BUILDING SUPPORT STRUCTURES

a visual study with computers

COMPUTER-BASED ANALYSIS AND DESIGN of BUILDING STRUCTURES

@Seismicisolation

This presentation addresses the **use of structure software** in architectural education besides being concerned with the

- the position of structures in architecture
- a clarification of structural engineering and architecture which often is confusing to the students

@Seismicisolation

GENERAL CONCPTS OF STRUCTURE

The necessity of Structure: ORDER, structure is a necessary part of life

The purpose of building structure:

- support structure
- ordering system
- form giver
- art

Building vs. Structure vs. Architecture:

structure is necessary for buildings but not for architecture: without structure there is no building, but architecture as an idea does not require structure

Expression of structure:

- hidden vs. exposed
- innovative vs. standard construction

Building structure type and use

Global vs. local scale of structure: DETAIL

STRUCTURE as *support* (*local and global scale*): structure holds the building up so it does not collapse or deform excessively; it makes the building and spaces within the building possible. Structure gives support to the material and therefore is necessary. It is directly related to the structural engineer who is responsible for safety, to him the building is a body that is alive, its bones and muscles are activated by external and internal forces. As it reacts, it deforms and suggests the pain it must endure at points of stress concentration. In other words engineers visualize buildings in an animated state moving back and forth as can be convincingly expressed by computers through virtual modeling. It is this world of engineering which fascinated the early modernists in architecture.

STRUCTURE as *form giver*: it defines the spatial configuration and reflects other meanings and is part of esthetics

STRUCTURE as *ordering system*, it functions as a spatial and dimensional organizer besides identifying assembly or construction systems: dimensional coordination

Most students have completed at least two prior courses in structures: *introduction to engineering analysis*, and *introduction to structural design* including concrete, steel and wood. Because of this background only static loading is considered and small displacement of elements so that the magnitude of the member forces only depends on relative stiffness values, in other words the original geometry is used and not the deformed geometry. Even if the **linear analysis** does not simulate the actual behavior, it does give for ordinary conditions member forces good enough for making design decisions, at least for preliminary design purposes, besides helping to develop an understanding for the behavior of the building structure. ***Indeterminate structures often are made determinate so that member forces are independent of stiffness that is independent of certain modeling assumptions, and manual calculations can be performed to check the computer output approximately. The actual displacement, which depends on the absolute stiffness, is usually only checked visually.***

Since the free academic version of SAP2000 is limited to **100 nodes**, generally only planar structures are investigated by the students. The computer program is introduced by proceeding from simple to more complex structures.

A. **General Introduction**

B. **Case Study**

@Seismicisolation

The goal of the use of computers in structure course is to develop an understanding for

the building structure as a system that supports and as a pattern that orders space and makes space possible.

The treatment of structures is broadened and enriched by integrating the traditionally separate fields of:

- **construction**
- **structural analysis**
- **structural design**
- **structural systems**
- **materials**
- **geometrical modeling**
- **visual communication**

The students have to synthesize the knowledge acquired in various courses and have to set up a mathematical model of the building support structure, rather than solving a given isolated analysis or design problem, as is usually done in education; they have to deal with the physical reality of the entire building rather than only an isolated part.

The primary structural engineering software used in this context is **SAP2000**, which is widely used in practice and in numerous universities internationally. The program has been developed by **COMPUTERS AND STRUCTURES**, INC (www.csiberkeley.com/SAP2000.htm)., Berkeley, CA.

Refer also to:
Beijing Civil King Software Technology Co., Ltd., Beijing, Chushu LI, PHD, S.E., Chief Executive, Tel:86-10-8838 3866-101, Mobile:13601318851, Fax:86-10-88381056, Email: csli@chinabuilding.com.cn, Web: www.bjcks.com

Qualified universities are eligible for free software for Education and Research.

Since the free academic version of SAP2000 is limited to **100 nodes**, generally only planar structures are investigated by the students. The computer program is introduced by proceeding from simple to more complex structures.

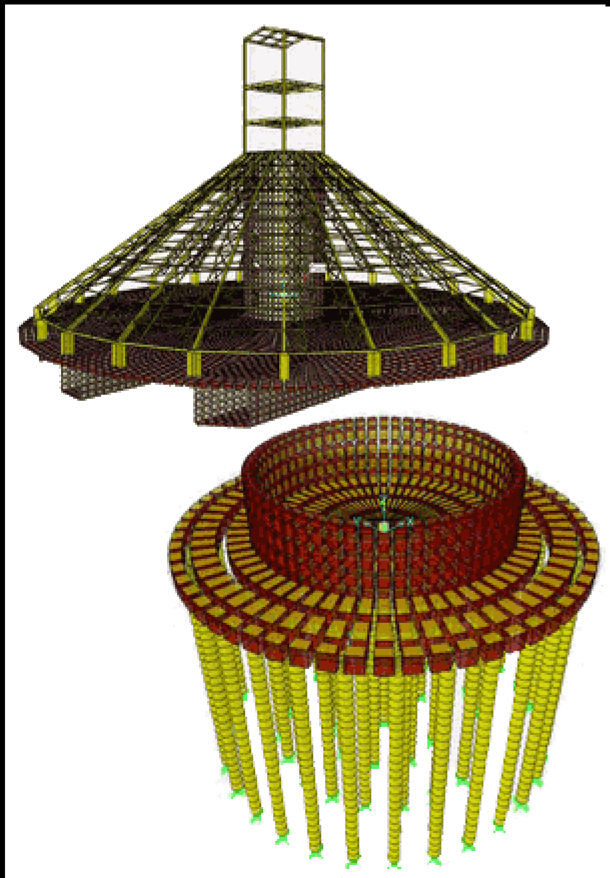
The program is fully integrated within Microsoft Windows and allows modeling of nearly all types of structures. The Windows based easy-to-use graphical interface permits the quick modeling of structures with templates and then to edit them via the graphical interface.

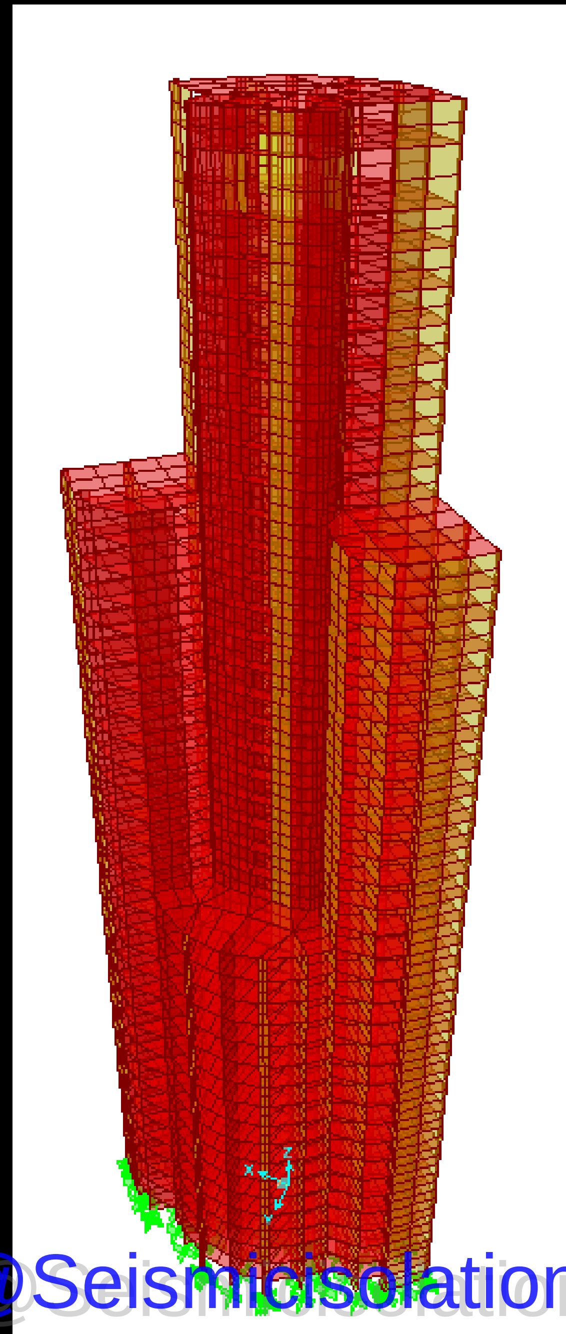
SAP2000

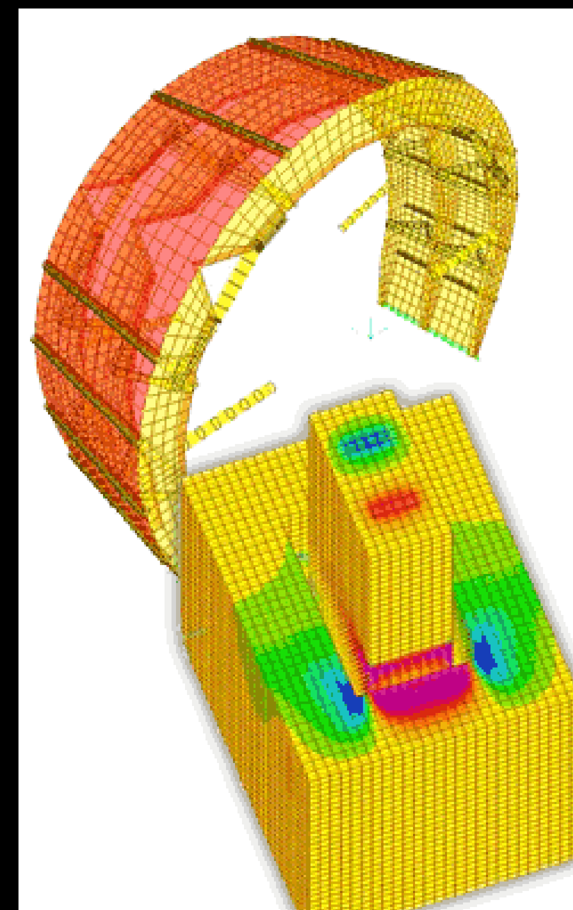
V11 – Non Linear

A Comprehensive Software for the Finite Element Modeling, Static, Dynamic and Non-Linear Analysis and Design of Structures

@Seismicisolation

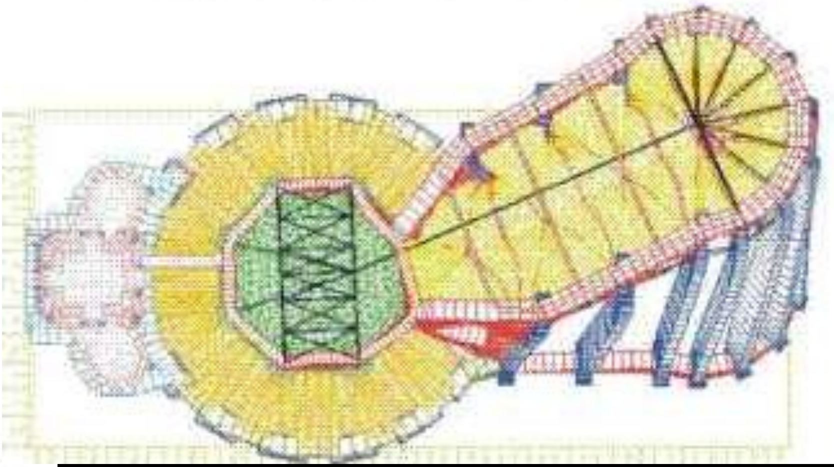






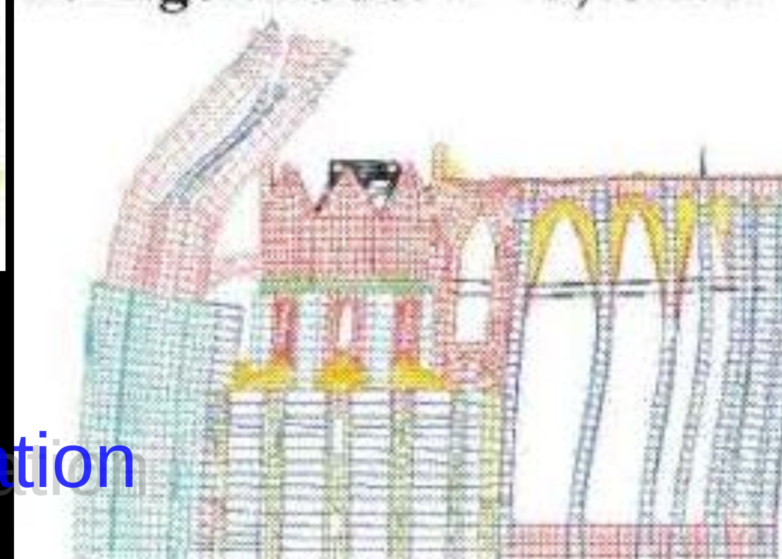
Finite-element model

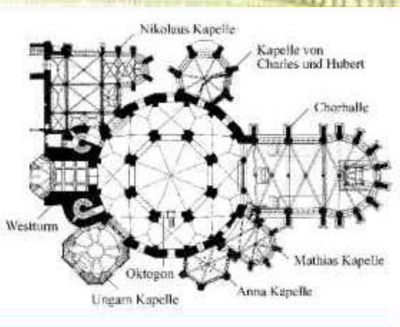
1. Eigenmode: f = 1,13 Hz



Seismic Vulnerability Assessment of the Historic Aachen *Cathedral*, *Germany*

2. Eigenmode: f = 2,13 Hz





To develop an understanding for the behavior of the **actual building** under load action, so it can be designed, engineers have created a **mathematical model** that is an abstract mathematical construction of an **idealized structure** that simulates the real building structure.

A typical mathematical model is defined by the principal components, that decomposes the building structure into its major parts or **BUILDING BLOCKS** of:

- ***elements*** (type, shape, material, other characteristics) and ***nodes***
- ***boundary conditions:***
  - ***support joints***
  - ***active member joints***
- ***material***
- ***action: external forces*** (e.g. nodal loads and displacements, member loads, temperature changes)

In other words, the structure consists of an assembly of members and joints at their ends or their corners, as well as the external supports of the global or overall structure.

INTRODUCTION

Most computer-based analyses of structures are based on finite element methods, where numerical methods approximate the equations for a continuous system. Here, the name *finite* suggests the distinction from *differential* elements in calculus. The finite element analysis was developed during the 1950s in the aerospace industry and advanced rapidly further during the 1960s with the growth of computer power as engineers turned to numerical methods using matrix mathematics rather than differential calculus. In other words, engineers began to replace classical techniques to solve unknowns by the flexibility method (i.e. force method) from static equilibrium of forces, or by the stiffness method (i.e. displacement method) from compatibility of displacements, with matrix analysis.

As the name suggests, in the finite element method the continuous structure is decomposed into a number of one-, two-, or three-dimensional elements with artificial joints and subsequently reassembled. The finite elements can represent a piece or part of a structure. For example, in a truss each member represents a natural element of the finite element model. Also, in a frame each shallow beam or column can be represented simply as bar-type elements, but they can be further subdivided by a number of elements for stress analysis that is to study in more detail the stress flow. Many structures such as skeletons already represent a clearly identified assemblage of discrete members and therefore can be idealized with one-dimensional or bar-type elements (e.g. beams, columns, arches, struts, and ties).

On the other hand, for surface structures and complex three-dimensional structures, the continuum must be divided into a temporary mesh or gridwork of finite pieces of polygonal elements which can have various shapes. For instance, typical slabs, deep beams, walls, and shells can be divided into rectangular grids of planar elements. Examples of finite element models in the following figure are Joan Miro's *30-ft* high outdoor sculpture in Chicago designed by SOM, or the shell dome according to Muecke Software GmbH.

The computer program uses the ***finite element*** *analysis* as the mathematical model, however the students do not have to be familiar with this numerical modeling method. It surely is helpful to **visualize that the model is divided into a large number of finite elements**. The elements are considered **connected to *nodes*** typically located at the ends or corners of members.

In other words, in the finite element method the continuous structure is decomposed into a number of one-, two-, or three-dimensional elements with artificial joints and subsequently reassembled. The finite elements can represent a piece or part of a structure. For example, in a truss each member represents a natural element of the finite element model. Also, in a frame each shallow beam or column can be represented simply as bar-type elements, but they can be further subdivided by a number of elements for stress analysis that is to study in more detail the stress flow. Many structures such as skeletons already represent a clearly identified assemblage of discrete members and therefore can be idealized with one-dimensional or bar-type elements (e.g. beams, columns, arches, struts, and ties).

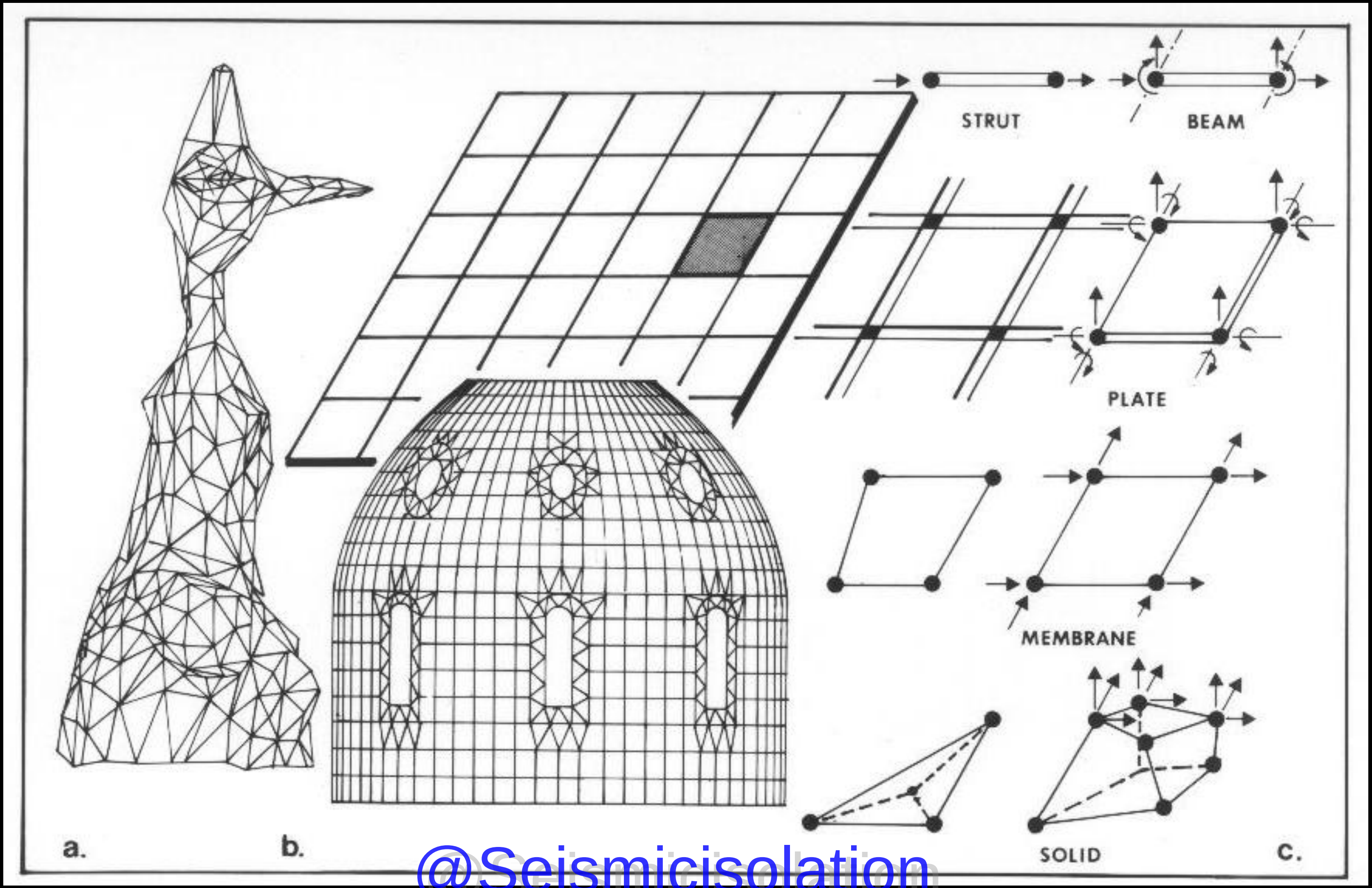
On the other hand, for surface structures and complex three-dimensional structures, the continuum must be divided into a temporary mesh or gridwork of finite pieces of polygonal elements which can have various shapes. For instance, typical slabs, deep beams, walls, and shells can be divided into rectangular grids of planar elements. Examples of finite element models in the following figure are Joan Miro's *30-ft* high outdoor sculpture in Chicago designed by SOM, or the shell dome according to Muecke Software GmbH.

@Seismicisolation

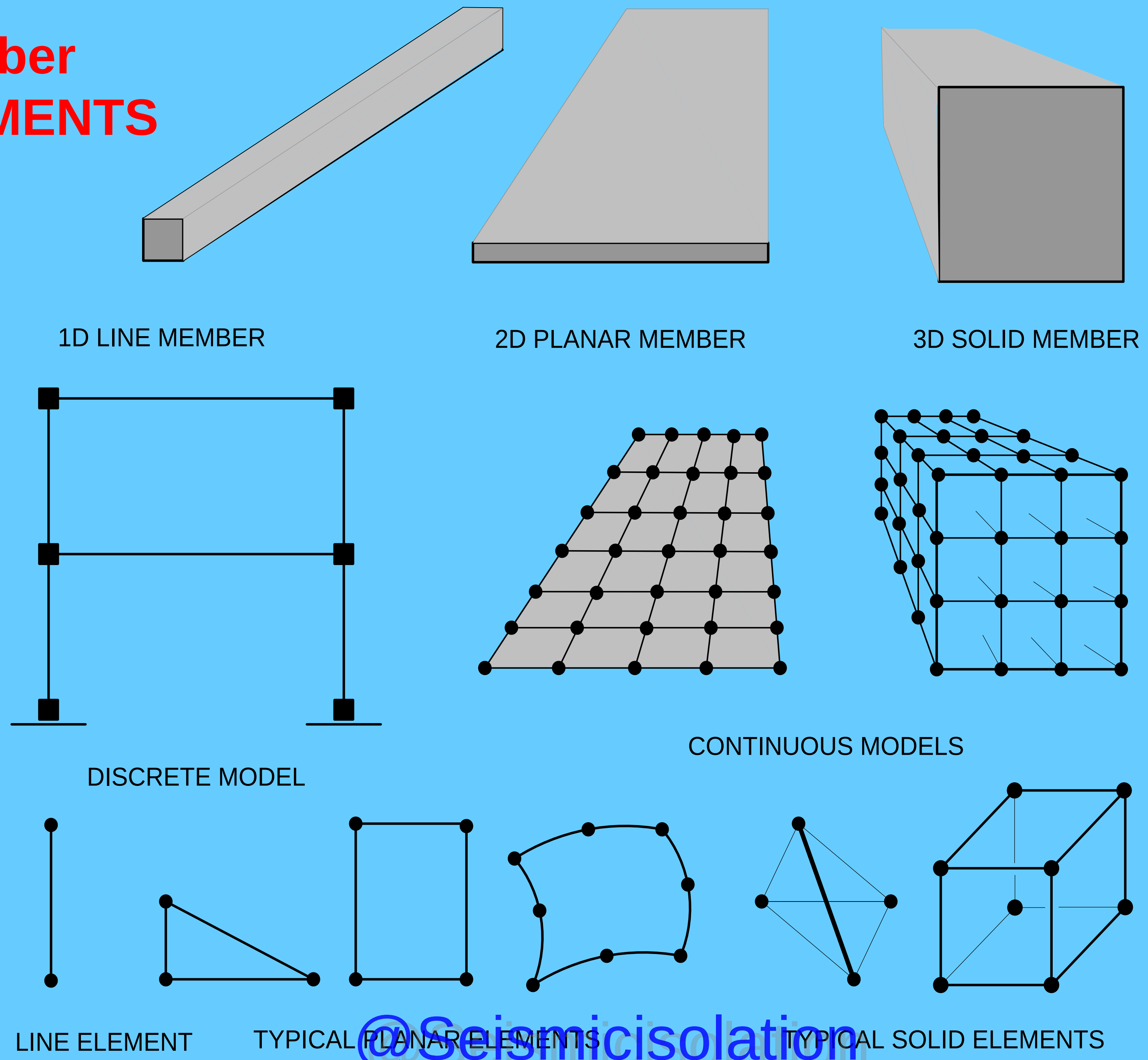
It is not necessary for the students to be able to set up the equations although assumptions and limitations of the method should be discussed. But it must be emphasized that finite element computer programs do not only represent a powerful method of engineering analysis, they also represent a **tool for learning**. **The student must understand the physical reality of the building structure in every detail to set up the mathematical model he puts into the computer**. He develops a feeling and control over the support structure by zooming from the **global scale** of the overall building behavior to the **local scale** of stress and detail.

