace existing loads

Delete existing loads

OK

Cancel

Analysis Options

- Static and/or dynamic response spectrum analysis
- P-delta analysis with either static or dynamic analysis
- Blocked active column equation solver
- Automated fast profile optimization
- Non-linear Pushover Analysis

Analysis Options

- Modal Analysis
- Eigenvalue analysis with an iteration algorithm
- Ritz analysis for fast prediction
- Harmonic Steady-State Analysis

Dynamic Analysis Parameters

Number of Modes

Type of Analysis
 Eigenvectors Ritz Vectors

EigenValue Parameters
 Frequency Shift (Center)
 Cutoff Frequency (Radius)
 Relative tolerance
 Include Residual-Mass Modes

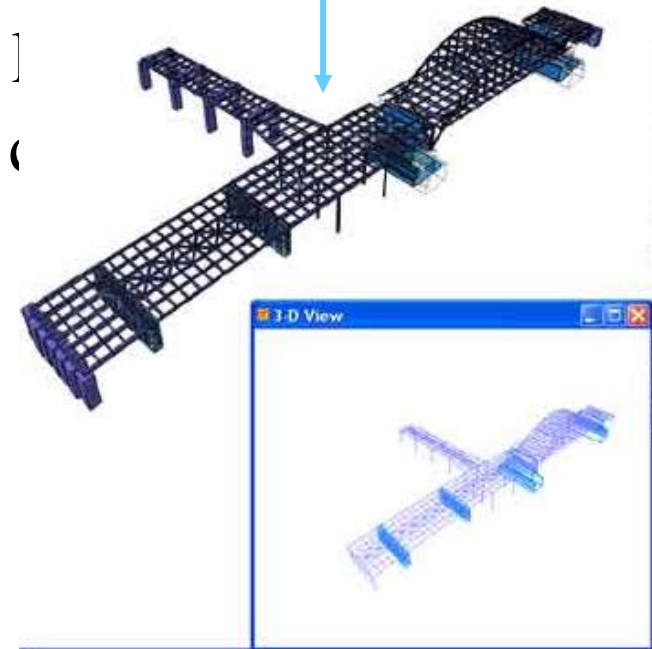
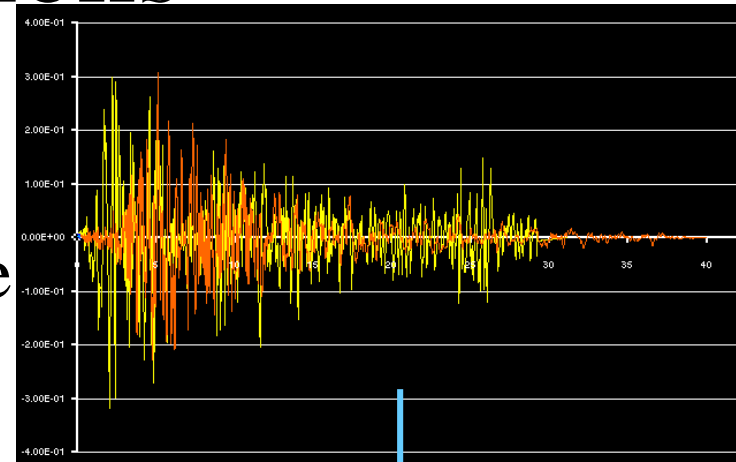
Starting Ritz Vectors

$$Ku(t) + M \ddot{u}(t) = r(t) = p \cos(\varpi t) \quad [K - \Omega^2 M] \Phi = 0$$

Analysis Options

Time History Analysis

- Ground acceleration and Multiple
- Sequential history cases
- Time history Windows AVI file
- Results can be combined with other 1 enveloping or step by step steel and c



Analysis Options

Response Spectrum Analysis

- Multiple response spectrum cases in
- Modal combinations by the SRSS, the
GMC (Gupta) method
- Directional combination by the ABS
method

Response Spectrum Case Data

Spectrum Case Name:

Excitation angle:

Modal Combination

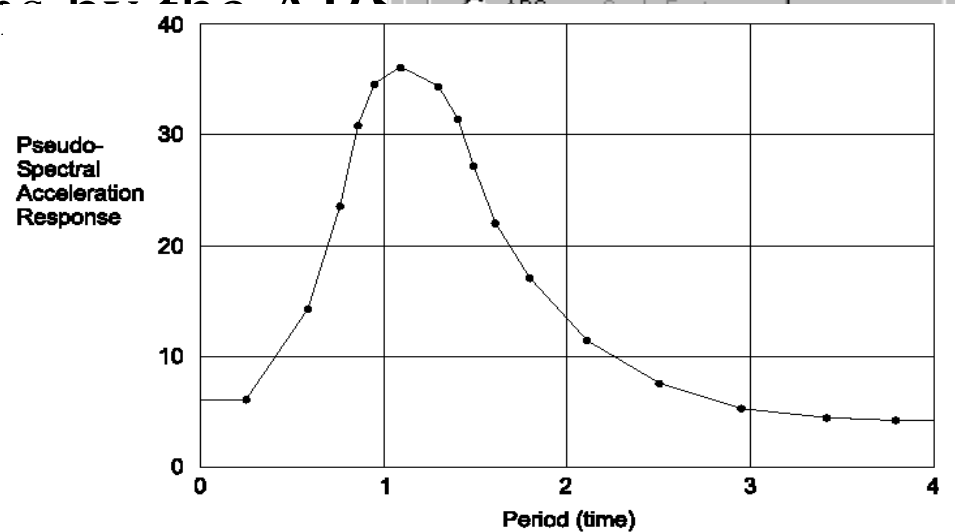
CQC SRSS ABS GMC

Damping:

F1: F2:

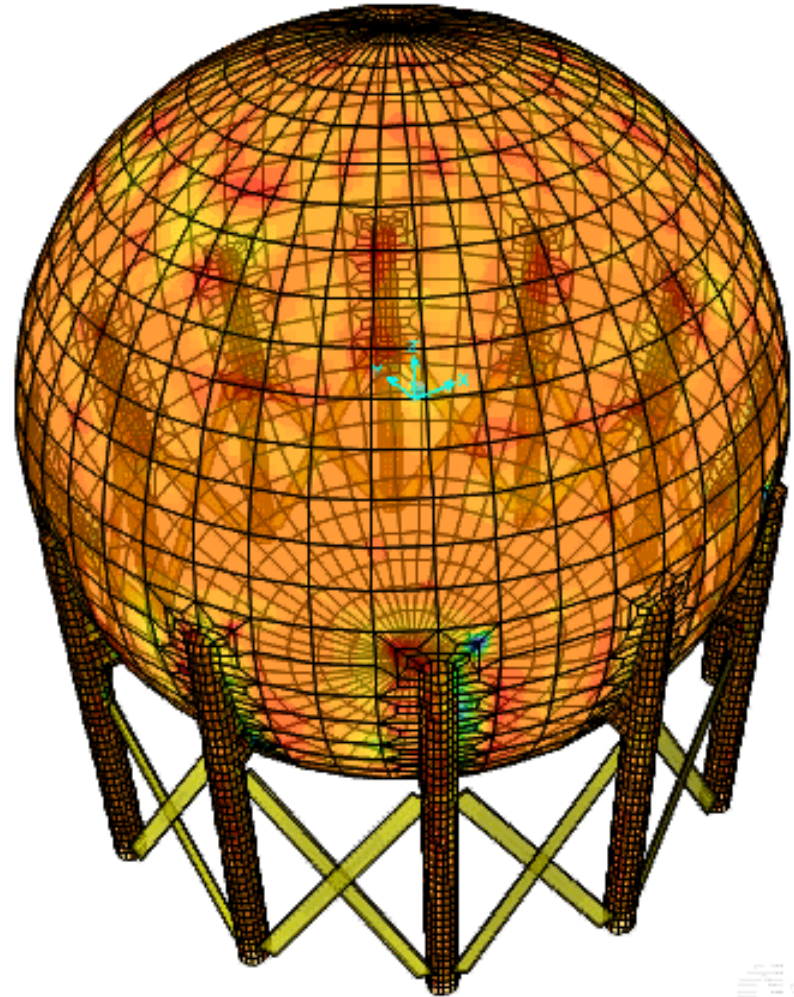
Directional Combination

SRSS ABS



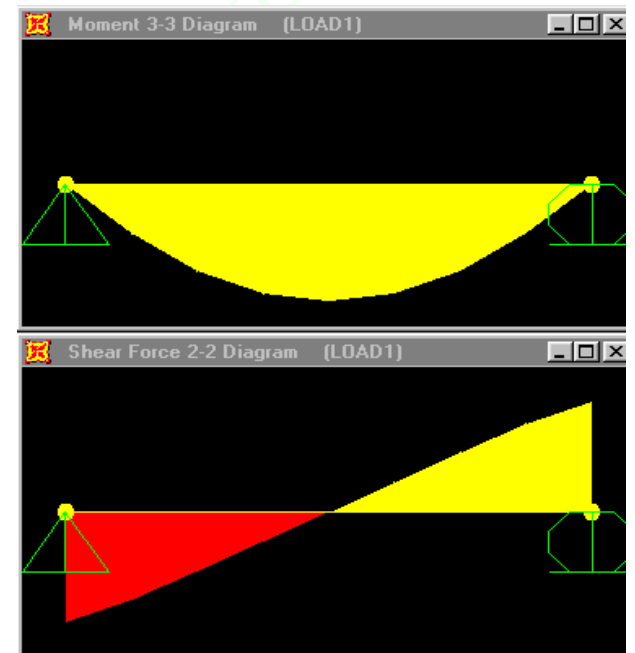
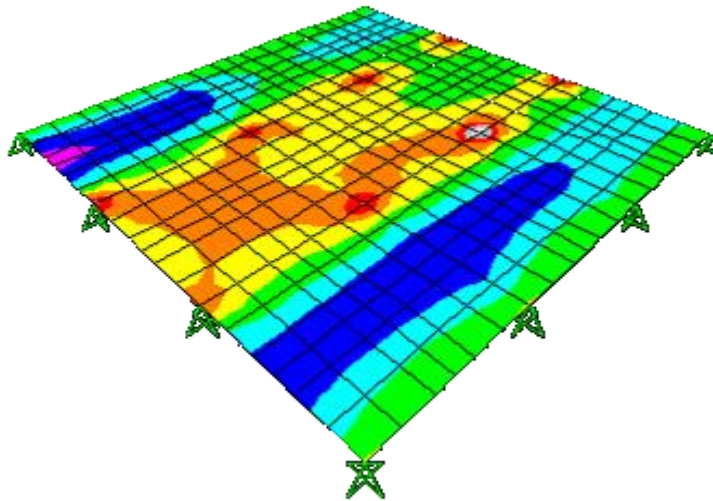
Viewing Results

- 3D perspective graphical and deformed structural
- Static deformed shapes and
- Static and dynamic load



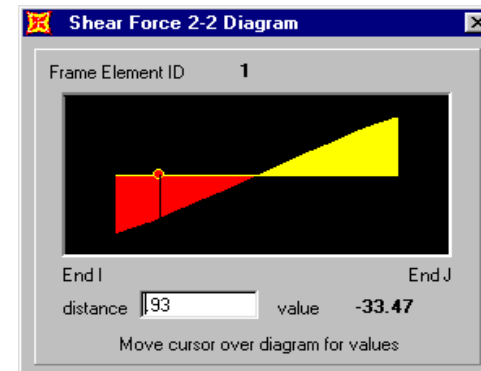
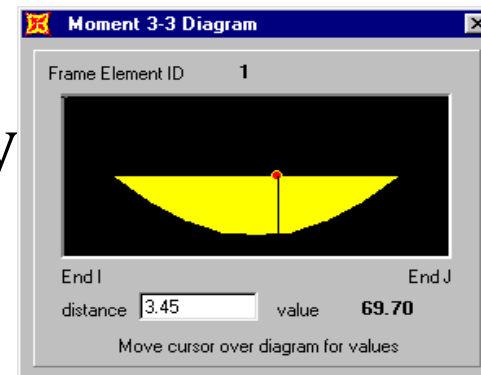
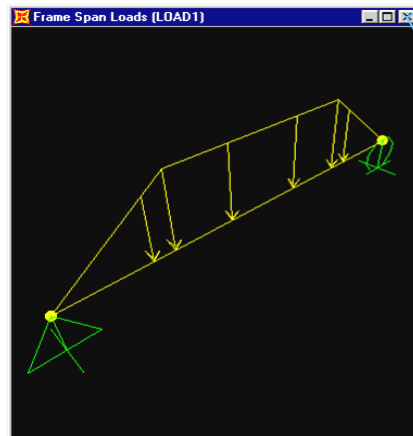
Viewing Results