@Seismicisolation

FINITE ELEMENTS



Member ELEMENTS



STRUCTURAL ELEMENTS

FRAME elements

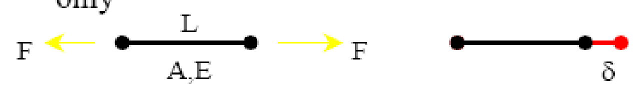
- **beam elements**
- **axial elements**
- **beam-column elements**

SURFACE elements (slabs, plates, shells)

SLID OBJECTS

AXIAL ELEMENTS: truss and cable elements

Defn: Two force members, ie axial loads at nodes only



Elastic Properties:

$k_a = EA/L$ (axial stiffness)

$\sigma = F/A$ (normal stress)

$\delta = FL / EA$ (deflection)

BEAM ELEMENTS

Defn: Members subject to bending and shear

Elastic Properties:

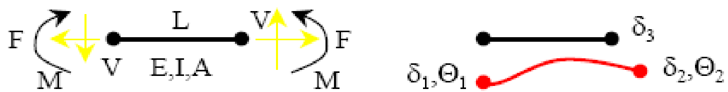
$k_b = f(EI/L^n)$ (bending) $\sigma = My/I$ (normal stress)

$k_s = GA/L$ (shear) $v = VQ/Ib$ (shear stress)

$\delta_b = f$(load, support conditions, L, E, I) (bending)

BEAM-COLUMN ELEMENTS

Defn: Members subject to bending, shear, and axial



Elastic Properties:

$k_a = EA/L$ (axial) $\sigma_a = F/A$ (normal stress)

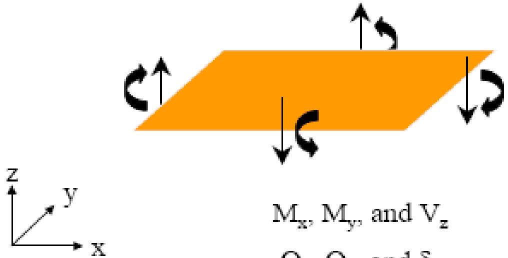
$k_b = f(EI/L^n)$ (bending) $\sigma_b = My/I$ (normal stress)

$k_s = GA/L$ (shear) $v = VQ/Ib$ (shear stress)

$\delta_b = f$(load, support conditions, L, E, I, A) (normal)

PLATE/SHELL ELEMENTS: slab elements

Defn: Members subject to bi-directional bending & shear

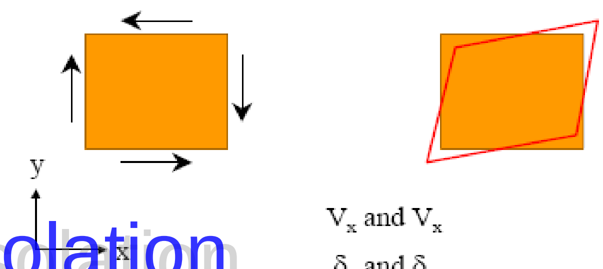


M_x, M_y, and V_z

Θ_x, Θ_y, and δ_z

MEMBRANE ELEMENTS: wall/diaphragm elements

Defn: Members subject to shear

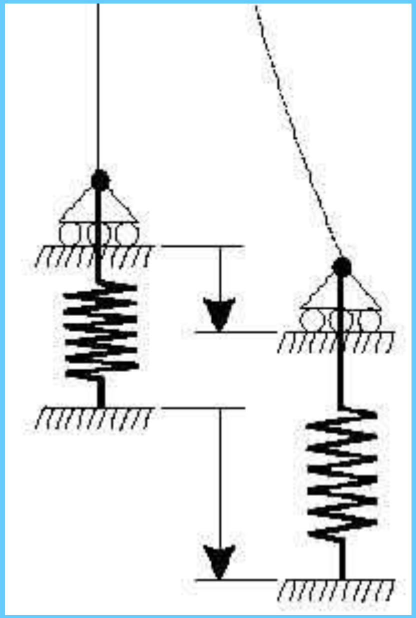


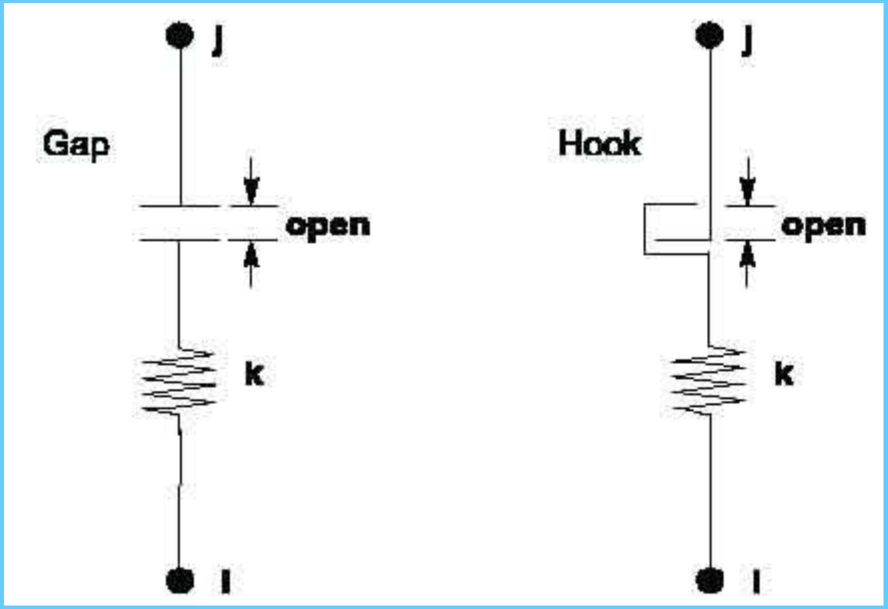
V_x and V_x

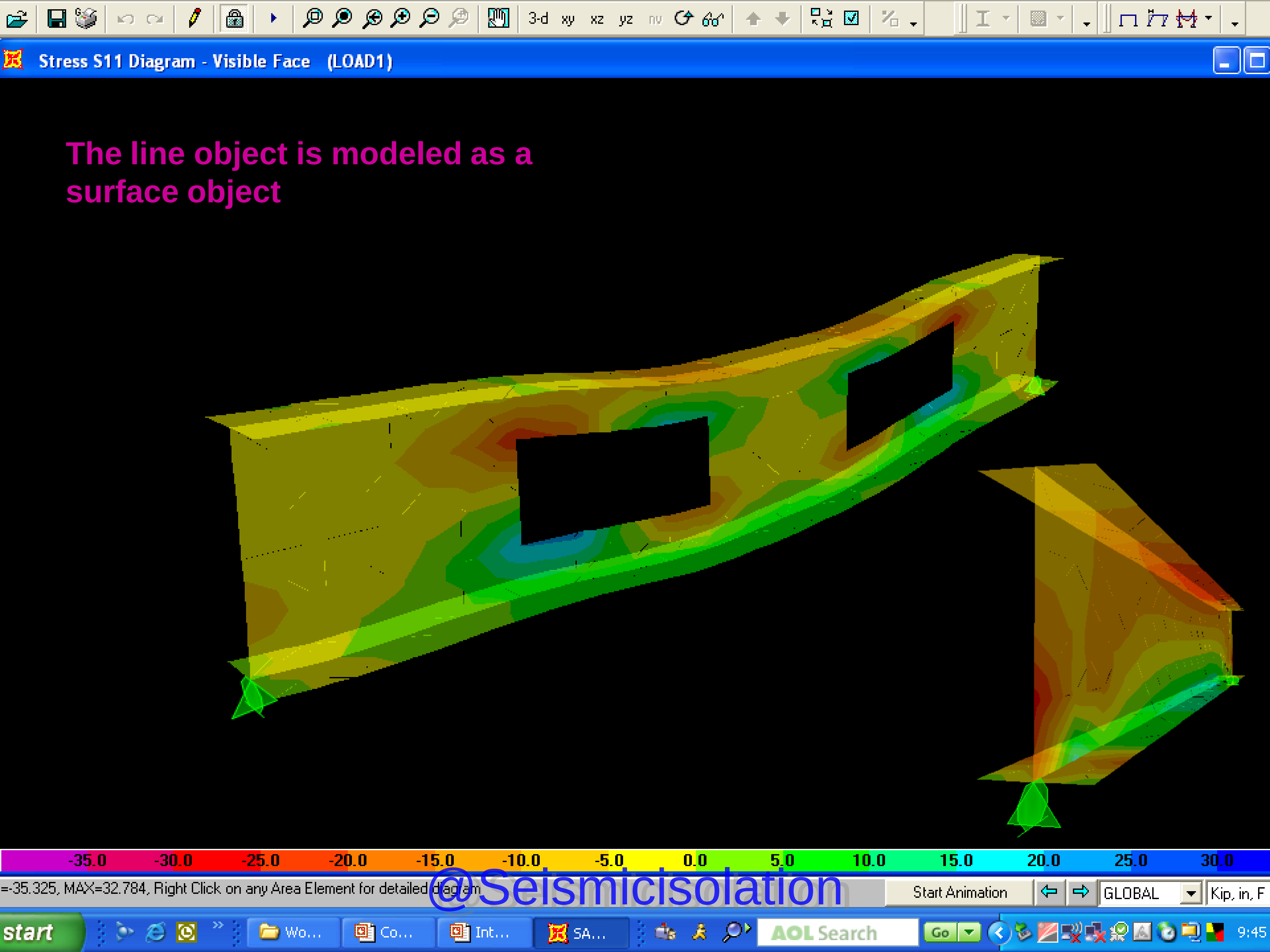
δ_x and δ_y

Other elements,

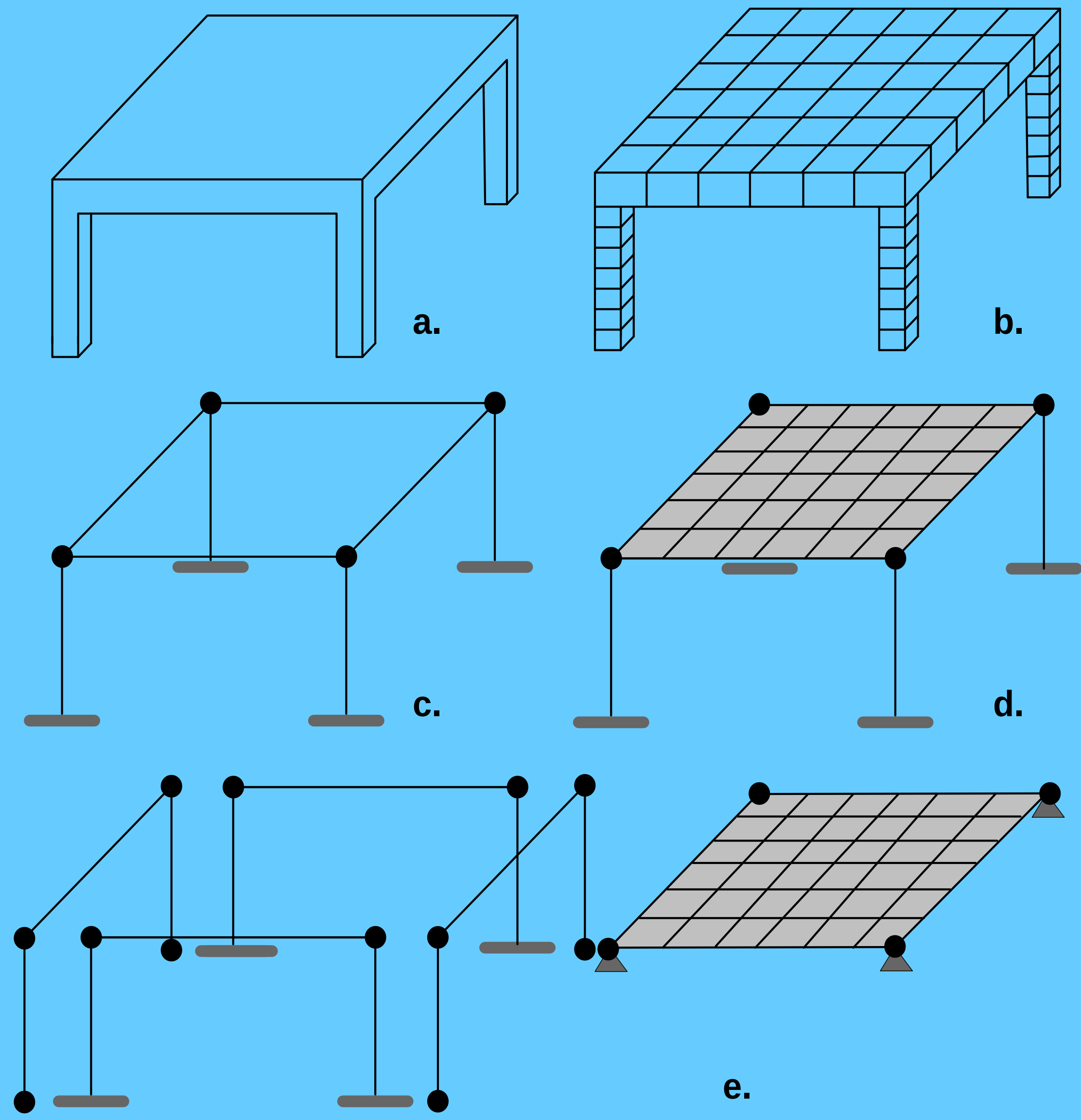
- **Spring element**
- **Link element**
- **Cable element**







Stress S11 Diagram - Visible Face (LOAD1)
The line object is modeled as a surface object
-35.0 -30.0 -25.0 -20.0 -15.0 -10.0 -5.0 0.0 5.0 10.0 15.0 20.0 25.0 30.0
=-35.325, MAX=32.784, Right Click on any Area Element for detailed diagram
@Seismicisolation
Start Animation
GLOBAL
Kip, in, F
start
AOL Search
Go
9:45

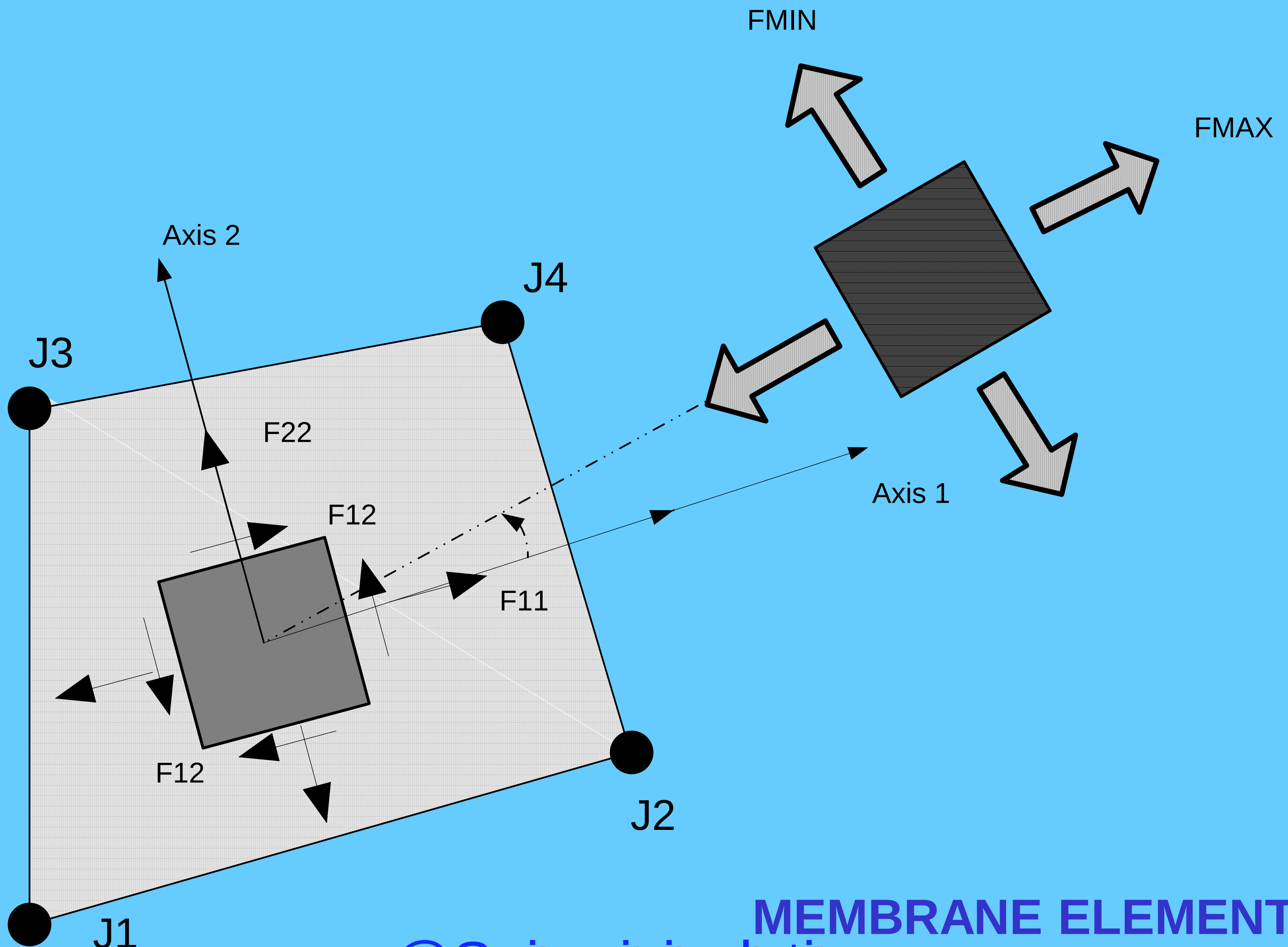


POSSIBILITIES of MODELLING a STRUCTURE

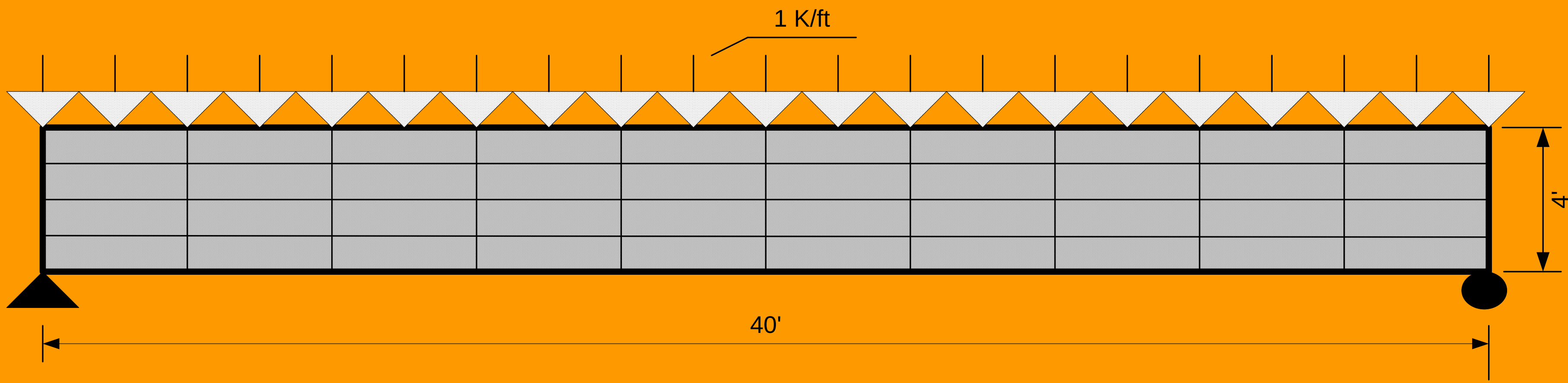
In the decomposed state, the elements (deformable bodies) are considered connected to *nodes* (infinitely small rigid bodies) typically located at the ends or corners of members (see previous figure).

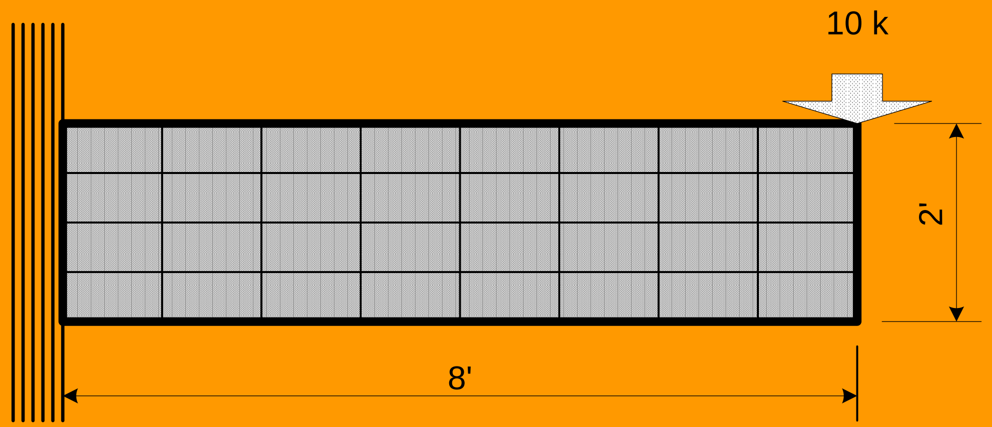
The loads, or in more general terms the action, on the elements are converted to nodal loads, which in turn causes the nodes to respond by displacing. *Displacement equations* (i.e. using typically the stiffness method) are formulated for each element. When the elements are reassembled to form the overall structure, compatibility at the boundaries of the elements must exist. In other words, independent, simultaneous equations are set up, which are equal to the number of unknowns and can then be solved by computer so the unknown displacements, hence internal forces, are found. Extensive mathematical computations are required because of the huge number of finite elements especially in surface structures and the corresponding number of simultaneous equations to be developed which can only be handled by computers.

One must keep in mind, however, that the finite element analysis represents a method of approximation. It does not give exact results except for simple frame and membrane problems which are similar to theoretical closed-form solutions.



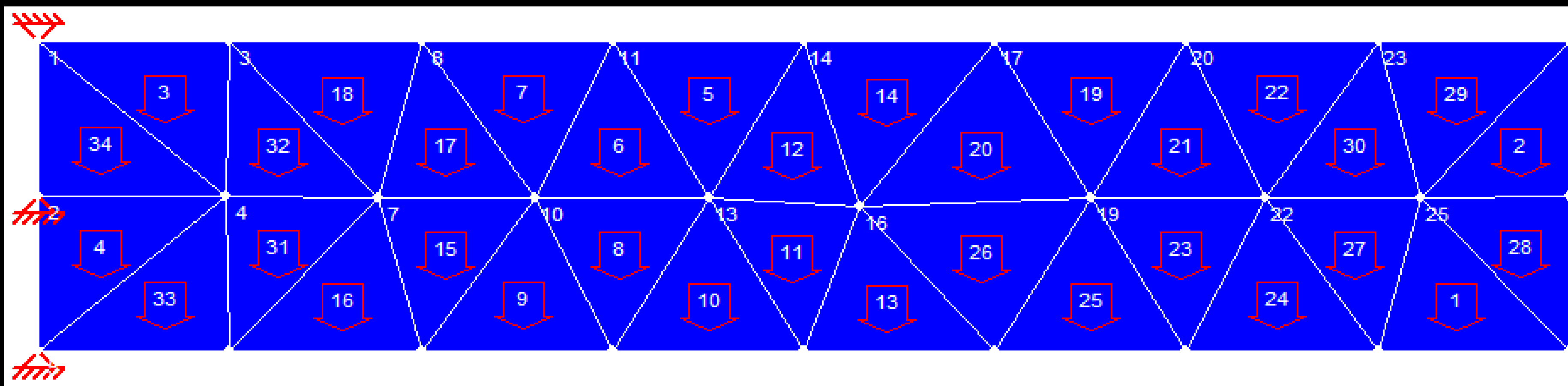
MEMBRANE ELEMENT





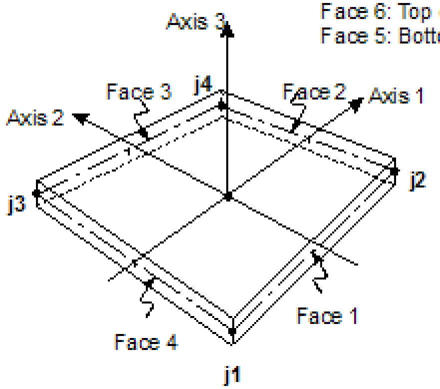
MEMBRANE ELEMENTS

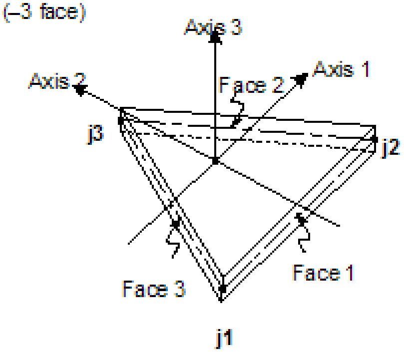
@Seismicisolation



MEMBRANE ELEMENTS

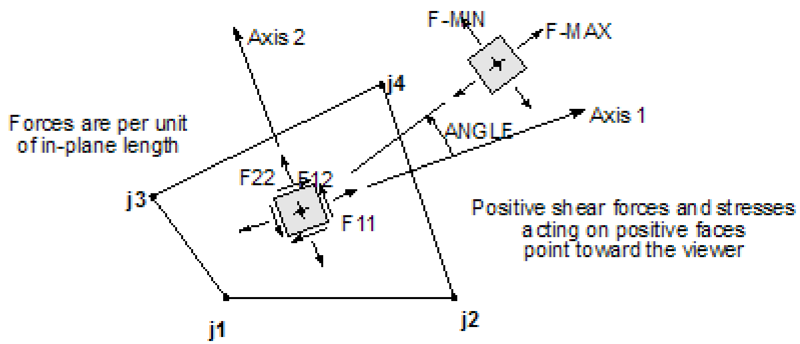
Shell Element





Four-Node Quadrilateral Shell Element

Three-Node Triangular Shell Element



Membrane Forces

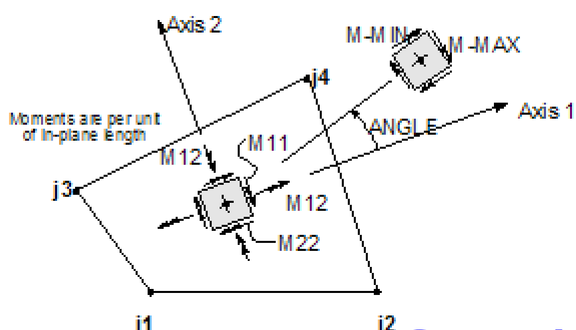


Plate Bending and Twisting Moments

Shell Internal Forces

FINT = F intermediate

FVM =√{1/2[(FMAX - FINT)^2+(FMAX - FMIN)^2+(FINT - FMIN)^2]}.

Also See: Von Mises Stress

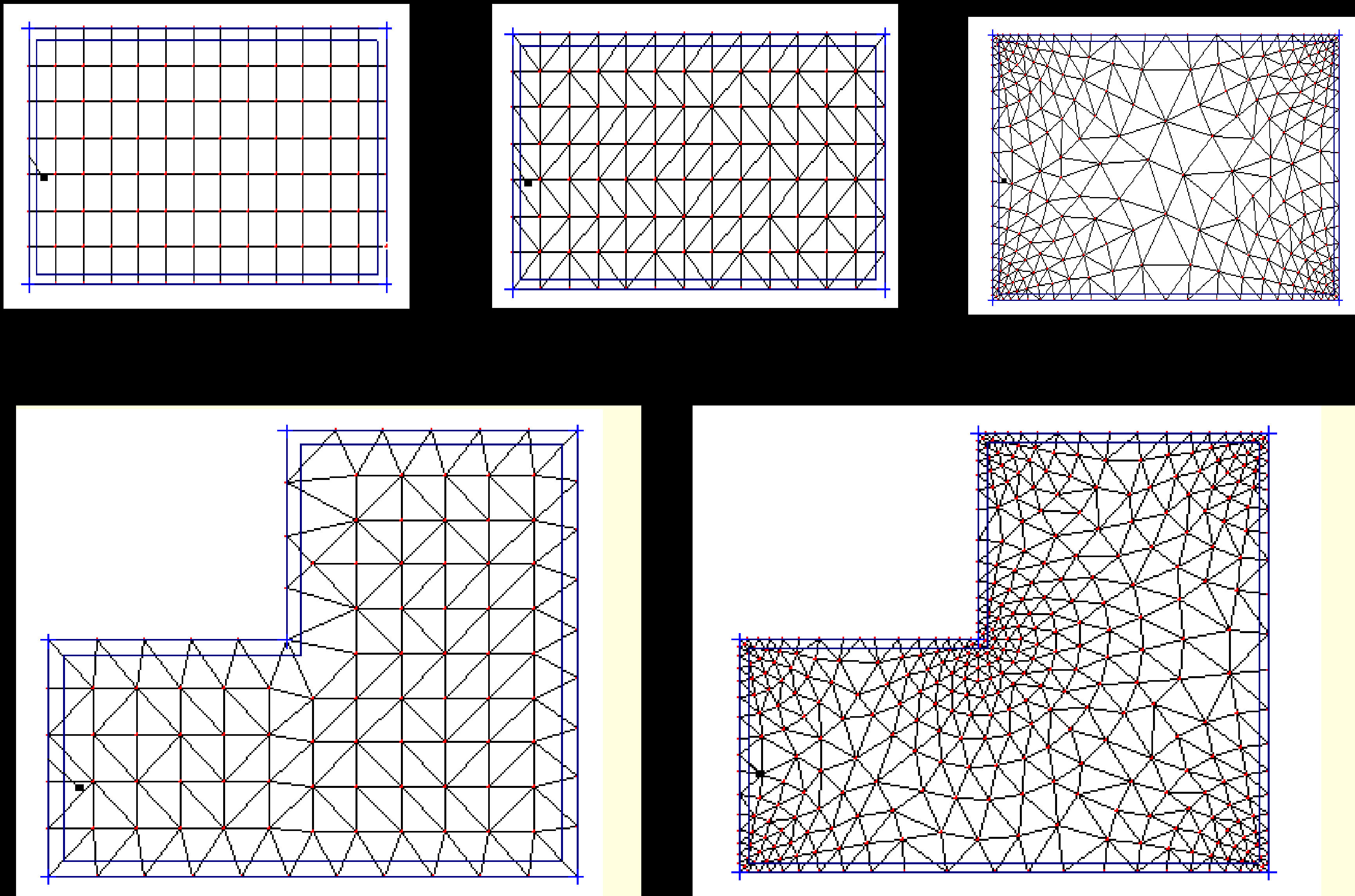
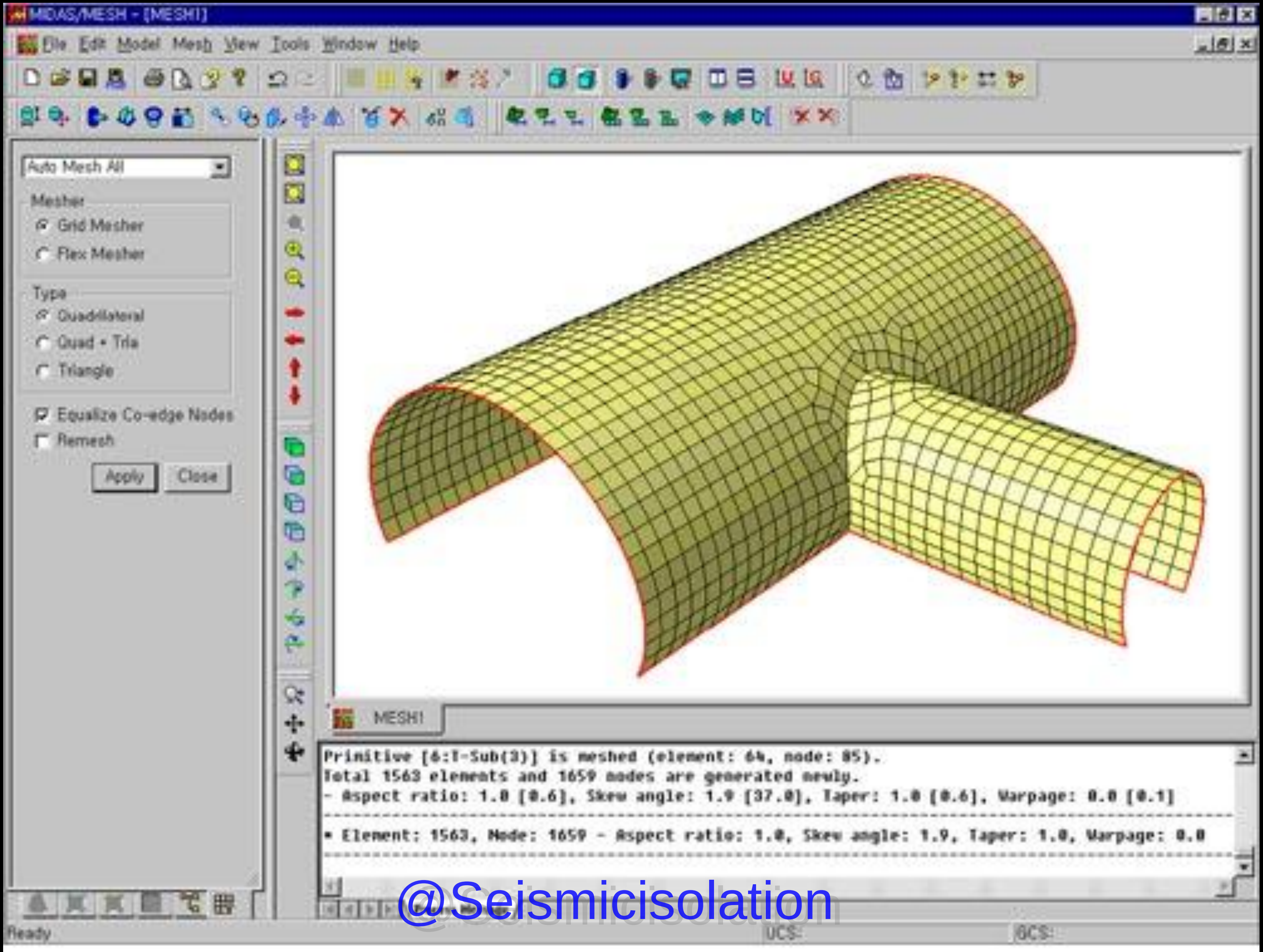
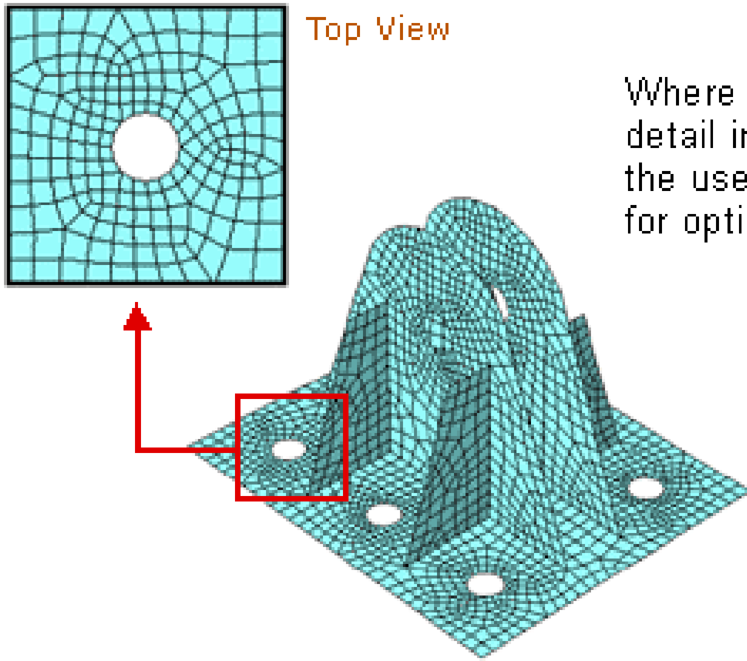


PLATE ELEMENTS

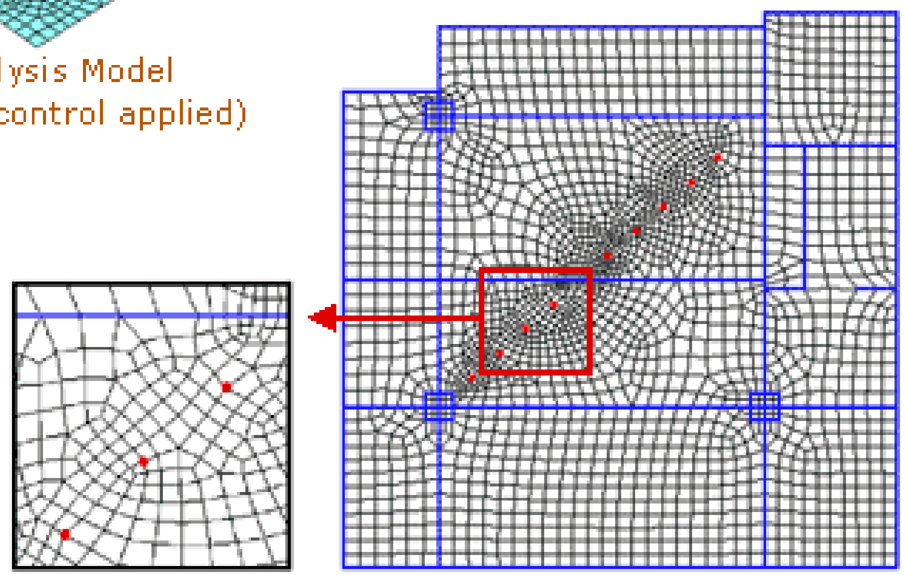

MIDAS/MESH - [MESH1]
File Edit Model Mesh View Tools Window Help
Auto Mesh All
Mesher
Grid Mesher
Flex Mesher
Type
Quadrilateral
Quad + Tria
Triangle
Equalize Co-edge Nodes
Remesh
Apply
Close
MESH1
Primitive [6:T-Sub(3)] is meshed (element: 64, node: 85).
Total 1563 elements and 1659 nodes are generated newly.
- Aspect ratio: 1.0 [0.6], Skew angle: 1.9 [37.0], Taper: 1.0 [0.6], Warpage: 0.0 [0.1]
* Element: 1563, Node: 1659 - Aspect ratio: 1.0, Skew angle: 1.9, Taper: 1.0, Warpage: 0.0
Ready
UCS:
GCS:

Where certain parts of a structure require detail investigation MESH enables the user to increase the mesh density for optimum meshing.



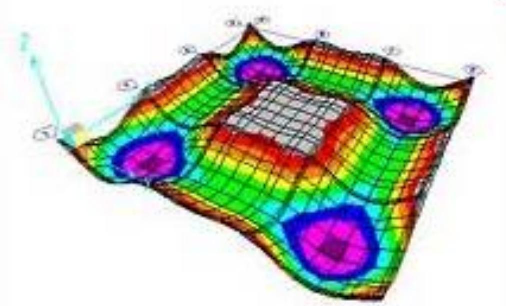
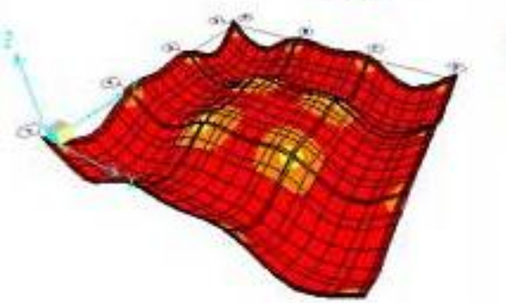
Base Plate Analysis Model
(Mesh density control applied)

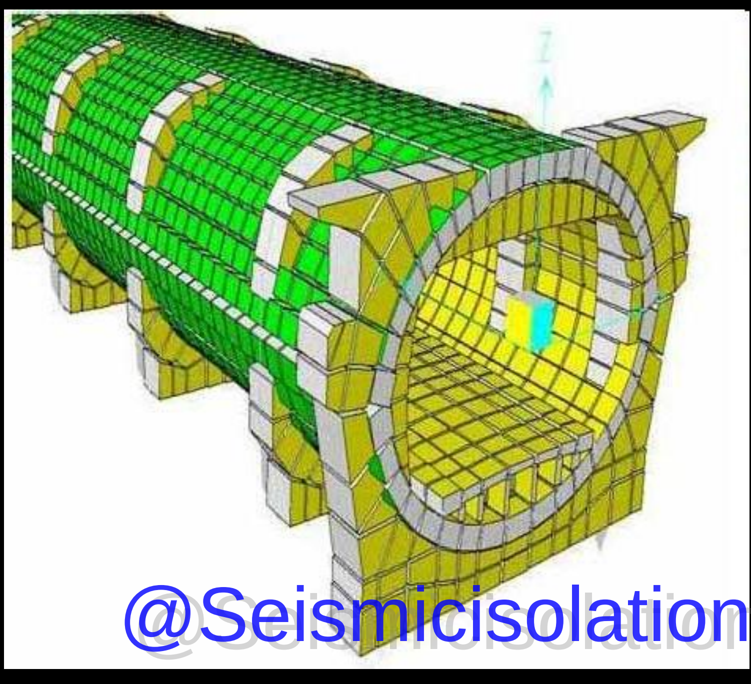
Locations of applied
walking loads

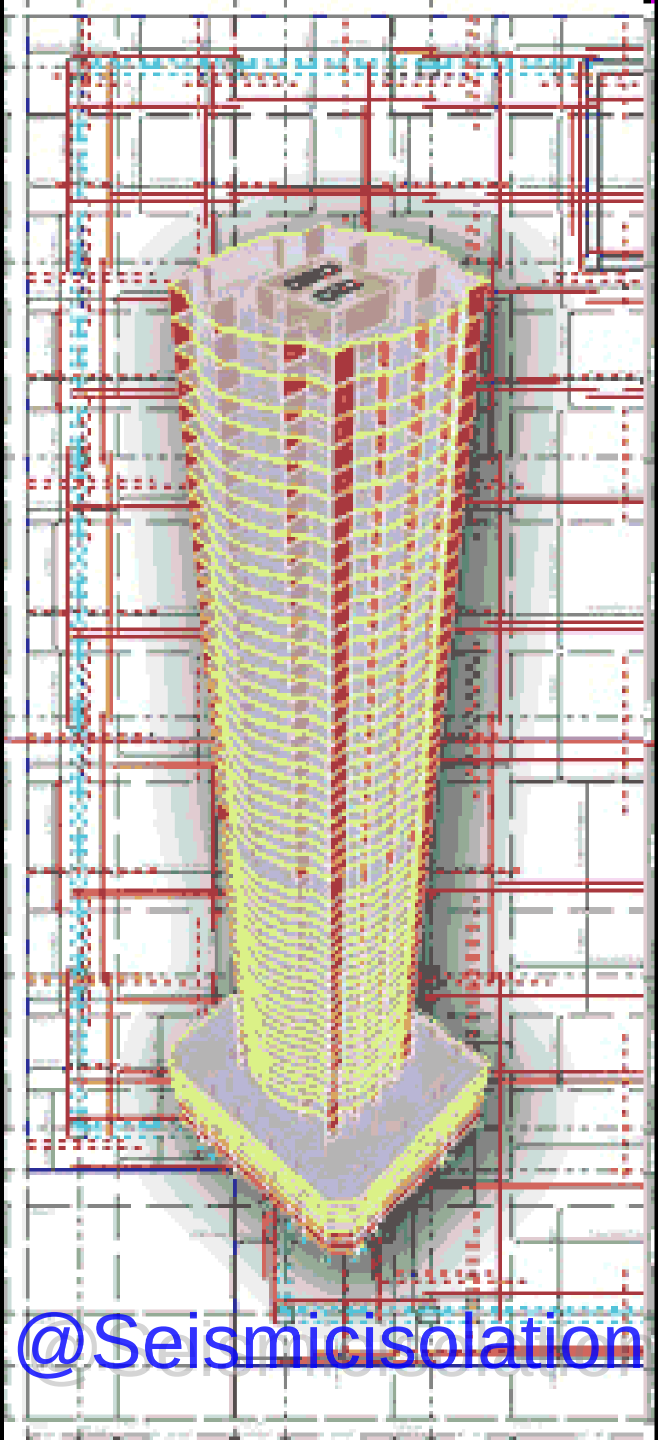


Floor structure model for vibration
analysis due to walking loads
(Mesh density control applied to nodes)







@Seismicisolation

The following typical principal steps for performing a finite element analysis, considering only basic, simple structures for preliminary design purposes, are:

CREATE the finite element model by dividing the structure into finite elements – idealize the structure

DEFINE
- GRID SIZE and UNITS
- MEMBER ELEMENTS and NODES: determine the type of elements which are appropriate for you model.
  In SAP2000 you only define elements - all nodes needed by the elements are automatically generated.
  - **Line elements**: truss, beam (beam column), cable
  - **Surface elements**: behavior (membrane-, plate-, shell-elements), shape (quadrilateral, triangular)
  - **3D elements**
- MATERIAL properties: steel, reinforced concrete, other
- MEMBER SECTIONS: geometric properties, shapes from library
- LOADING conditions: static load cases, load combinations

DRAW GEOMETRY: define the geometry of structure, i.e. the location of members on grid

ASSIGN properties, loading, and boundary conditions to the structure
- MATERIAL
- MEMBER SECTIONS
- EXTERNAL SUPPORTS: free, fixed, pinned, roller, spring
- NODAL RESTRAINTS: FRAME END RELEASES, constraints, springs
- EXTERNAL FORCES/DISPLACEMENTS on nodes and members
- DIAPHRAGM CONSTRAINTS
- MEMBER CHARACTERISTICS: clear length, unbraced length
- etc.

CHECK INPUT DATA

ANALYZE: select analysis type and run analysis
- DISPLAY the model (e.g. reactions, forces, stresses, deformations and animations)
- EVALUATE and CHECK important results

DESIGN
- MEMBER PROERTIES
- MODIFY MEMBER PROPERTIES

RE-ANALYZE and RE-DESIGN

PRINTING and PLOTTING of results

Structure software helps students to **VISUALIZE** the building as an assembly of,

- **linear members** (e.g. beams, columns, arches, cables),
- **planar members** (e.g. walls, slabs, shells, flexible membranes), or
- **spatial members** (e.g. solids).

1. First students have to **DEFINE**:

geometry
material
member types
member sections
static load cases
load combinations

2. Then students set up the **mathematical model for the building support structure** by **DRAWING** the geometry of the structure and then to

3. ASSIGN the

member types and *sections*
external support joints
possibly *constraints*
frame end releases (internal member joints)
load types

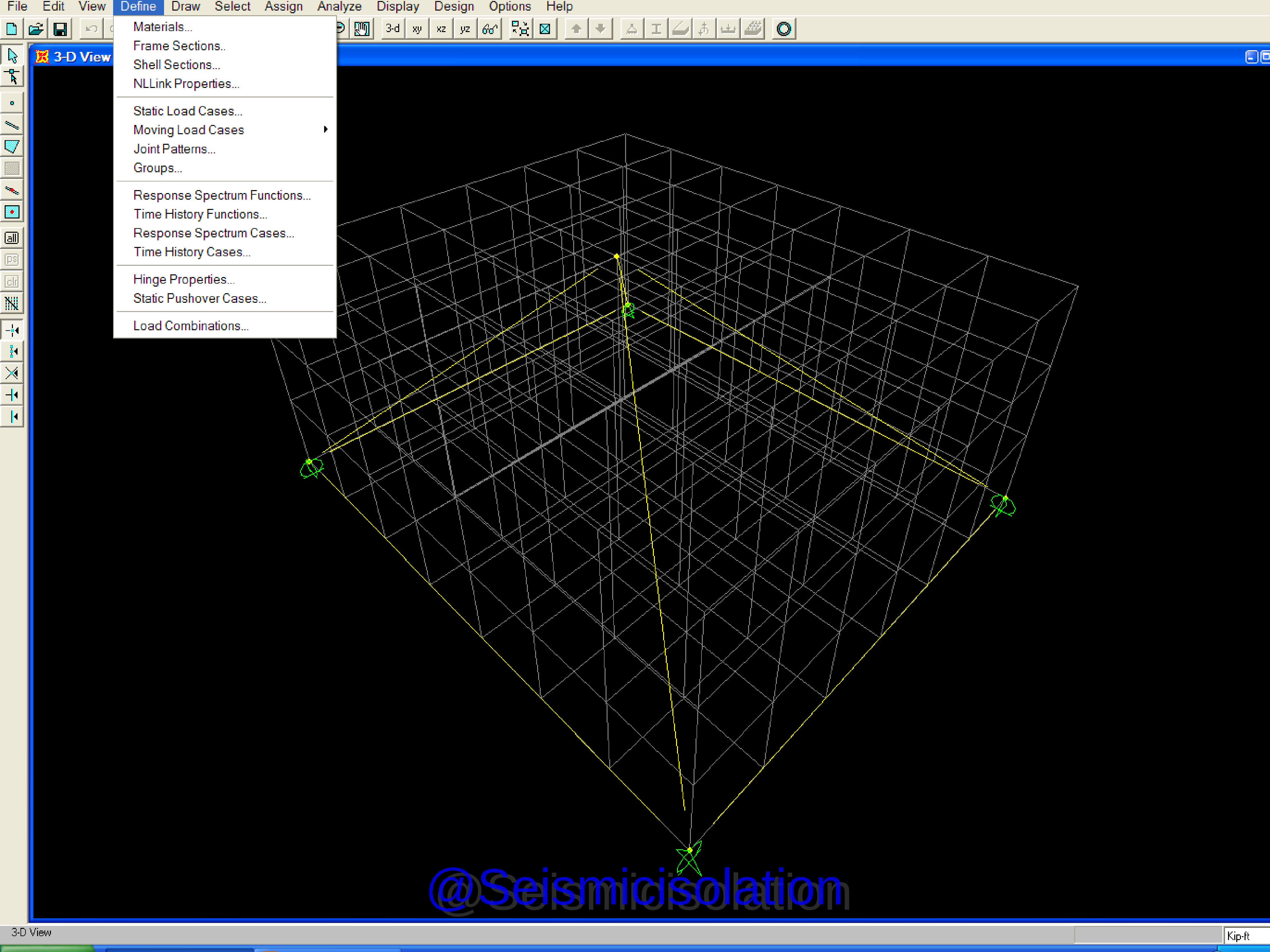
If you are upgrading from a previous version of SAP2000 (Version 11 and earlier), you should be aware that there has been a significant change in nomenclature regarding loading:

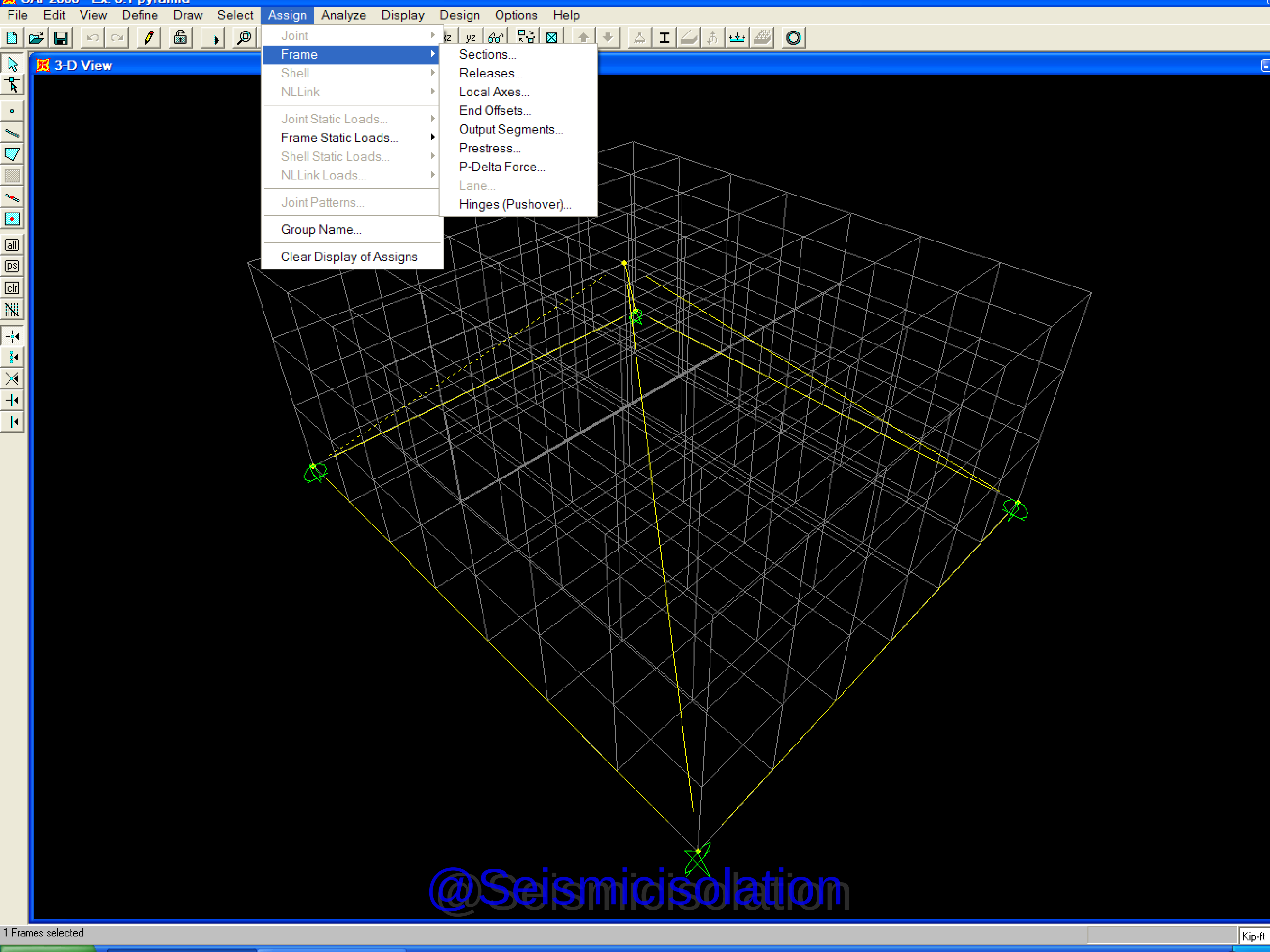
- The term "**load pattern**" replaces "**load cases**" of previous versions.

- The term "**load cases**" replaces "**analysis cases**" of previous versions.

- The term "**load combinations**" has not changed.

@Seismicisolation


File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-D View
Materials...
Frame Sections...
Shell Sections...
NLLink Properties...
Static Load Cases...
Moving Load Cases
Joint Patterns...
Groups...
Response Spectrum Functions...
Time History Functions...
Response Spectrum Cases...
Time History Cases...
Hinge Properties...
Static Pushover Cases...
Load Combinations...
3-D View
Kip-ft


File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-D View
Joint
Frame
Shell
NLLink
Joint Static Loads...
Frame Static Loads...
Shell Static Loads...
NLLink Loads...
Joint Patterns...
Group Name...
Clear Display of Assigns
Sections...
Releases...
Local Axes...
End Offsets...
Output Segments...
Prestress...
P-Delta Force...
Lane...
Hinges (Pushover)...
@Seismicisolation
1 Frames selected
Kip-ft

To model a structure the designer must be familiar with the following criteria:

- **building structure primary function**
- **types of loads**
- **building materials** (i.e. important material characteristics)
- **structural members**
- **member connections**
- **structural systems**

COMPUTER-BASED ANALYSIS AND DESIGN OF STRUCTURES

INTRODUCTION

A. MODELING THE STRUCTURE
B. CHECK INPUT DATA
C. ANALYSIS
D. EVALUATE and CHECK RESULTS
E. DESIGN
F. MODIFY MEMBER PROPERTIES
G. RE-ANALYZE and RE-DESIGN
H. COMPUTER OUTPUT: display results (printing and plotting)

A. MODELING THE STRUCTURE

A typical computer input of mathematical models is presented to demonstrate the decomposition of the structure into its constituent parts. It is assumed for ordinary conditions that the original geometry of the structure does not change and that it remains linearly elastic. The initial modeling of the structure is defined by *geometry*, *members* and *nodes*, *material* and *member properties*, *supports*, *loads* and *type of analysis*.