- View Loading diagrams
- Bending moment and shear force diagrams
- Stress contours



Viewing Results

- Instantaneous graphical and tabulated output details for specific joint or element with right button click
- Multiple windows display parameters

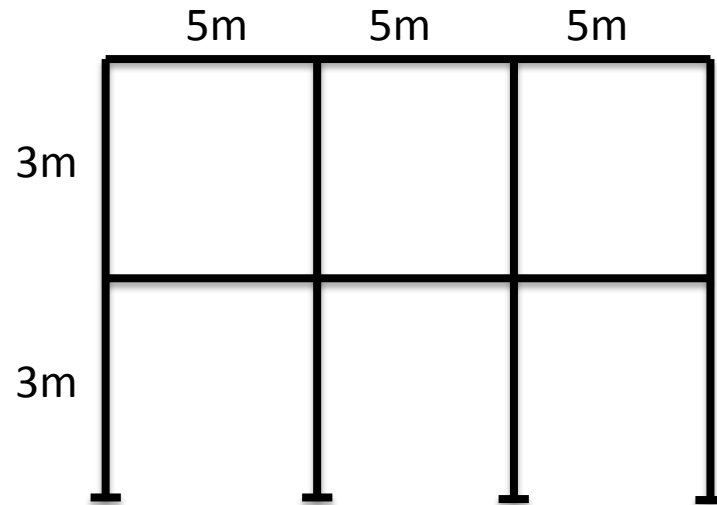
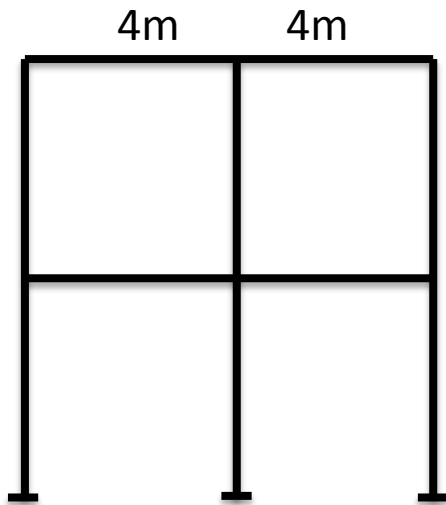


Tutorial Example: Static and Dynamic Analysis of a 3D Truss Frame

Tutorial Example

- 3D RC-Frame (C30/37) with slabs.
- 3x2 spans, 2 storeys
- Dimensions:
 - Span in x-dir= 5m; y-dir= 4m; storey height = 3m
 - Beams: $h=50$; $w = 30$ cm
 - Columns: $w1 = w2 = 60$ cm
 - Slabs: $t = 20$ cm
- Loads: Self weight, live load, SLS (Factors= 1)

Tutorial Example



Further Reading

- <http://www.csiamerica.com/products/sap2000/watch-and-learn>
- http://ocw.mit.edu/courses/civil-and-environmental-engineering/1-051-structural-engineering-design-fall-2003/projects/SAP2000_Tutorial2.pdf
- http://www.grad.hr/csi/web_manuals/01%20%20%20SapStart.pdf.
- <http://ctgttp.edu.free.fr/TRUNGWEB/Bai%20Giang/Bai%20giang%20va%20Vi%20du%20SAP/SAPWEB01.pdf>

Q&A