Building structures are three-dimensional where the applied loads flow along the members and joints to the external supports usually the foundations. Since the applied loads cause internal forces in the members and joints, one can visualize the support structure as replaced by corresponding, complex three-dimensional force systems. In other words under the load action the support structure displaces and changes its geometry thereby causing deformations in the members which, in turn correspond to internal forces.

Ordinary buildings can generally be considered as an assembly of independent horizontal and vertical planar structures (at least for preliminary design purposes) so that force systems can be treated as two-dimensional or coplanar. Therefore, typically, the first approach towards modeling the structure will consist of a simplified version of the overall structure or it will be only a portion of it. The investigation of the various structure systems, in this context, is primarily concerned with planar structures and force systems rather than modeling the true spatial building structure.

- It is not the intention here to introduce a detailed discussion of the finite element method, since you will not write your own finite element programs but only use finite element programs. **It must be emphasized that a precise understanding of the finite element is not a prerequisite for working with structure finite element programs**

In other words, it is important not to see the finite element analysis as a numerical technique involving matrix mathematics to solve large sets of simultaneous algebraic equations, but to see it as an abstract model that makes it possible to learn about the behavior of structures in general. **To see it also as a powerful educational tool since it has the unique feature of integrating all the structural concepts of *loads, language of forces, geometry, structure systems, member behavior, stability of building, materials, codes*, etc., all necessary components of design, into one system. In the traditional approach, all those topics are treated more or less in separate courses.**

- Finite element analysis makes indeterminate structures now accessible to general designers during the preliminary design stage. To study the effect of *stiffness* (e.g. member size) on force flow, as well as the effect of change of geometry, member arrangement and support locations, is most important for developing a feel of structure. The same is true for studying the effect of the type of member assembly with respect to local and global stability by considering rigid joints and/or bracing. In addition, to become aware of **structural systems** (thereby developing a sense of what is possible), as well as a learning model that decomposes the building structure into geometry, its major parts of: ***elements*** and ***nodes*** *or* ***joints***, ***support joints***, ***active member joints***, ***material***, and ***applied loads*.**
- The primary difficulty in using structural computer programs is to not truly understand the physical action and boundary conditions as well as the behavior of the structure elements of the actual structure, which is especially true for the behavior of the joints. The true constraints (i.e. restraint to movement) that each member exerts on the adjacent ones, can only be approximated. It is apparent, **that the mathematical model may not represent the reality and true character of the structure at all.**

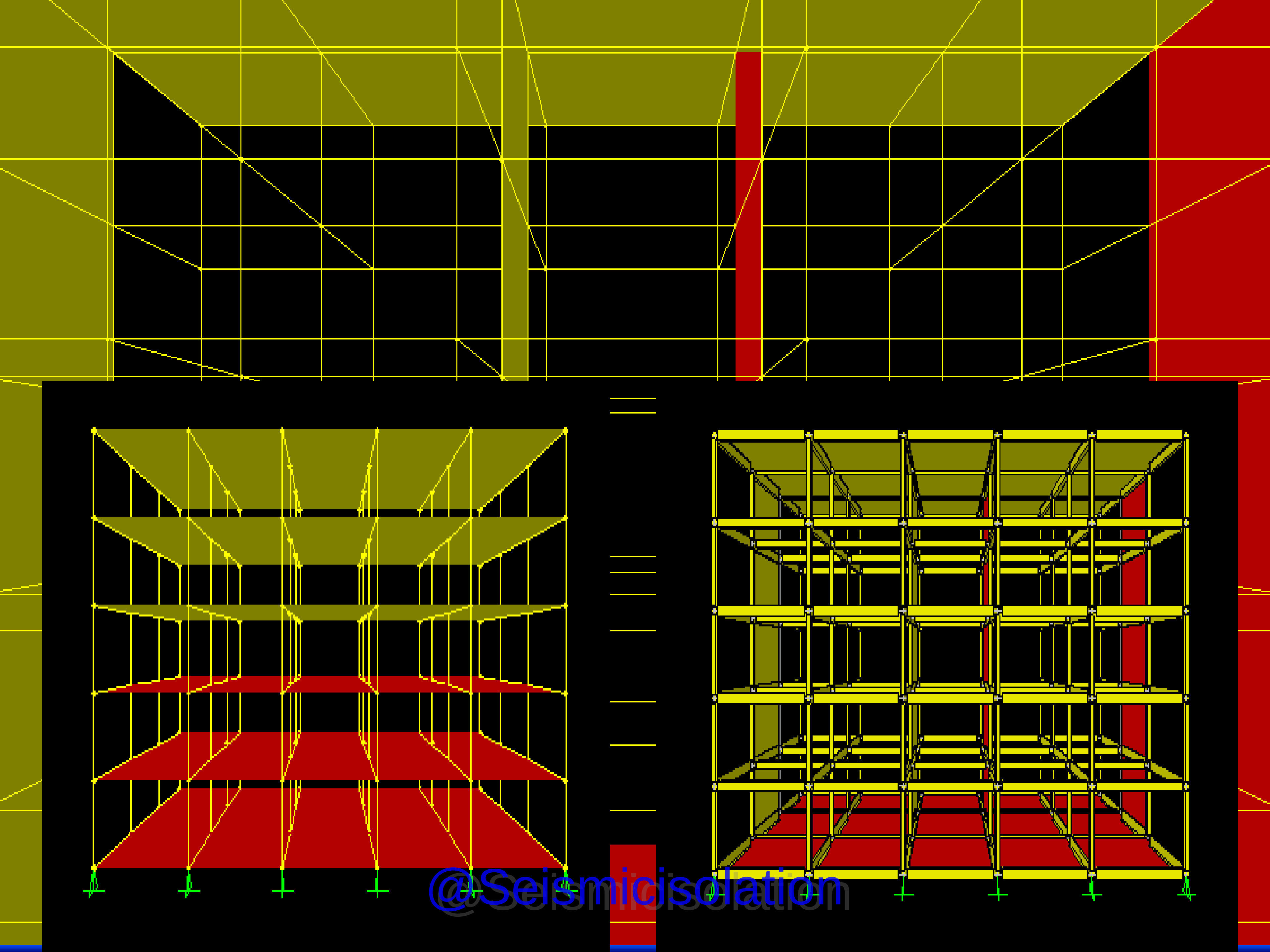
Modeling the structure involves the creation of an abstract mathematical model of the building structure and the loads acting on it. A building structure is defined by the

a. GRID SIZE and UNITS
   GEOMETRY: define the geometry of structure, i.e. the location of members on grid
b. MATERIAL properties: aluminum, steel, reinforced concrete, etc.
c. MEMBER ELEMENTS and NODES (line elements, surface elements, 3D elements): in SAP2000 you only define elements - all nodes needed by the elements are automatically generated.
d. MEMBER CHARACTERISTICS: clear length, unbraced length
e. MEMBER SECTIONS: geometric properties, shapes from library
f. EXTERNAL FORCES/DISPLACEMENTS on nodes and members
g. LOADING conditions: static load cases, load combinations
h. EXTERNAL SUPPORTS: free, fixed, pinned, roller, spring
i. NODAL RESTRAINTS: FRAME END RELEASES, constraints, springs
j. DIAPHRAGM CONSTRAINTS
k. etc.

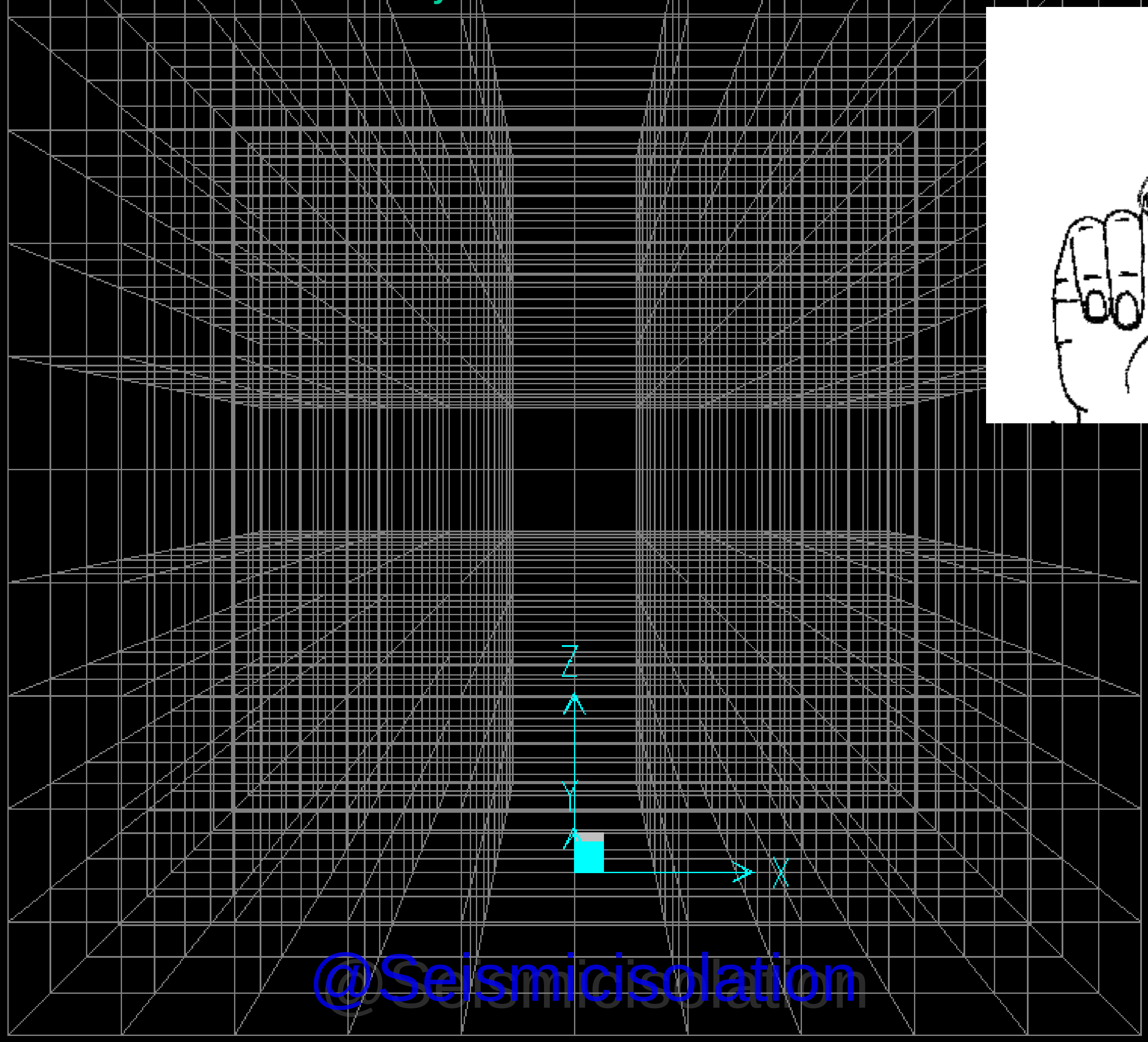
GEOMETRY

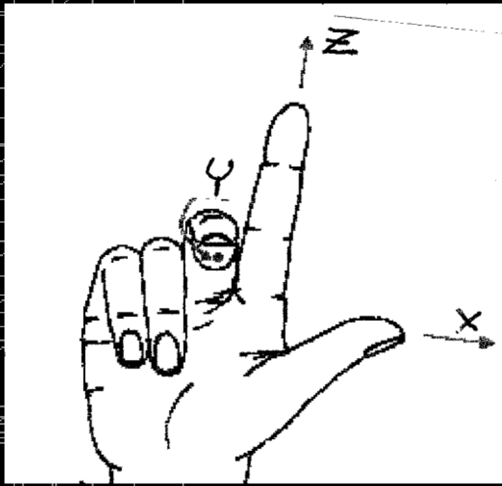
The geometry of structures may be generated in computers graphically and/or using spreadsheets, or importing DXF files. Depending on the structural engineering software, however, many structure types may be automatically generated with the assistance of templates. Most computer software packages provide a library of 2D and 3D structure layouts that include typical systems such as various types of beams, trusses, frames, and grids. Some software include in addition more specialized structures such as cables, shear walls, floor framing, as well as particular types of space frames, shells, geodesic and latticed domes, pre-stressed tensile membranes, and so on.

The geometry of the systems may be predetermined for line elements (e.g. as polygonal, circular, elliptical, parabolic, spiral), planar elements (possibly as derived for curvilinear surfaces from sphere, cone, cylinder, tube, hyperbolic paraboloid, torus, or other behavioral phenomena), and for spatial elements or blocks (as defined for instance for ordinary conditions by solid geometry for basic shapes of polyhedrons, prisms, pyramids, cylinders, cones, spheres, etc.). Large models can be generated automatically using templates. The models can then be sculptured with on screen editing to satisfy specific situations.



In general, a building structure is located in a spatial grid and defined by a single **GLOBAL** coordinate system **X-Y-Z** that follows the right-hand rule, where the **Z-X plane** is the vertical plane and the **Z-axis** the vertical axis with +Z being upward; the **X-Y plane** is horizontal. Each part of the structure (e.g. joint, element) has its own **LOCAL** coordinate system **1-2-3**.

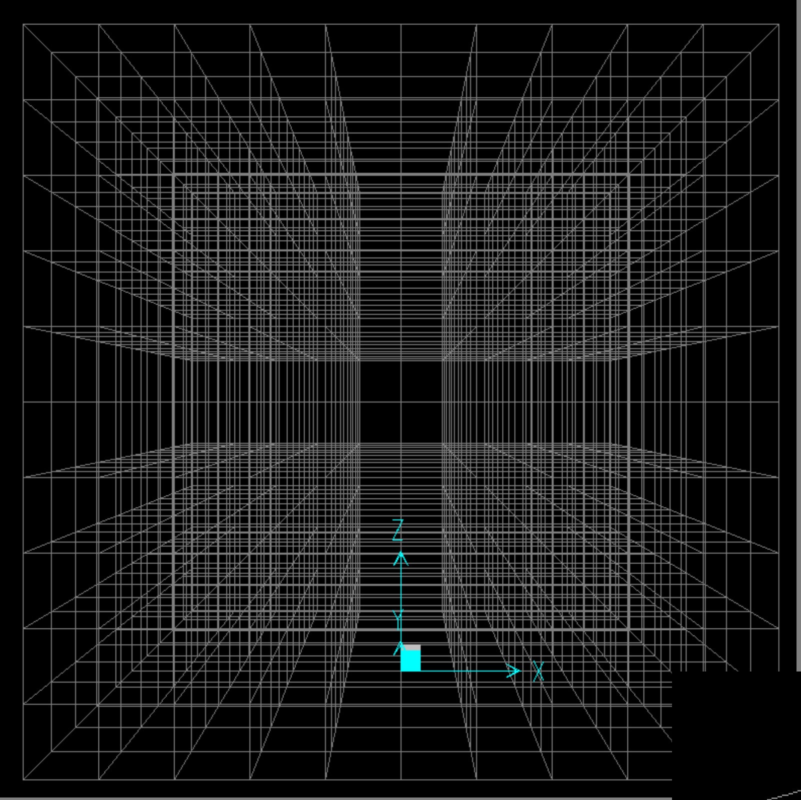


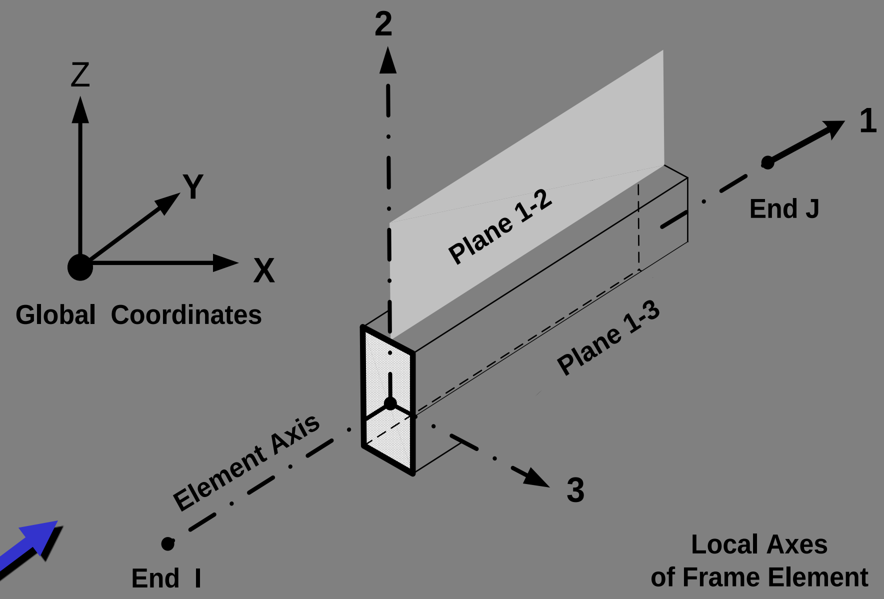


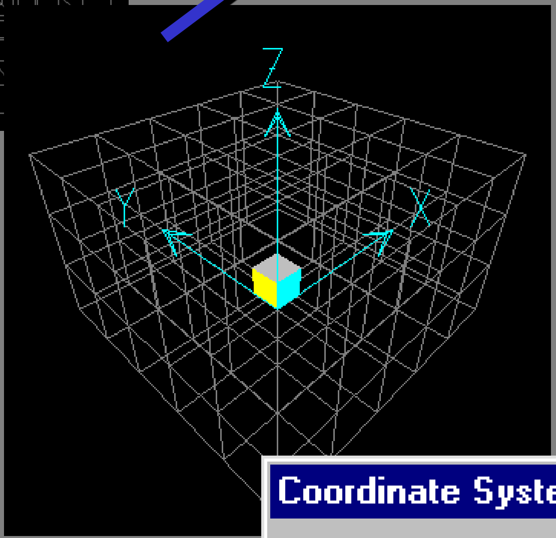
The building structure is located in a spatial grid and defined by a single **GLOBAL** coordinate system **X-Y-Z**, where the Z-X plane is the vertical plane and the Z-axis the vertical axis with +Z being upward; the X-Y plane is horizontal.

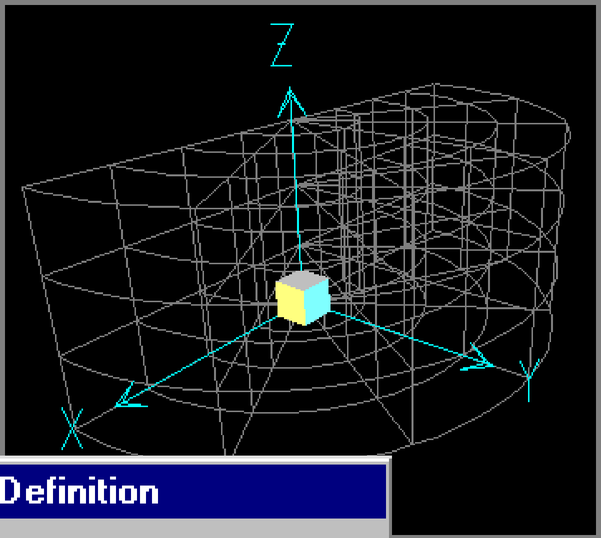
Each part of the structure (e.g. joint, element) has its own **LOCAL** coordinate system **1-2-3**. In other words, each frame element has its own local coordinate system for defining section properties and loads, and for interpreting output data.

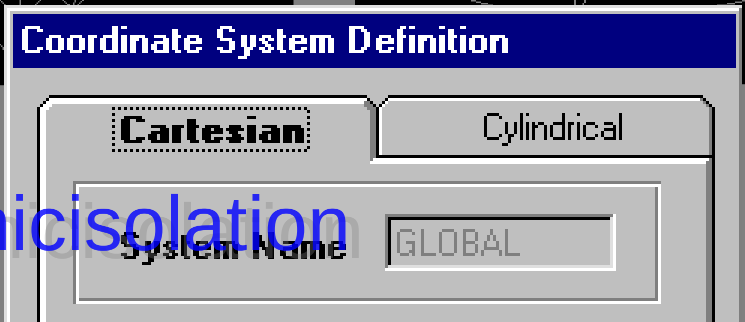
- The **joint local coordinate system** is normally the same as the global X-Y-Z coordinate system.
- For the elements, one of the **element local axes** is determined by the geometry of the individual element; the orientation of the remaining two axes is defined by specifying a single angle of rotation. For frame elements, for example, the **local axis 1** is always the longitudinal axis of the element with the positive direction from **I** to **J**. The default orientation of the **local 1-2 plane** is taken to be vertical (i.e. parallel to the Z-axis). The **local 3-axis** is always horizontal (i.e. lies in the X-Y plane).







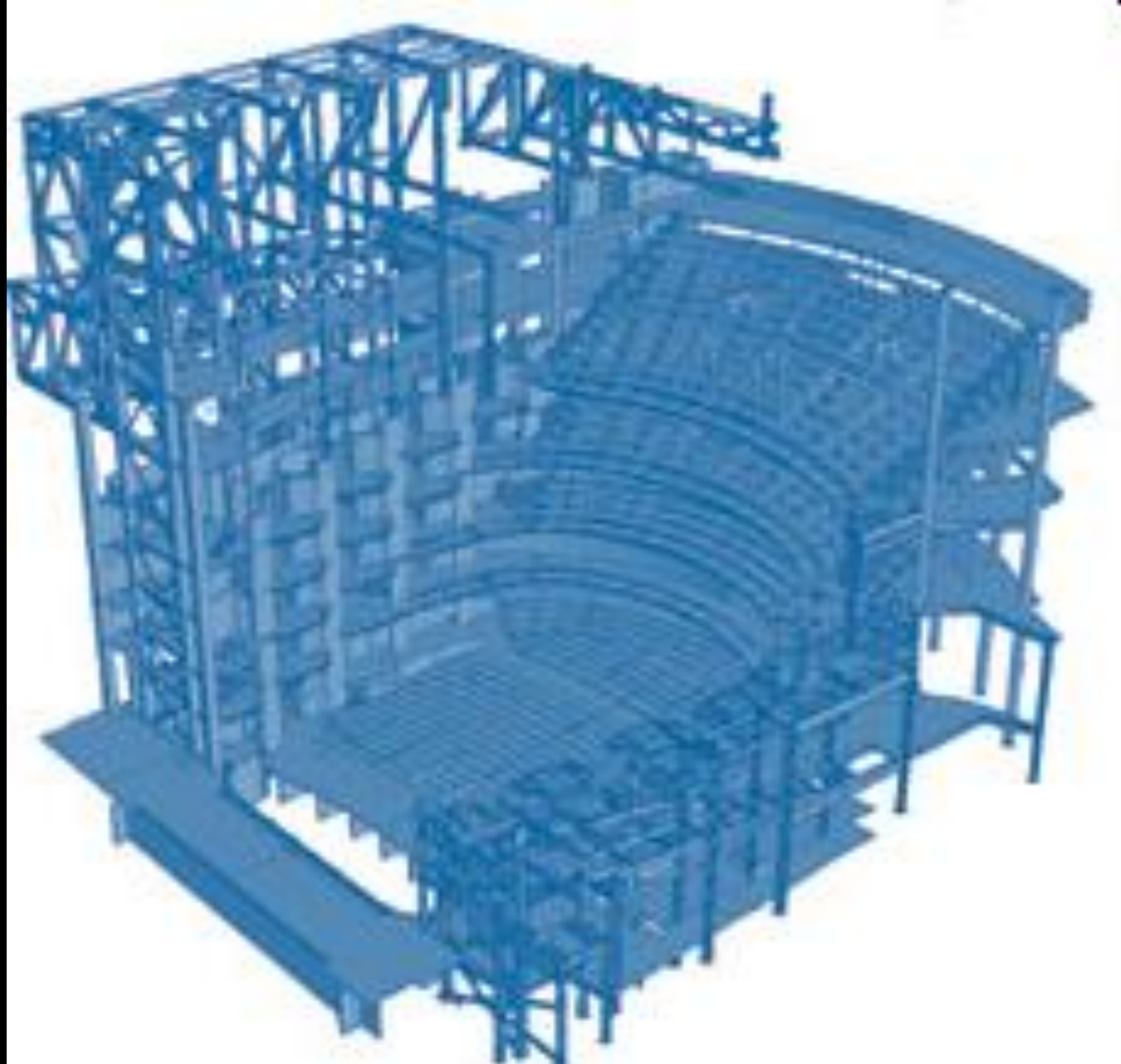




Guides and Grids

- Accurate dimensioning with guidelines and snapping

- Multiple rectangular and cylindrical coordinate systems with flexible grid systems



Structural design is gradually shifting from 2D drafting towards 3D modelling. Tekla has developed an innovative solution for Structural Building Information Modelling, a subset of the commonly used concept Building Information Modelling (BIM).

MATERIAL

The typical material properties identified by structural engineering software are:

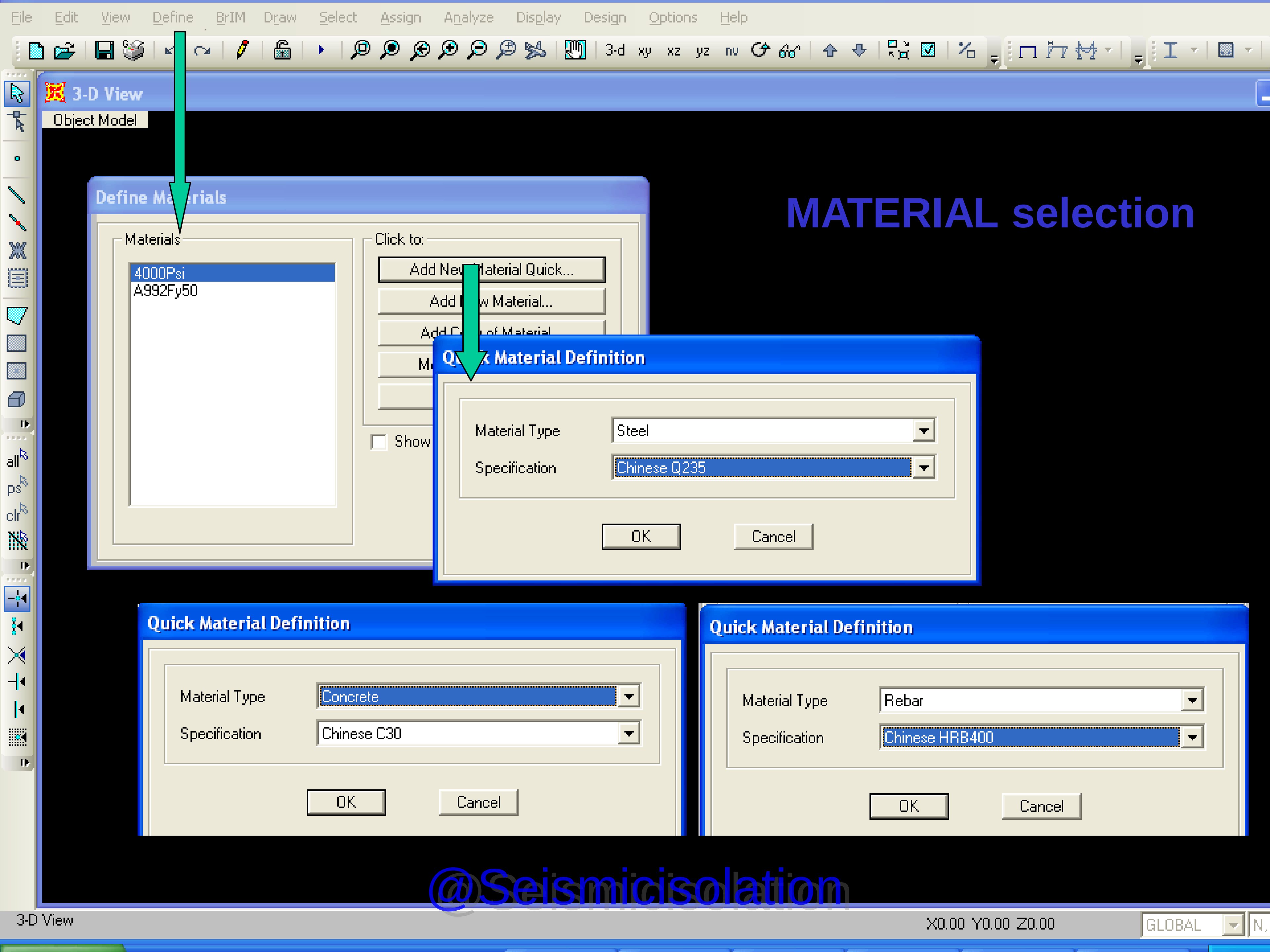
Specific weight or weight density (weight per unit volume)
Mass density (mass per unit volume)
Modulus of elasticity: *E*
Shear modulus: *G*
Poisson's ratio: ν (N_u)
Coefficient of thermal expansion: ε_t or μ (M_u)
Material strength, or yield strength for steel (f_y)
Type of material: *isotropic* or *orthotropic*
For orthotropic materials the values for *E*, *G*, ν, and μ must be defined for the three local axes of the material.

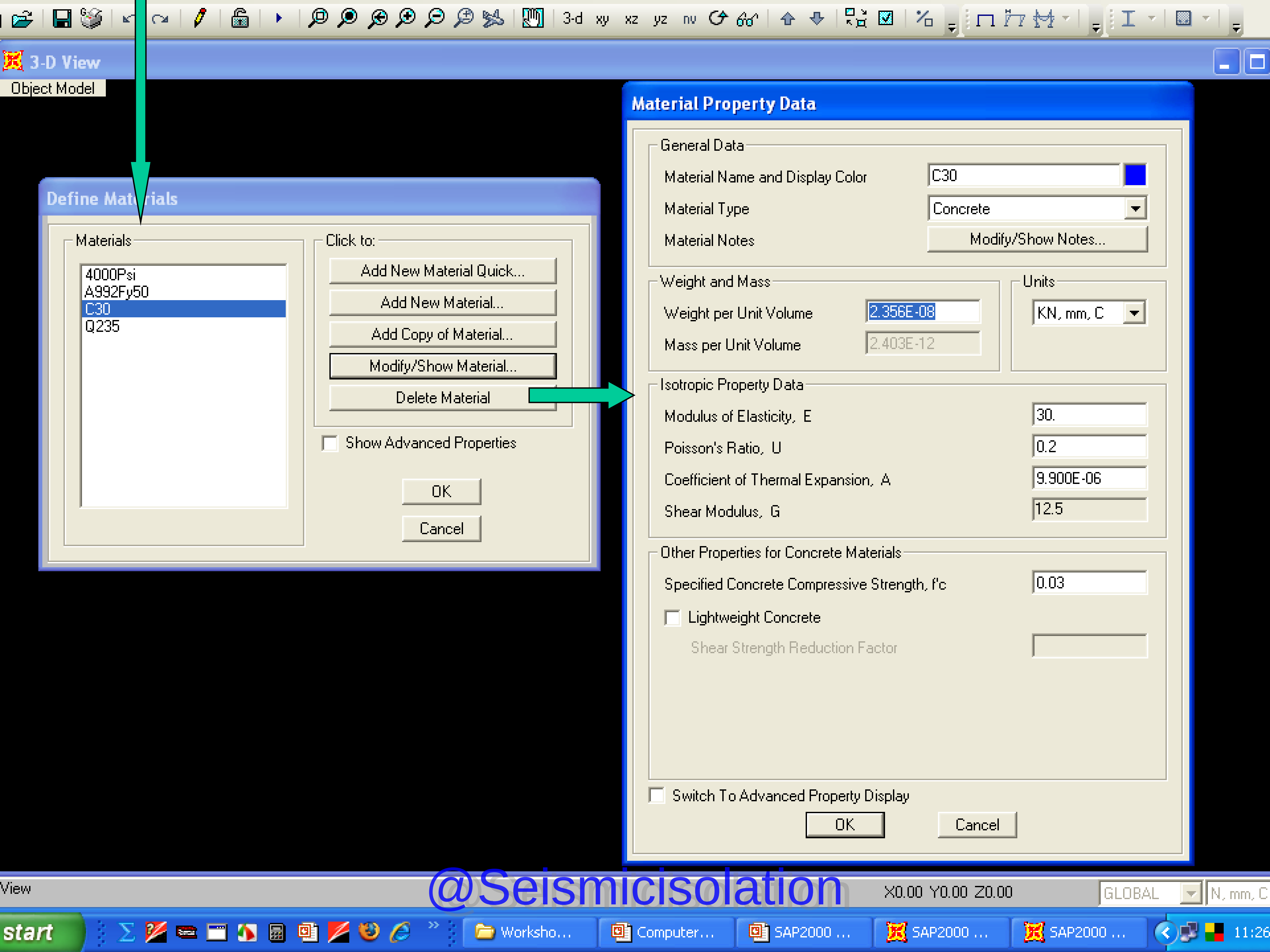
Program defined materials such as: aluminum, *steel* and *reinforced concrete;* possibly *user defined material* for *wood* and masonry but that depends on the structural engineering software

Typical allowable stresses for ordinary materials:

	Permissible stress
	N/mm^2 (MPa) ksi
Steel	**150** **22**
Aluminium	**100** **14.5**
Concrete	**10** **1.5**
Wood	**7** **1**

@Seismicisolation


File Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-D View
Object Model
MATERIAL selection
Define Materials
Materials
4000Psi
A992Fy50
Click to:
Add New Material Quick...
Add New Material...
Quick Material Definition
Material Type
Steel
Specification
Chinese Q235
OK
Cancel
Quick Material Definition
Material Type
Concrete
Specification
Chinese C30
OK
Cancel
Quick Material Definition
Material Type
Rebar
Specification
Chinese HRB400
OK
Cancel
@Seismicisolation
3-D View
X0.00 Y0.00 Z0.00
GLOBAL


3-D View
Object Model
Define Materials
Materials
4000Psi
A992Fy50
C30
Q235
Click to:
Add New Material Quick...
Add New Material...
Add Copy of Material...
Modify/Show Material...
Delete Material
Show Advanced Properties
OK
Cancel
Material Property Data
General Data
Material Name and Display Color
C30
Material Type
Concrete
Material Notes
Modify/Show Notes...
Weight and Mass
Weight per Unit Volume
2.356E-08
Mass per Unit Volume
2.403E-12
Units
KN, mm, C
Isotropic Property Data
Modulus of Elasticity, E
30.
Poisson's Ratio, U
0.2
Coefficient of Thermal Expansion, A
9.900E-06
Shear Modulus, G
12.5
Other Properties for Concrete Materials
Specified Concrete Compressive Strength, f'c
0.03
Lightweight Concrete
Shear Strength Reduction Factor
Switch To Advanced Property Display
OK
Cancel
View
X0.00 Y0.00 Z0.00
GLOBAL
N, mm, C
start
Worksho...
Computer...
SAP2000 ...
SAP2000 ...
SAP2000 ...
11:26

MEMBER ELEMENTS

In the mathematical model the building structure is first modeled purely from a geometrical point of view using line elements, planar elements and possibly three-dimensional elements; these elements represent only geometry, they have no other properties (e.g. strength) assigned to them. For example, a building structure that consists of an assembly of **linear bar** or **frame elements** (e.g beams, columns, arches, cables), the frames are idealized as *line diagrams* where the lines are located at the geometric axes of the structural elements. For the situation where the building structure is made up of **planar elements** (e.g. walls, slabs, rigid shells, flexible membranes), the mathematical model treats them as surface elements located at the mid-surface of the real structures. **Three-dimensional** or **solid elements** are used to model spatial structures and solids. Each **joint element**, in the mathematical model, has its own symbol as is discussed in the next picture.
The element types, in turn, represent physical action in the real member as indicated in Fig. 3.2, such as for,

LINE ELEMENTS: **axial systems** (e.g. truss, tie, cable), **flexural systems** (e.g. beams), **axial-flexure systems** (e.g. beam-columns, frames, arches),

SURFACE ELEMENTS: **axial systems** (e.g. tensile membranes , **axial-shear** systems (e.g. thin shells), **axial-flexure systems** (e.g. plates, walls, slabs, thick shells).

In SAP2000, the physical members are represented by objects, where the geometry of the object corresponds to that of the member. The typical objects are as follows:

- **Point objects**: (1) joint objects, which are automatically created at the ends or corners of objects, (2) support objects
- **Line objects**: (1) frame/cable/tendon objects, (2) link objects
- **Area objects**: shell type objects
- **Solid objects**

By assigning properties and loads to the object, the physical member is modeled. SAP2000 converts automatically the object-based model into a finite element-based model that is used for analysis.

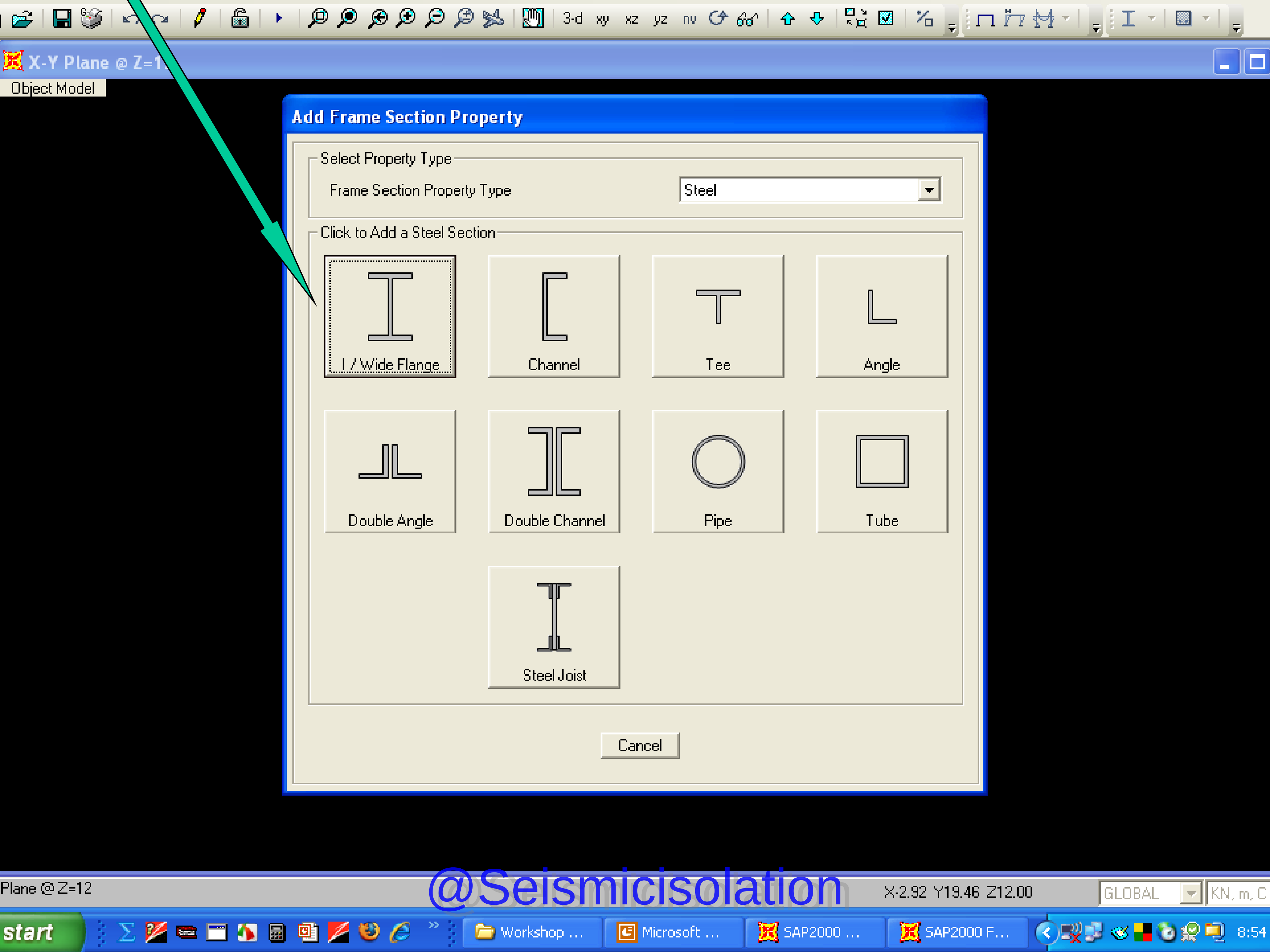
FRAME MEMBER SECTIONS

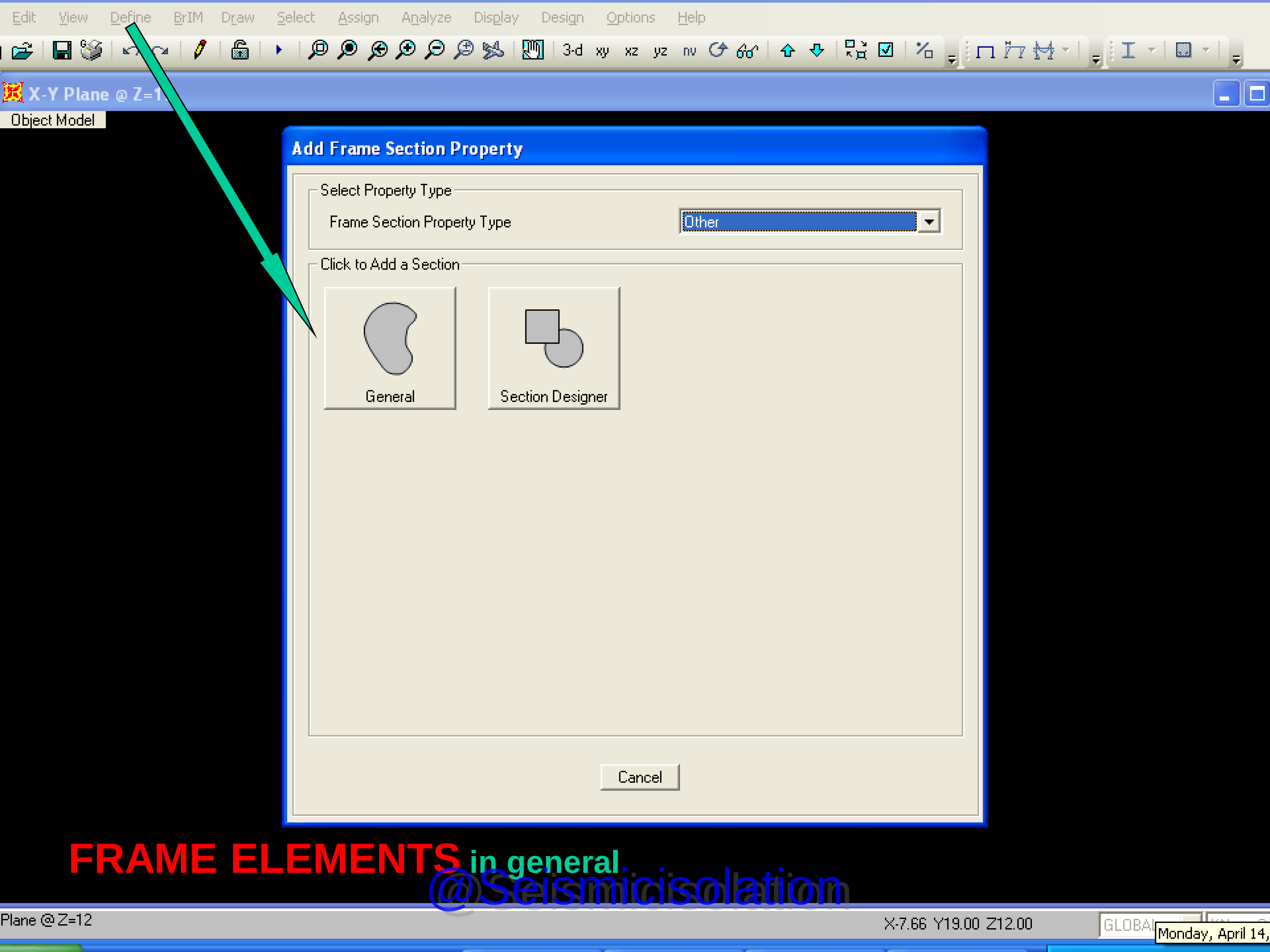
Frame **or** ***shell sections*** **(section properties such as outline, reinforcement, cross-sectional area, moment of inertia, etc.)**

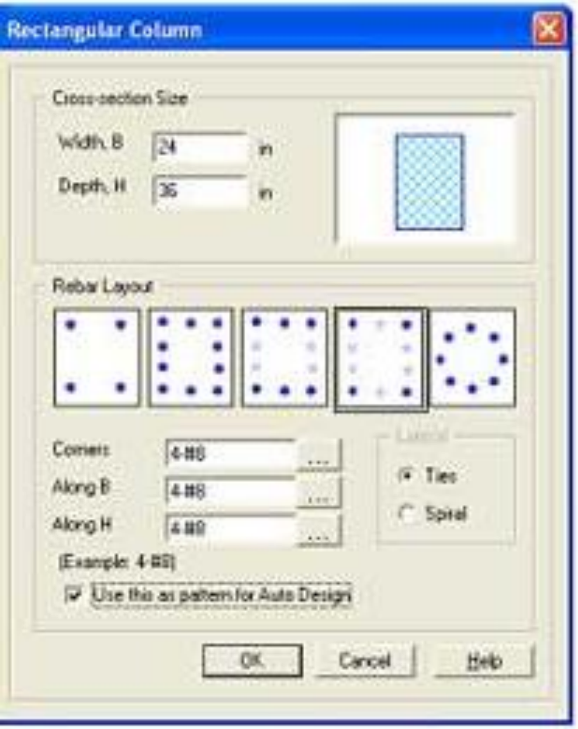
Standard structural steel shapes are the wide-flange shape (*W*), American standard beams (*S*), miscellaneous shapes (*M*), bearing pile shapes (*HP*), channel shapes (*C, MC*), equal leg and unequal leg angles (*L*), structural tees split from *W*, *M* and *S* shapes (*WT, MT* and *ST*), round steel pipe and structural tubing (*TS*).

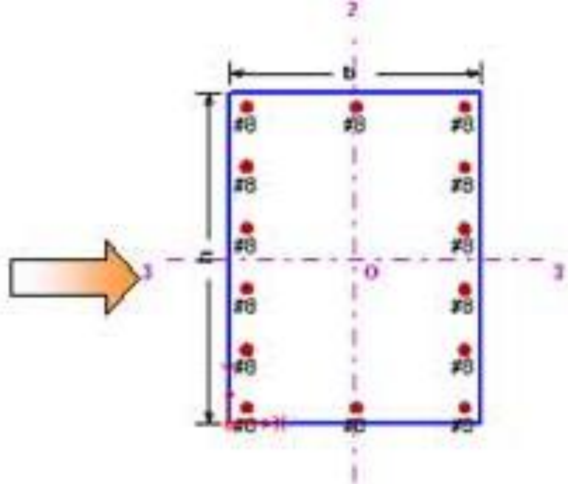
The typical geometric section properties for prismatic (i.e. not tapered members) used in structural engineering software are:

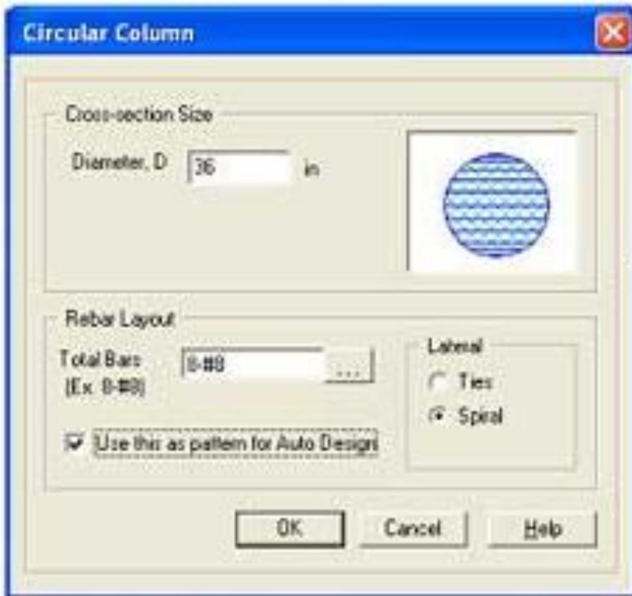
- **section dimensions**
- **cross-sectional area (A)**
- **moment of inertia (I_x, I_y)**
- **section modulus (S_x, S_y)**
- **radius of gyration (r_x, r_y, r_z)**
- **plastic section modulus (Z_x, Z_y)**
- **shear area (A_{sx}, A_{sy})**
- **torsional constant (J)**

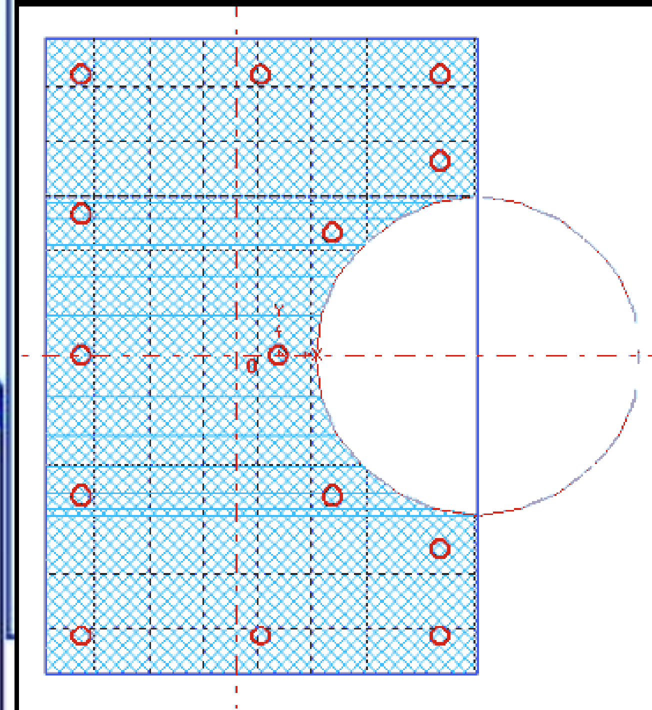

X-Y Plane @ Z=1
Object Model
Add Frame Section Property
Select Property Type
Frame Section Property Type
Steel
Click to Add a Steel Section
I / Wide Flange
Channel
Tee
Angle
Double Angle
Double Channel
Pipe
Tube
Steel Joist
Cancel
Plane @ Z=12
@Seismicisolation
X-2.92 Y19.46 Z12.00
GLOBAL
KN, m, C
start
Workshop ...
Microsoft ...
SAP2000 ...
SAP2000 F...
8:54

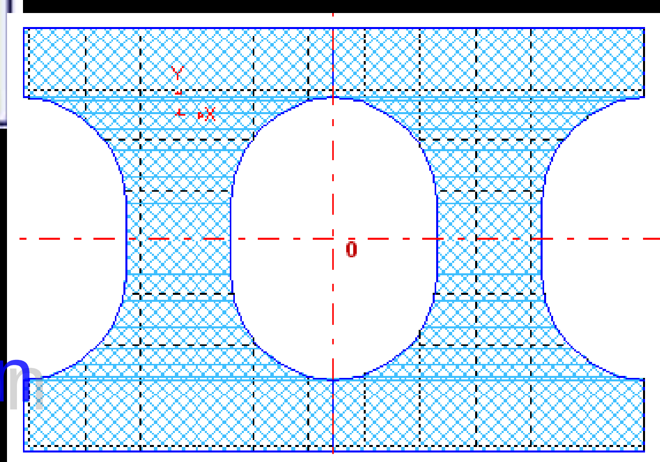

Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
X-Y Plane @ Z=1
Object Model
Add Frame Section Property
Select Property Type
Frame Section Property Type
Other
Click to Add a Section
General
Section Designer
Cancel
FRAME ELEMENTS in general.
@Seismicisolation
Plane @ Z=12
X-7.66 Y19.00 Z12.00
GLOBAL
Monday, April 14,


Rectangular Column
Cross-section Size
Width, B
24
in
Depth, H
36
in
Rebar Layout
Corners
4-#8
Along B
4-#8
Along H
4-#8
(Example: 4-#8)
Ties
Spiral
Use this as pattern for Auto Design
OK
Cancel
Help

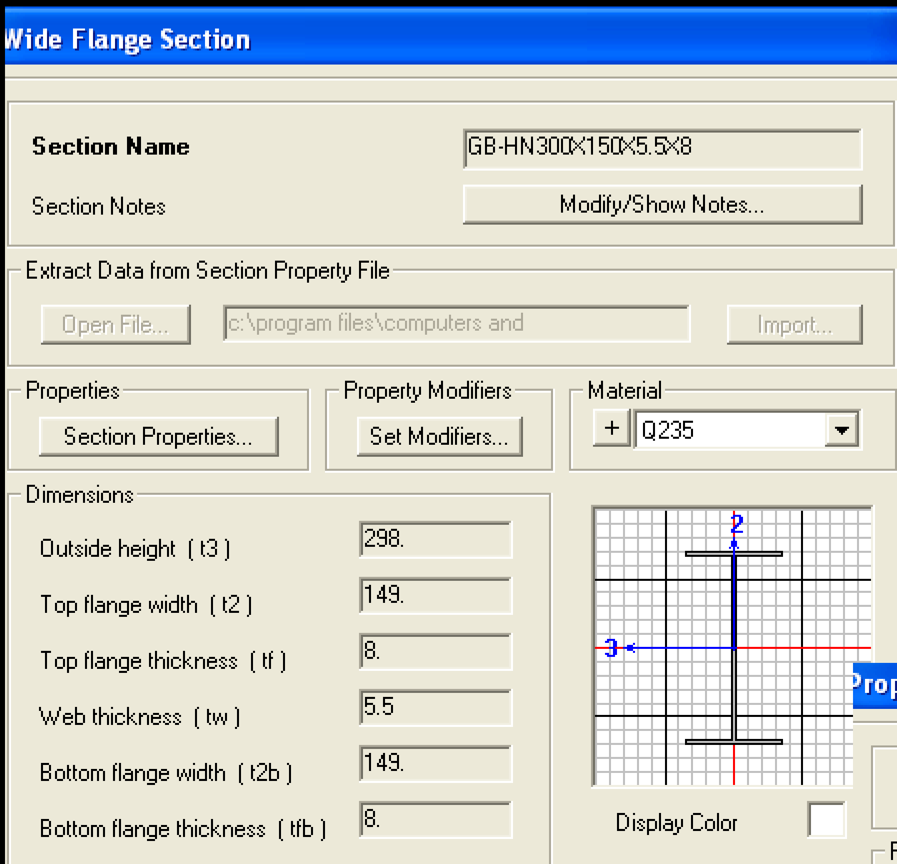

#8

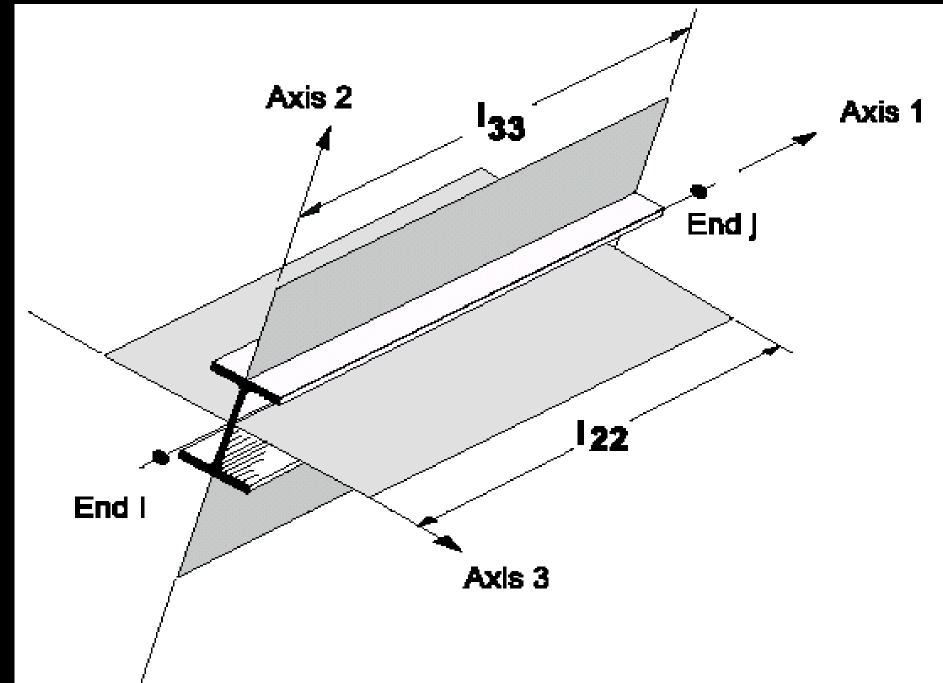

Circular Column
Cross-section Size
Diameter, D
36
in
Rebar Layout
Total Bars
(Ex. 8-#8)
8-#8
Lateral
Ties
Spiral
Use this as pattern for Auto Design
OK
Cancel
Help

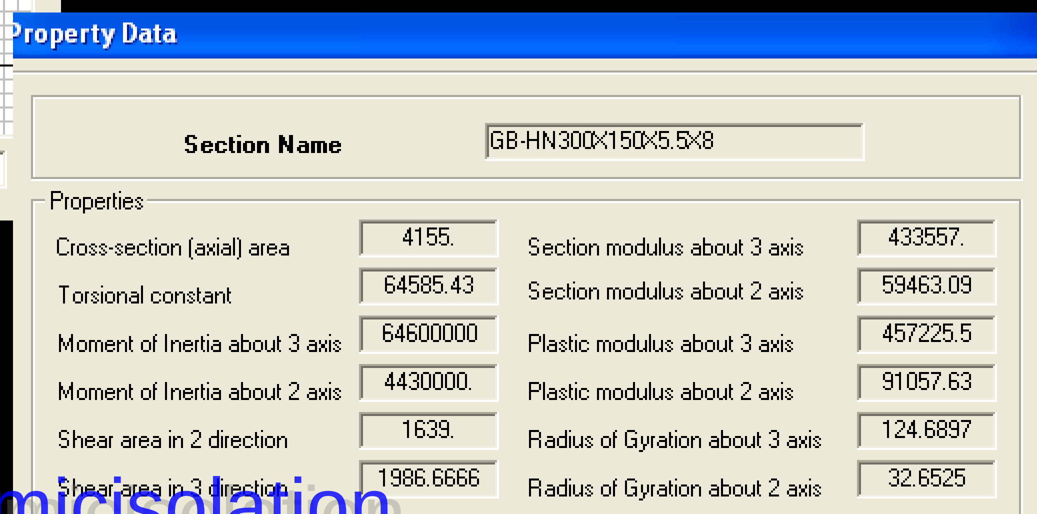




STEEL MEMBER PROPERTIES







CONCRETE MEMBER PROPERTIES

Rectangular Section

Section Name CONBM

Section Notes — Modify/Show Notes...

Properties: Section Properties...

Property Modifiers: Set Modifiers...

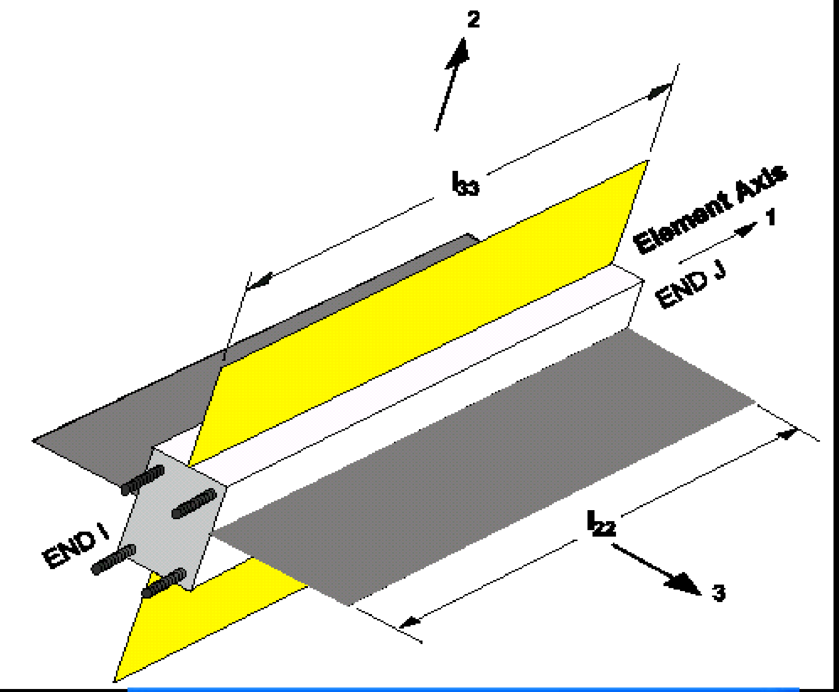
Material: + C30

Dimensions

Depth (t3)	500.
Width (t2)	250.

Display Color

Concrete Reinforcement...



Property Data

Section Name CONBM

Properties

Cross-section (axial) area	125000.	Section modulus about 3 axis	10416667
Torsional constant	1.788E+09	Section modulus about 2 axis	5208333.
Moment of Inertia about 3 axis	2.604E+09	Plastic modulus about 3 axis	15625000
Moment of Inertia about 2 axis	6.510E+08	Plastic modulus about 2 axis	7812500.
Shear area in 2 direction	104166.67	Radius of Gyration about 3 axis	144.3376
Shear area in 3 direction	104166.67	Radius of Gyration about 2 axis	72.1688

Reinforcement Data

Rebar Material

Longitudinal Bars	+ HRB400
Confinement Bars (Ties)	+ HRB400

Design Type

- Column (P-M2-M3 Design)
- Beam (M3 Design Only)

Concrete Cover to Longitudinal Rebar Center

Top	60.
Bottom	60.

Reinforcement Overrides for Ductile Beams

	Left	Right
Top	0.	0.
Bottom	0.	0.

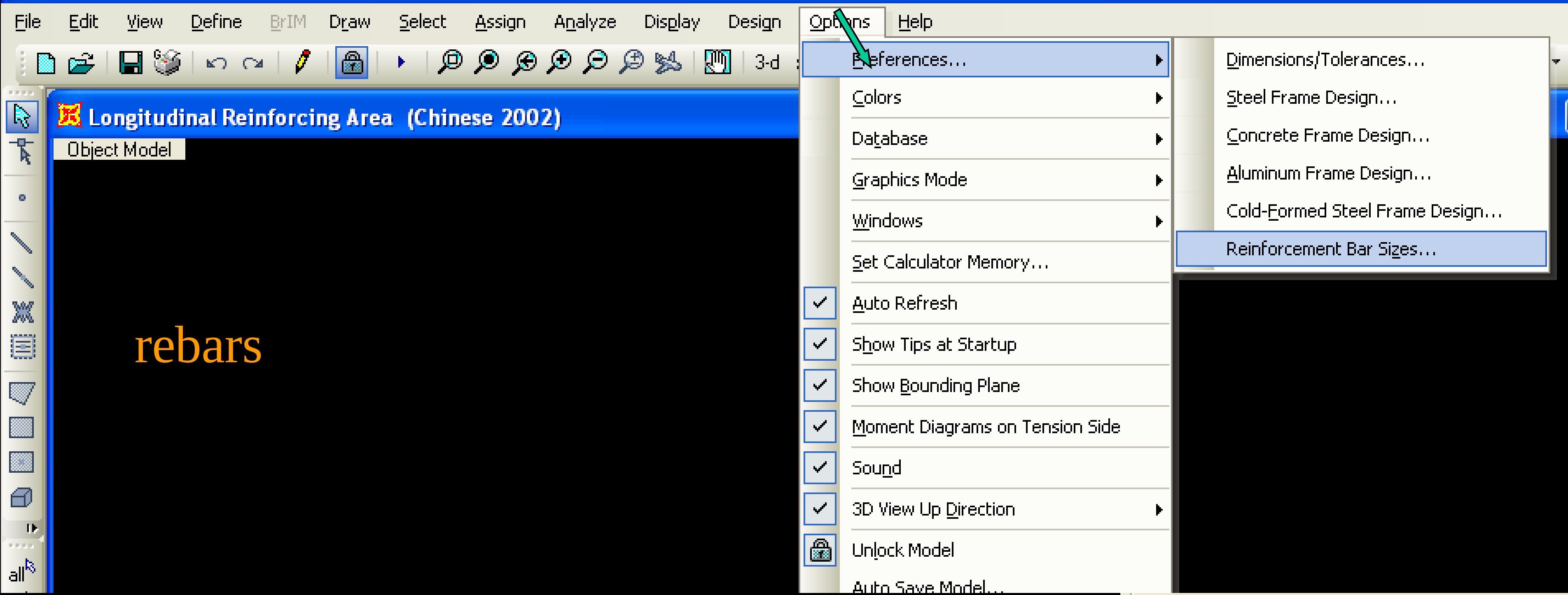
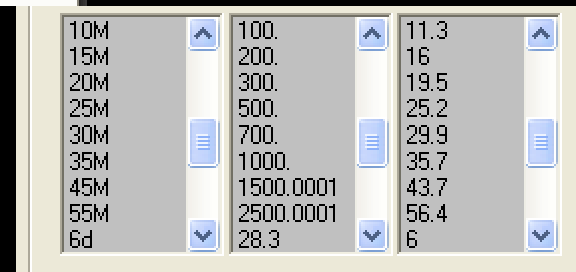
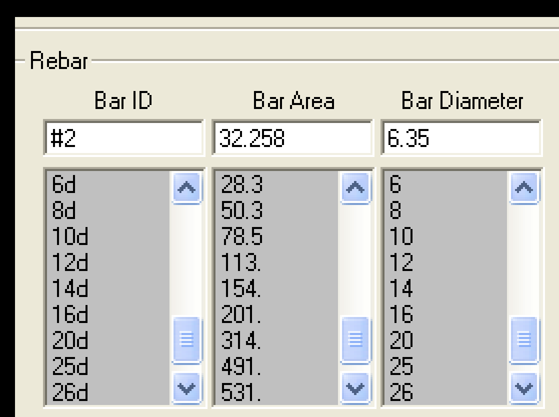


TABLE B.3
ASTM standard reinforcing bars

Nominal Dimensions

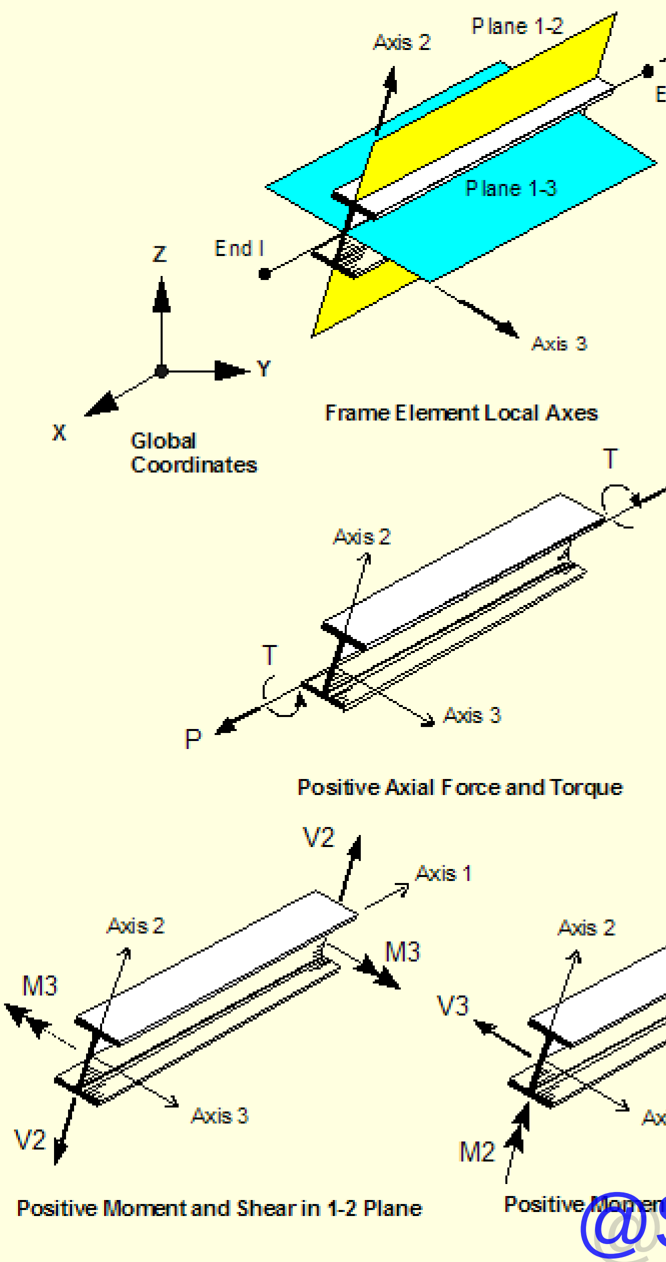
Bar Size[a]	(SI)[b]	Diameter in	mm	Cross-Sect. Area in²	mm²	Weight lbs/ft	Mass kg/m
#3	#10	0.375	9.5	0.11	71	0.376	0.560
#4	#13	0.500	12.7	0.20	129	0.668	0.944
#5	#16	0.625	15.9	0.31	199	1.043	1.552
#6	#19	0.750	19.1	0.44	284	1.502	2.235
#7	#22	0.875	22.2	0.60	387	2.044	3.042
#8	#25	1.000	25.4	0.79	510	2.670	3.973
#9	#29	1.128	28.7	1.00	645	3.400	5.060
#10	#32	1.270	32.3	1.27	819	4.303	6.404
#11	#36	1.410	35.8	1.56	1006	5.313	7.907





Frame and Nllink Element

FRAME element follows the designer's convention.

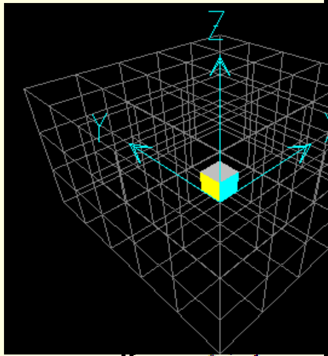


Frame Element Internal Forces

MEMBER ORIENTATION

Is defined by local coordinate system

Typical: **Moment 3-3**, **Shear 2-2**



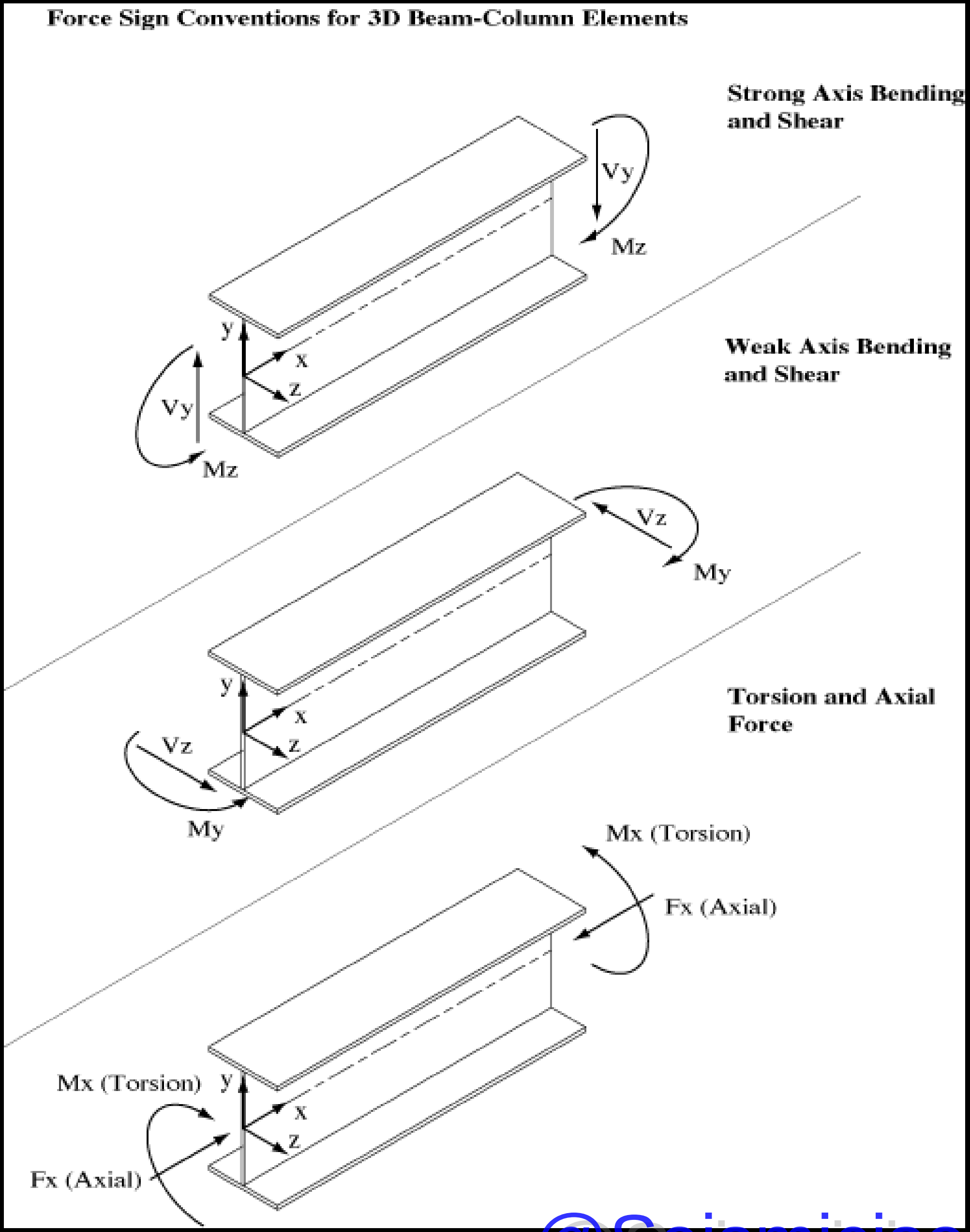
Each part of the structure (e.g. joint, element) has its own **LOCAL** coordinate system **1-2-3**.

The joint local coordinate system is normally the same as the global X-Y-Z coordinate system.

For the elements, one of the element local axes is determined by the geometry of the individual element; the orientation of the remaining two axes is defined by specifying a single angle of rotation.

For frame elements, for example, **the local axis 1 is always the longitudinal axis of the element with the positive direction from I to J.** The default orientation of the local **1-2** plane in SAP is taken to be vertical (i.e. parallel to the Z-axis). The local **3**-axis is always horizontal (i.e. lies in the X-Y plane).

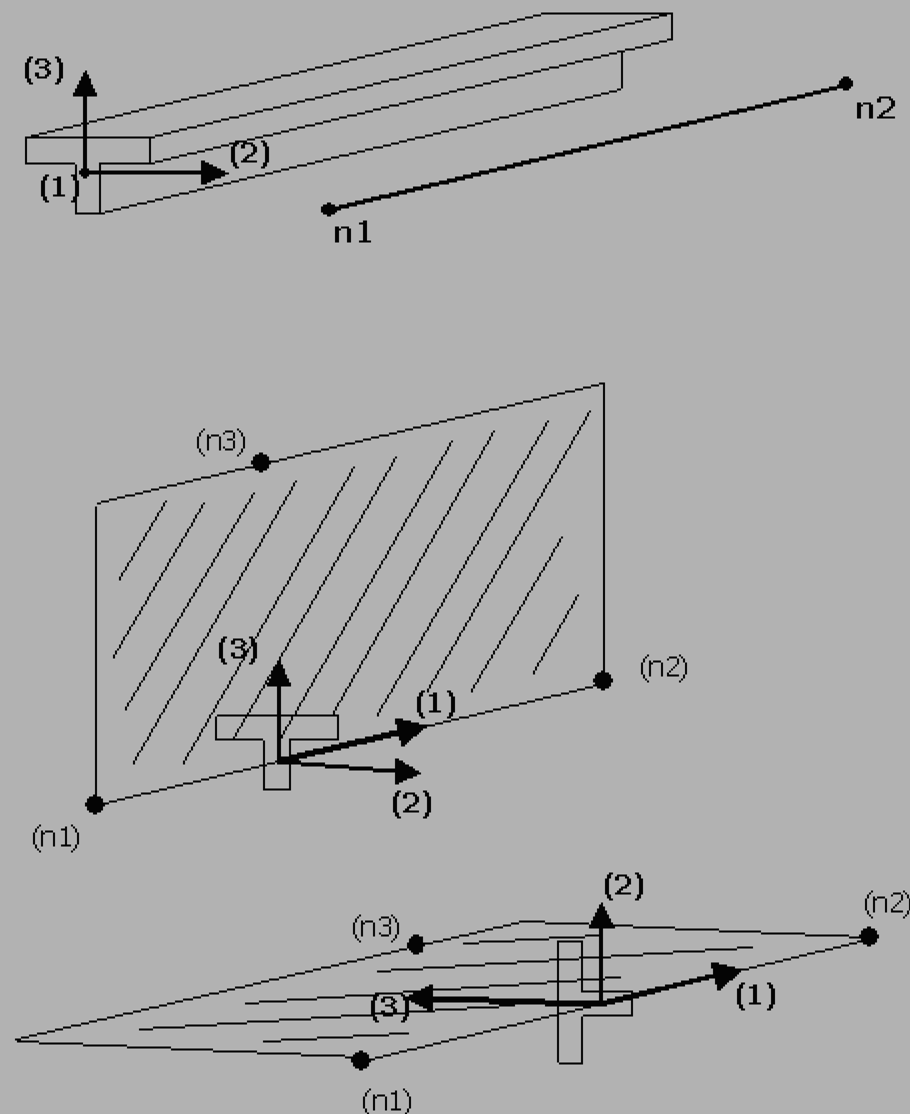
INTERNAL MEMBER FORCES

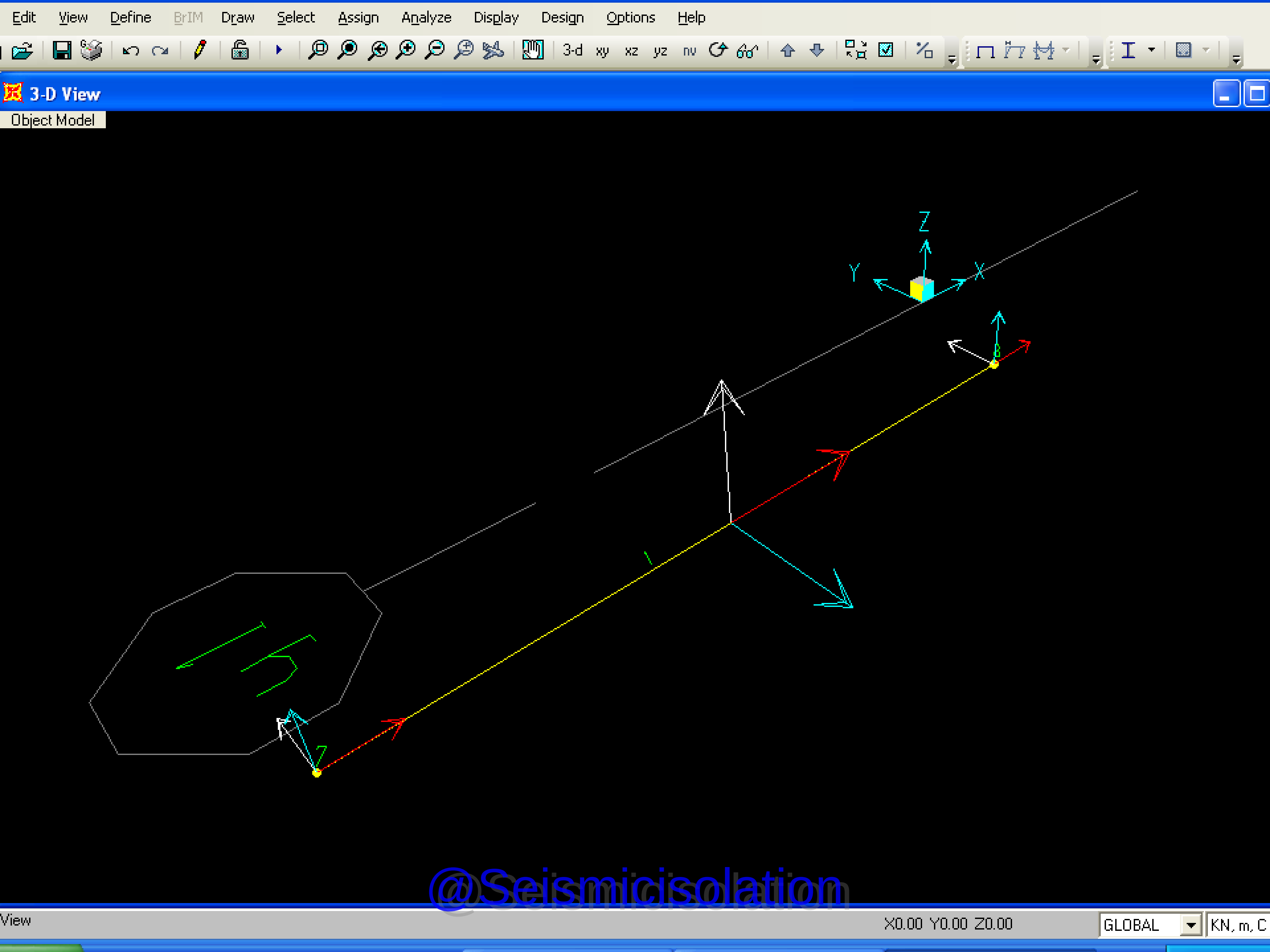


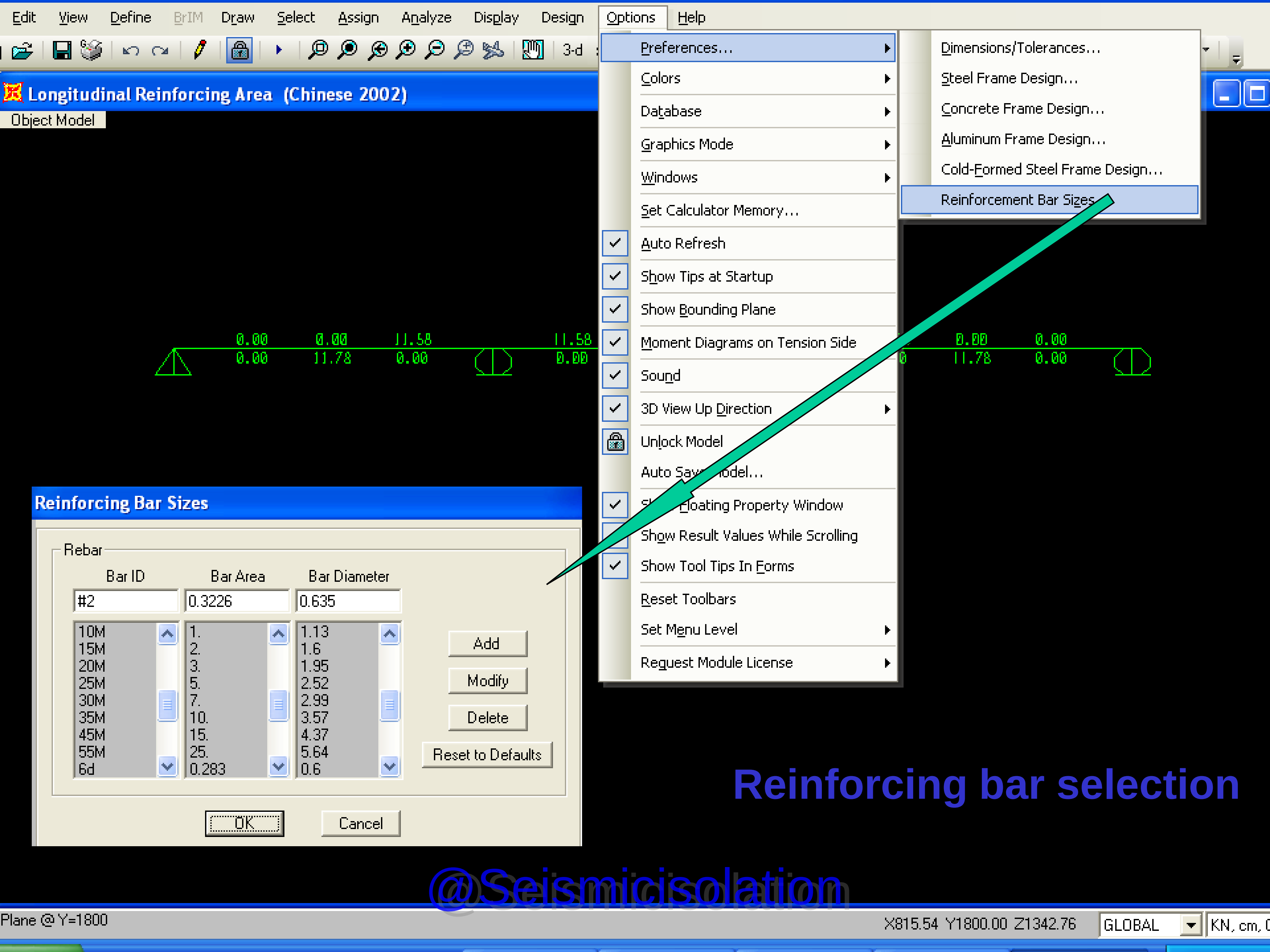
A member has six potential internal force quantities in three dimensional space. Internal moments are defined as positive according to the **right-hand rule** acting at the i-node end.

In two dimensional or planar space, there are only three force quantities:

- **Bending moment**
- **Shear**
- **Axial force.**


(3)
(2)
(1)
n2
n1
(n3)
(3)
(n2)
(1)
(2)
(n1)
(2)
(n2)
(n3)
(3)
(1)
(n1)


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
3-D View
Object Model
Z
Y
X
View
X0.00 Y0.00 Z0.00
GLOBAL
KN, m, C



Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
Longitudinal Reinforcing Area (Chinese 2002)
Object Model
Preferences...
Colors
Database
Graphics Mode
Windows
Set Calculator Memory...
Auto Refresh
Show Tips at Startup
Show Bounding Plane
Moment Diagrams on Tension Side
Sound
3D View Up Direction
Unlock Model
Auto Save Model...
Show Floating Property Window
Show Result Values While Scrolling
Show Tool Tips In Forms
Reset Toolbars
Set Menu Level
Request Module License
Dimensions/Tolerances...
Steel Frame Design...
Concrete Frame Design...
Aluminum Frame Design...
Cold-Formed Steel Frame Design...
Reinforcement Bar Sizes...
Reinforcing Bar Sizes
Rebar
Bar ID
Bar Area
Bar Diameter
#2
0.3226
0.635
10M 15M 20M 25M 30M 35M 45M 55M 6d
1. 2. 3. 5. 7. 10. 15. 25. 0.283
1.13 1.6 1.95 2.52 2.99 3.57 4.37 5.64 0.6
Add
Modify
Delete
Reset to Defaults
OK
Cancel
Plane @ Y=1800
X815.54 Y1800.00 Z1342.76
GLOBAL
KN, cm, C
Reinforcing bar selection

LOADS

Joint (**nodal**) **loads**, **element loads** (concentrated forces and moments, and distributed loads on frames and surfaces)**:** ***static loads*** (e.g. gravity, pressure, prestress, torsional, thermal) and **displacements** applied to nodes or in elements, ***thermal loads*** and other *indirect loads* due to *creep, shrinkage* or *settlement,* ***moving loads***, ***vibrational loads***, ***seismic loads***; sloped loads vs. projected loads, **local loads** versus **global loads**.
Self-weight is included automatically in structural software programs

The common loads used in structural engineering software correspond to the typical loading conditions.

Dead load and self-weight
Live load
Snow load, roof live load
Wind and earthquake loads
Water and earth pressure loads
Loads due to restrained volume change:
  thermal loads, joint and support displacement loads, prestress loads
Dynamic loads including time-history loads
Abnormal loads including accidental loads
Moving vehicle and lane loading

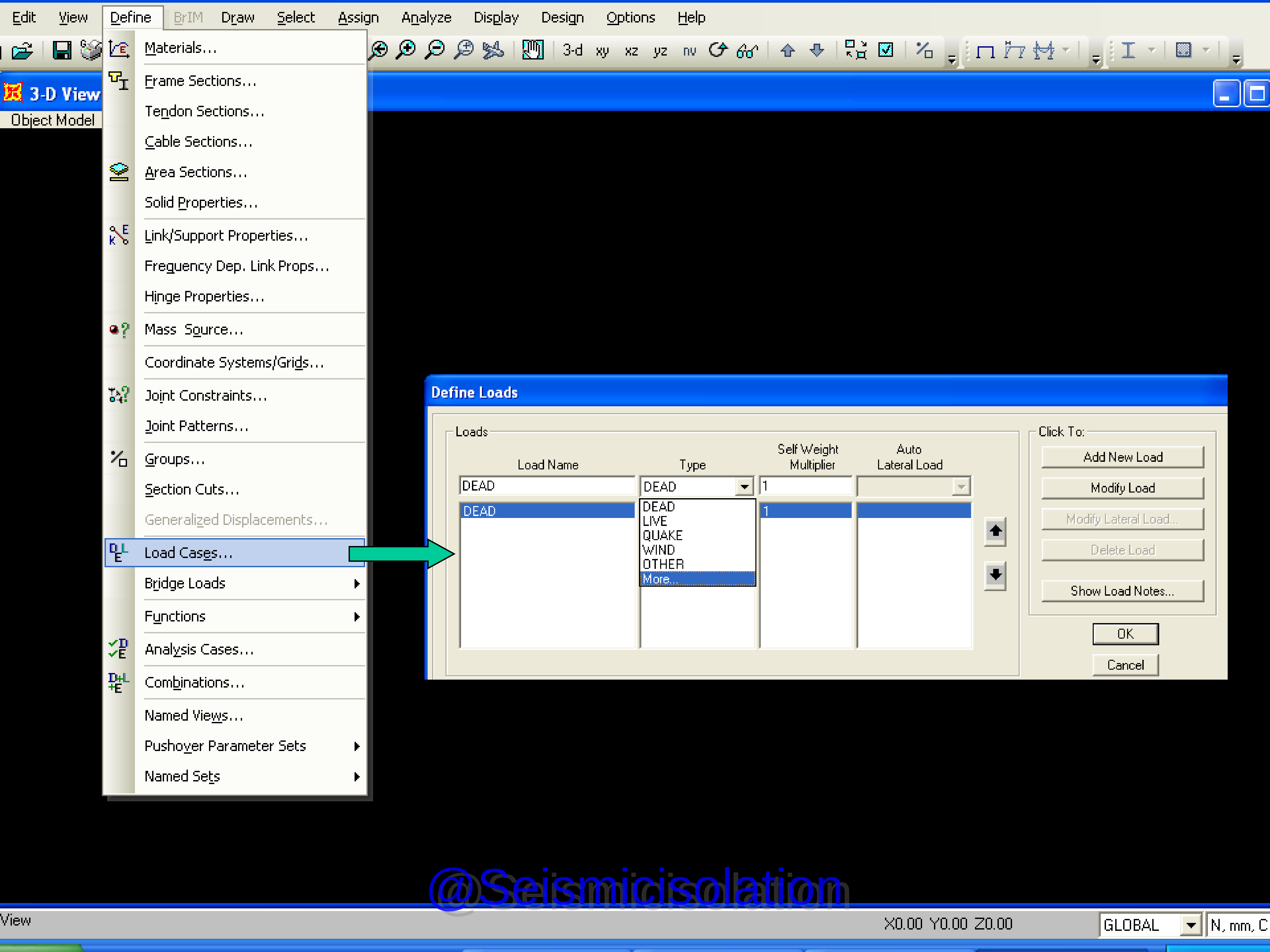
Some of the structural engineering software provides automatic wind load generation from user-specified wind intensity as well as UBC Seismic load generation for calculation and automatic distribution of base shear according to code stipulation

Introduction to:

(i) Self-weight of the structure
(ii) Vertical loads (dead loads and live loads)
(iii) Horizontal loads (wind loads)
(iv) Total loads

Typical loads

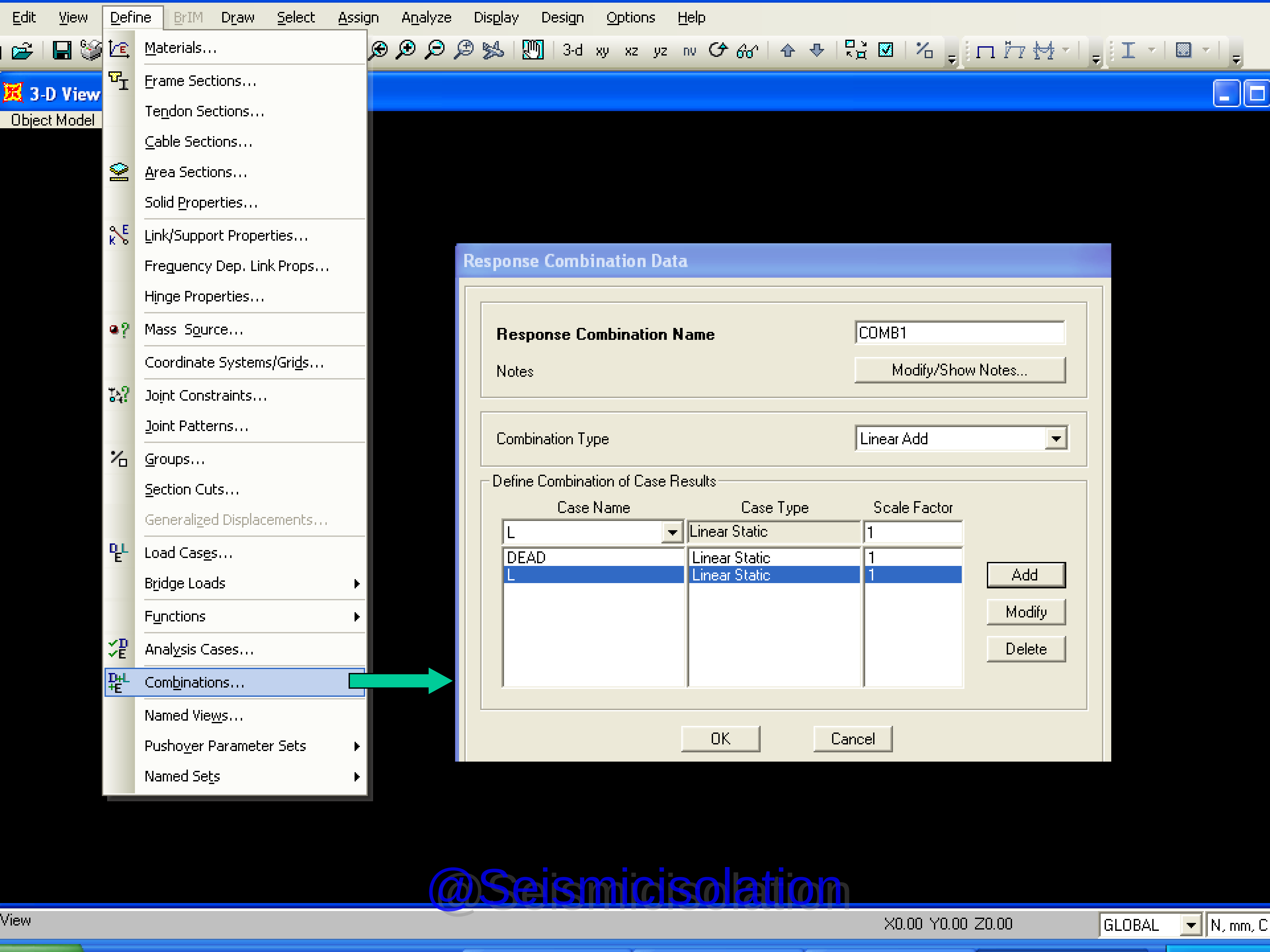
	Dead loads	Live loads	Snow loads	Wind loads
	kN/m^2 (kPa) psf	kN/m^2 psf	kN/m^2 psf	kN/m^2 psf
Floors	3.0 - 4.0 60 - 80	2.0 - 4.0 40-80	#	#
Roofs	2.00 - 2.50 40 - 50	1.0 – 2.0 20 - 40	1.00 20	#
Walls	#	#	#	1.0 - 1.5 20 - 30

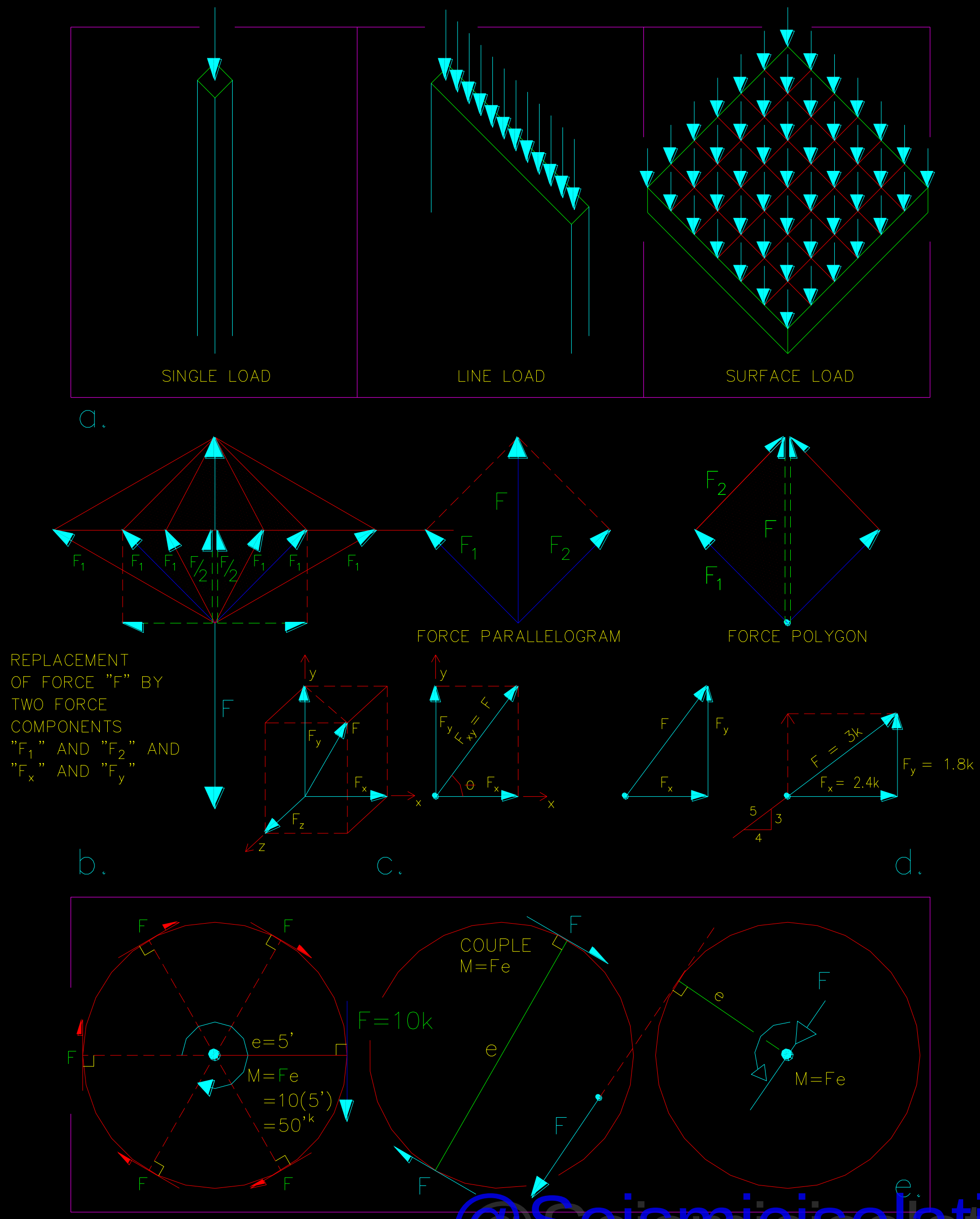

Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-D View
Object Model
Materials...
Frame Sections...
Tendon Sections...
Cable Sections...
Area Sections...
Solid Properties...
Link/Support Properties...
Frequency Dep. Link Props...
Hinge Properties...
Mass Source...
Coordinate Systems/Grids...
Joint Constraints...
Joint Patterns...
Groups...
Section Cuts...
Generalized Displacements...
Load Cases...
Bridge Loads
Functions
Analysis Cases...
Combinations...
Named Views...
Pushover Parameter Sets
Named Sets
Define Loads
Loads
Load Name
Type
Self Weight Multiplier
Auto Lateral Load
DEAD
DEAD
1
DEAD
LIVE
QUAKE
WIND
OTHER
More...
Click To:
Add New Load
Modify Load
Modify Lateral Load...
Delete Load
Show Load Notes...
OK
Cancel
@Seismicisolation
X0.00 Y0.00 Z0.00
GLOBAL
N, mm, C

LOAD COMBINATION

In structural engineering software programs the user defines first the various load cases and then may set up the load combinations depending on the type of program. It may be the responsibility of the user to determine which combination will control the design.

Some programs assemble automatically basic load cases into load combinations; they define also the load factors for each load according to codes and type of analysis. In other words, they provide already sets of default design load combinations so that the user only defines the required combinations according to the selected code (e.g. model codes, *ASCE 7-98, AISC-ASD 89, AISC-LRFD 99, ACI 318-99, NDS 1997*). Then the program may select automatically the critical load combination for design.


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-D View
Object Model
Materials...
Frame Sections...
Tendon Sections...
Cable Sections...
Area Sections...
Solid Properties...
Link/Support Properties...
Frequency Dep. Link Props...
Hinge Properties...
Mass Source...
Coordinate Systems/Grids...
Joint Constraints...
Joint Patterns...
Groups...
Section Cuts...
Generalized Displacements...
Load Cases...
Bridge Loads
Functions
Analysis Cases...
Combinations...
Named Views...
Pushover Parameter Sets
Named Sets
Response Combination Data
Response Combination Name
COMB1
Notes
Modify/Show Notes...
Combination Type
Linear Add
Define Combination of Case Results
Case Name
Case Type
Scale Factor
L
Linear Static
1
DEAD
Linear Static
1
L
Linear Static
1
Add
Modify
Delete
OK
Cancel
X0.00 Y0.00 Z0.00
GLOBAL
N, mm, C



Properties of Forces

- **magnitude**
- **direction**
- **location**

BOUNDARY CONDITIONS: **nodal restraints** (frame end releases), **external supports**

The deflection of the structural model is governed by the displacements of the nodes where **every node may have six displacement components**. The node may **translate** along its three local axes as denoted by **U1**, **U2**, **U3**, and it may **rotate** about its three local axes as denoted by **R1**, **R2**, **R3**. The **six** displacement components are known as the **degrees of freedom** of the node. For **planar structures** you may want to restrict the available degrees of freedom to **three** such as for the **vertical X-Z** or **horizontal X-Y planes**.

Each degree of freedom must be one of the following types:

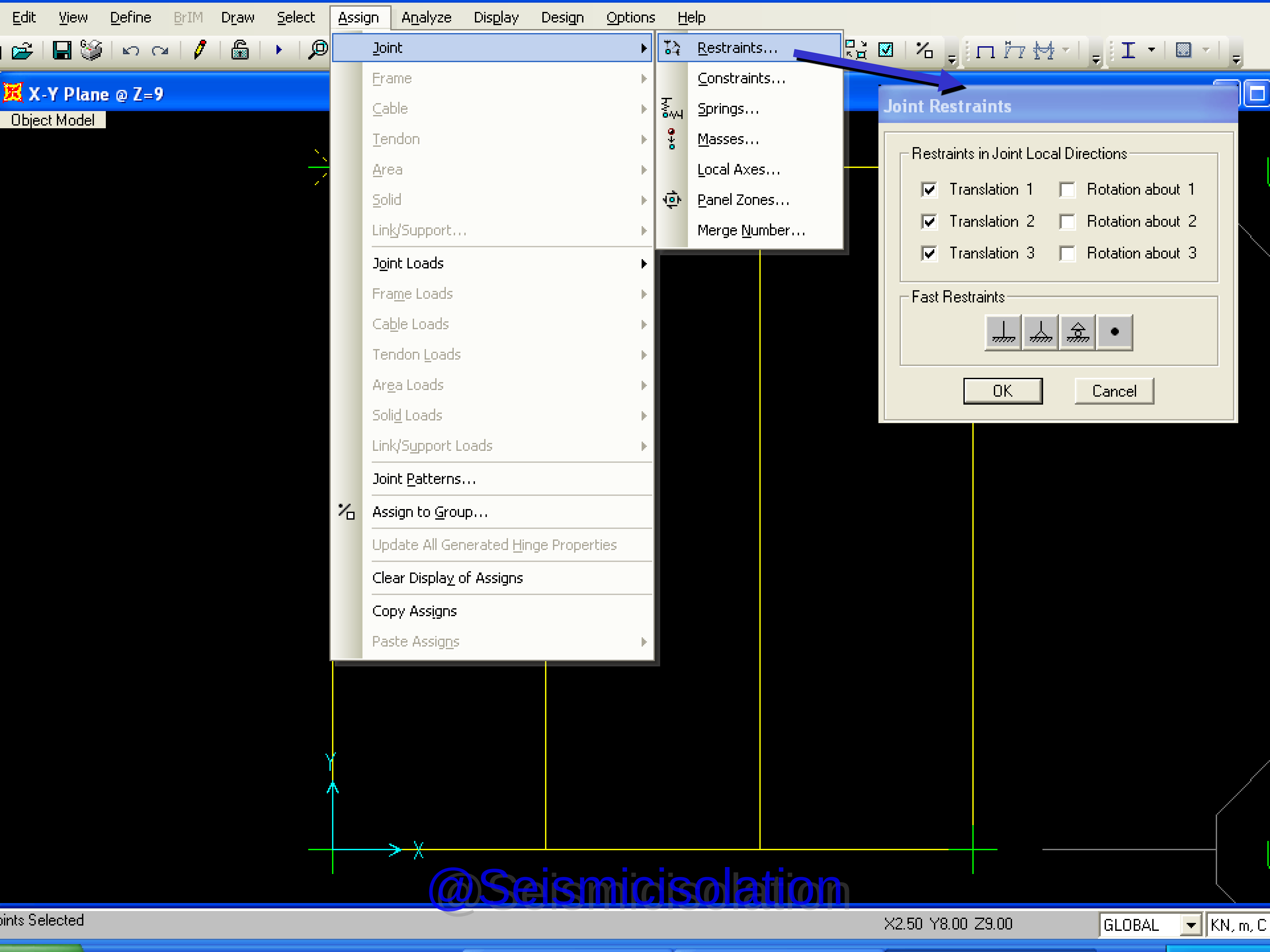
Active, where the displacement is computed during the analysis

Restrained, where the displacement is specified, and the corresponding ***reaction*** is computed during the analysis (e.g. external supports)

Constrained, where the displacement is determined from the displacements at other degrees of freedom, as applied to rigid body behavior (e.g. rigid diaphragms) to connect together different parts of the model, and to impose certain types of symmetry conditions. Using constraints reduces the number of degrees of freedom in the model, which can make the analysis run faster.

Null, where the displacement does not affect the structure and is ignored by the analysis

Unavailable, where the displacement has been explicitly excluded from the analysis


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
X-Y Plane @ Z=9
Object Model
Joint
Frame
Cable
Tendon
Area
Solid
Link/Support...
Joint Loads
Frame Loads
Cable Loads
Tendon Loads
Area Loads
Solid Loads
Link/Support Loads
Joint Patterns...
Assign to Group...
Update All Generated Hinge Properties
Clear Display of Assigns
Copy Assigns
Paste Assigns
Restraints...
Constraints...
Springs...
Masses...
Local Axes...
Panel Zones...
Merge Number...
Joint Restraints
Restraints in Joint Local Directions
Translation 1
Rotation about 1
Translation 2
Rotation about 2
Translation 3
Rotation about 3
Fast Restraints
OK
Cancel
Y
X
@Seismicisolation
X2.50 Y8.00 Z9.00
GLOBAL
KN, m, C

B. CHECK INPUT DATA

C. ANALYSIS

The **ANALYSIS** determines the **reactions**, **internal forces**, and **displacements** of the model in response to the **loads**.

Three key concepts are underlying the theory of structural analysis:

- **equilibrium of forces**
- **relationship between forces and displacements**
- **compatibility of displacements**

Depending on the loading conditions and type of structure either a **static** or **dynamic analysis** must be performed. Some structural engineering software also offer in addition other types of analyses for special situations such as: **moving load analysis**, **buckling analysis**, and **fatigue analysis**.

TYPE OF ANALYSIS: many different types of analysis are available such as the common given below

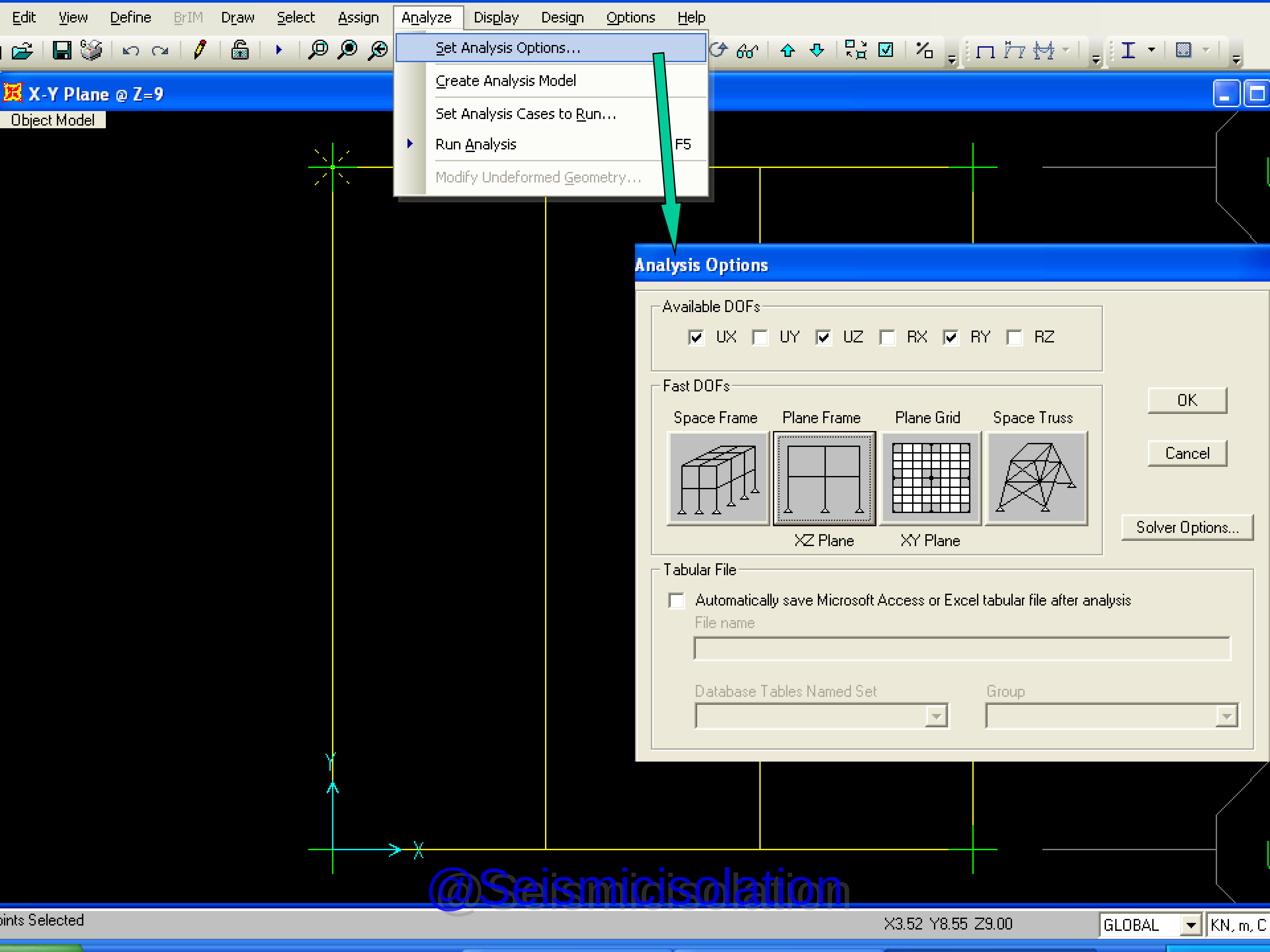
STATIC ANALYSIS:

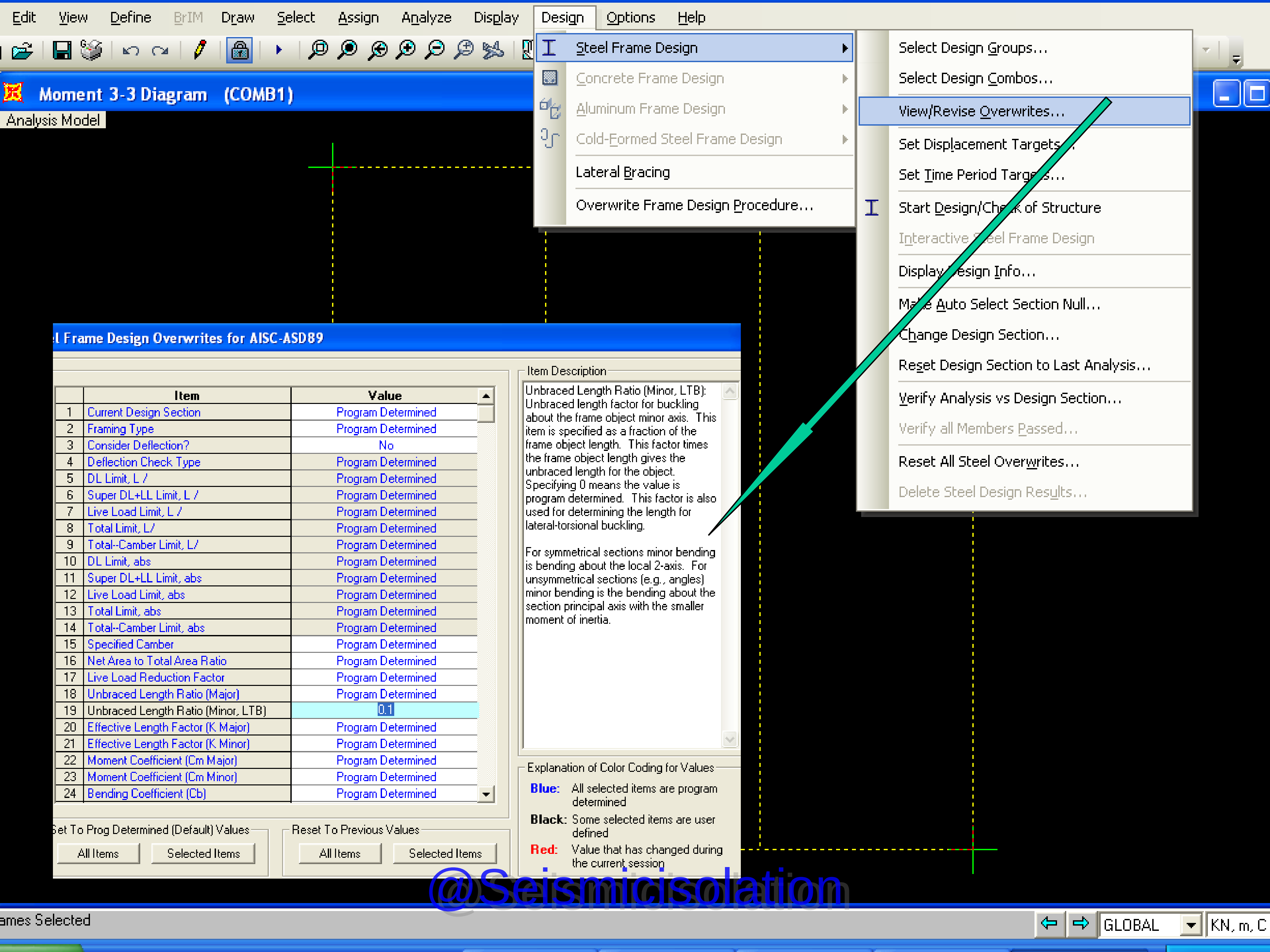
- **Linear Static Analysis**
- **Nonlinear Static Analysis:**
  - **P-Delta Analysis**
  - **Pushover Analysis**
- **Seismic Analysis: Equivalent Lateral Force Analysis**

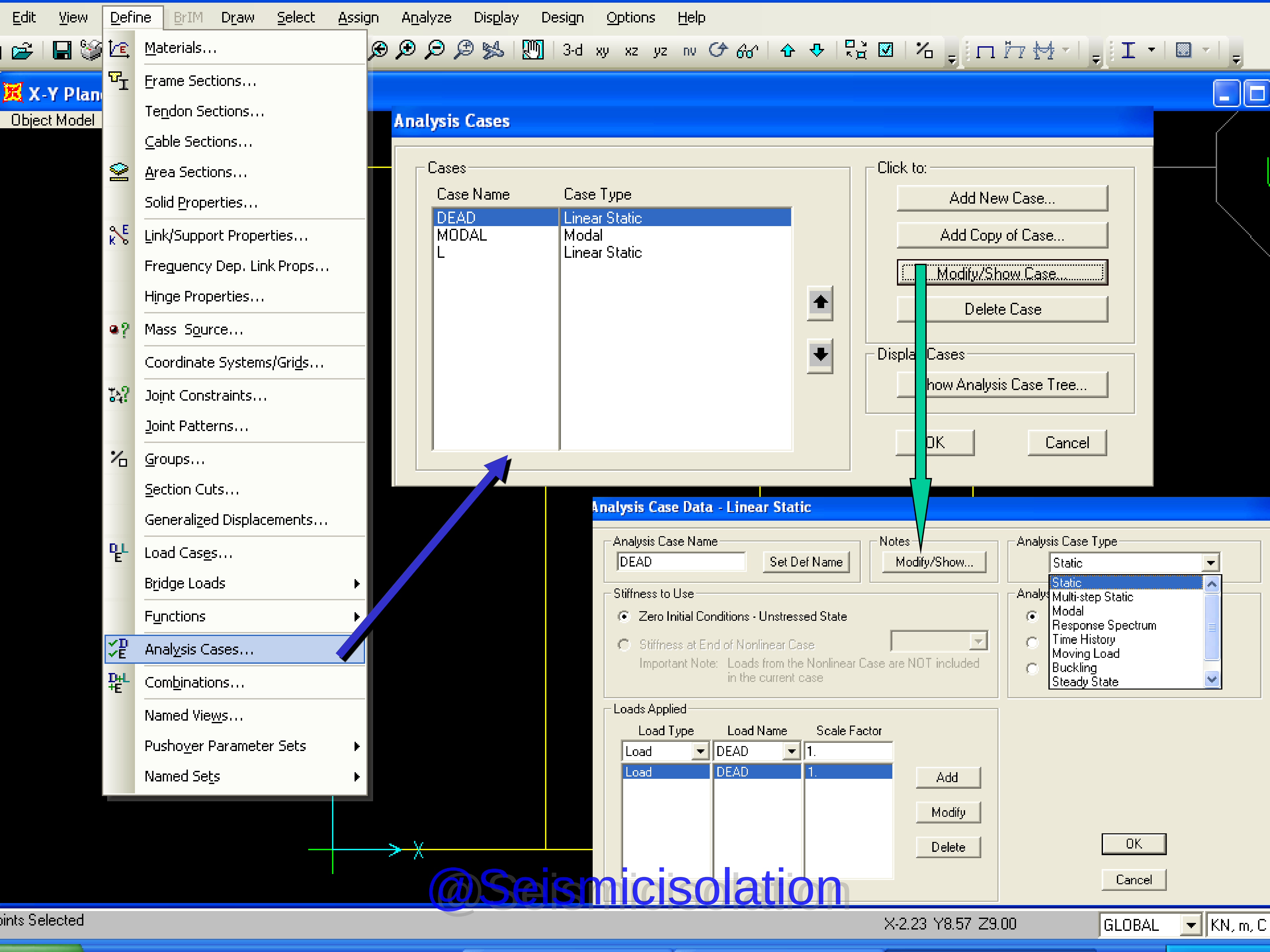
DYNAMIC ANALYSIS:

- **Modal Analysis for vibration modes:**
  - **Eigenvalue Analysis**
- **Harmonic Steady-State Analysis**
- **Seismic Analysis:**
  - **Modal Response-Spectrum Analysis**
  - **Linear Time-history Analysis**
  - **Nonlinear Time-history Analysis**

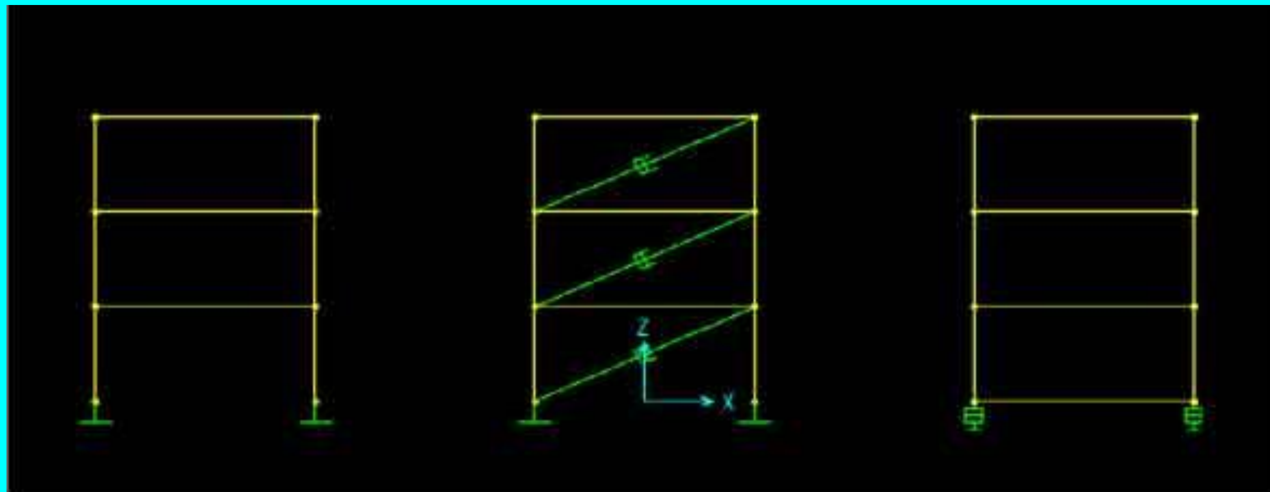
Other load analyses include: MOVING LOAD ANALYSIS, LINEAR BUCKLING-MODE ANALYSIS


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
Set Analysis Options...
Create Analysis Model
Set Analysis Cases to Run...
Run Analysis F5
Modify Undeformed Geometry...
X-Y Plane @ Z=9
Object Model
Analysis Options
Available DOFs
UX UY UZ RX RY RZ
Fast DOFs
Space Frame Plane Frame Plane Grid Space Truss
XZ Plane XY Plane
OK
Cancel
Solver Options...
Tabular File
Automatically save Microsoft Access or Excel tabular file after analysis
File name
Database Tables Named Set
Group
@Seismicisolation
X3.52 Y8.55 Z9.00
GLOBAL
KN, m, C

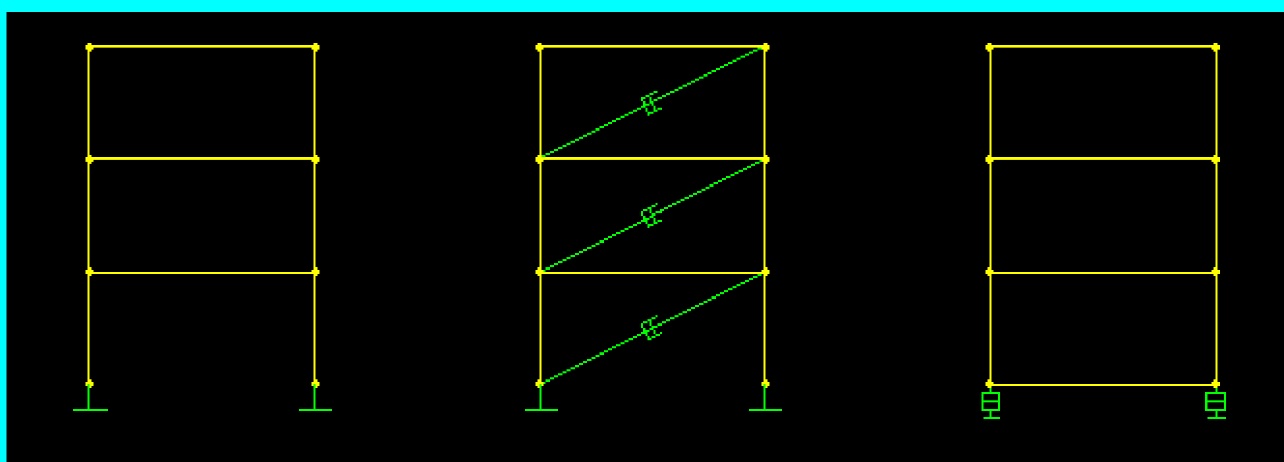

Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
Moment 3-3 Diagram (COMB1)
Analysis Model
Steel Frame Design
Concrete Frame Design
Aluminum Frame Design
Cold-Formed Steel Frame Design
Lateral Bracing
Overwrite Frame Design Procedure...
Select Design Groups...
Select Design Combos...
View/Revise Overwrites...
Set Displacement Targets...
Set Time Period Targets...
Start Design/Check of Structure
Interactive Steel Frame Design
Display Design Info...
Make Auto Select Section Null...
Change Design Section...
Reset Design Section to Last Analysis...
Verify Analysis vs Design Section...
Verify all Members Passed...
Reset All Steel Overwrites...
Delete Steel Design Results...
l Frame Design Overwrites for AISC-ASD89
Item
Value
1 Current Design Section Program Determined
2 Framing Type Program Determined
3 Consider Deflection? No
4 Deflection Check Type Program Determined
5 DL Limit, L / Program Determined
6 Super DL+LL Limit, L / Program Determined
7 Live Load Limit, L / Program Determined
8 Total Limit, L/ Program Determined
9 Total--Camber Limit, L/ Program Determined
10 DL Limit, abs Program Determined
11 Super DL+LL Limit, abs Program Determined
12 Live Load Limit, abs Program Determined
13 Total Limit, abs Program Determined
14 Total--Camber Limit, abs Program Determined
15 Specified Camber Program Determined
16 Net Area to Total Area Ratio Program Determined
17 Live Load Reduction Factor Program Determined
18 Unbraced Length Ratio (Major) Program Determined
19 Unbraced Length Ratio (Minor, LTB) 0.1
20 Effective Length Factor (K Major) Program Determined
21 Effective Length Factor (K Minor) Program Determined
22 Moment Coefficient (Cm Major) Program Determined
23 Moment Coefficient (Cm Minor) Program Determined
24 Bending Coefficient (Cb) Program Determined
Item Description
Unbraced Length Ratio (Minor, LTB): Unbraced length factor for buckling about the frame object minor axis. This item is specified as a fraction of the frame object length. This factor times the frame object length gives the unbraced length for the object. Specifying 0 means the value is program determined. This factor is also used for determining the length for lateral-torsional buckling.
For symmetrical sections minor bending is bending about the local 2-axis. For unsymmetrical sections (e.g., angles) minor bending is the bending about the section principal axis with the smaller moment of inertia.
Explanation of Color Coding for Values
Blue: All selected items are program determined
Black: Some selected items are user defined
Red: Value that has changed during the current session
Set To Prog Determined (Default) Values
All Items
Selected Items
Reset To Previous Values
All Items
Selected Items
ames Selected
GLOBAL
KN, m, C
@Seismicisolation


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
X-Y Plane
Object Model
Materials...
Frame Sections...
Tendon Sections...
Cable Sections...
Area Sections...
Solid Properties...
Link/Support Properties...
Frequency Dep. Link Props...
Hinge Properties...
Mass Source...
Coordinate Systems/Grids...
Joint Constraints...
Joint Patterns...
Groups...
Section Cuts...
Generalized Displacements...
Load Cases...
Bridge Loads
Functions
Analysis Cases...
Combinations...
Named Views...
Pushover Parameter Sets
Named Sets
Analysis Cases
Cases
Case Name
Case Type
DEAD Linear Static
MODAL Modal
L Linear Static
Click to:
Add New Case...
Add Copy of Case...
Modify/Show Case...
Delete Case
Display Cases
Show Analysis Case Tree...
OK
Cancel
Analysis Case Data - Linear Static
Analysis Case Name
DEAD
Set Def Name
Notes
Modify/Show...
Analysis Case Type
Static
Multi-step Static
Modal
Response Spectrum
Time History
Moving Load
Buckling
Steady State
Stiffness to Use
Zero Initial Conditions - Unstressed State
Stiffness at End of Nonlinear Case
Important Note: Loads from the Nonlinear Case are NOT included in the current case
Loads Applied
Load Type
Load Name
Scale Factor
Load DEAD 1.
Add
Modify
Delete
OK
Cancel
X-2.23 Y8.57 Z9.00
GLOBAL
KN, m, C

Static linear & nonlinear analysis

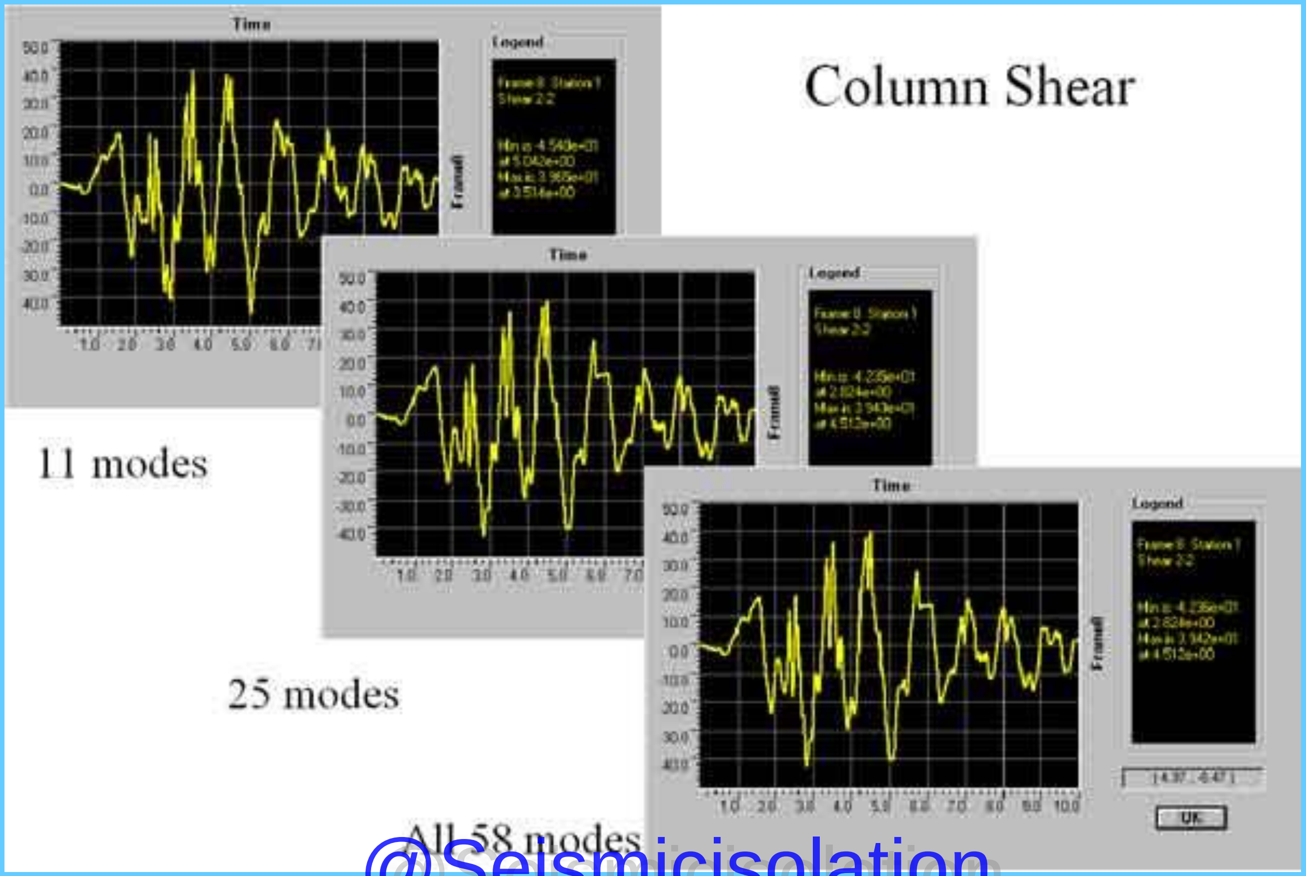


Linear Structure | Structure with Dampers | Base-isolated Structure

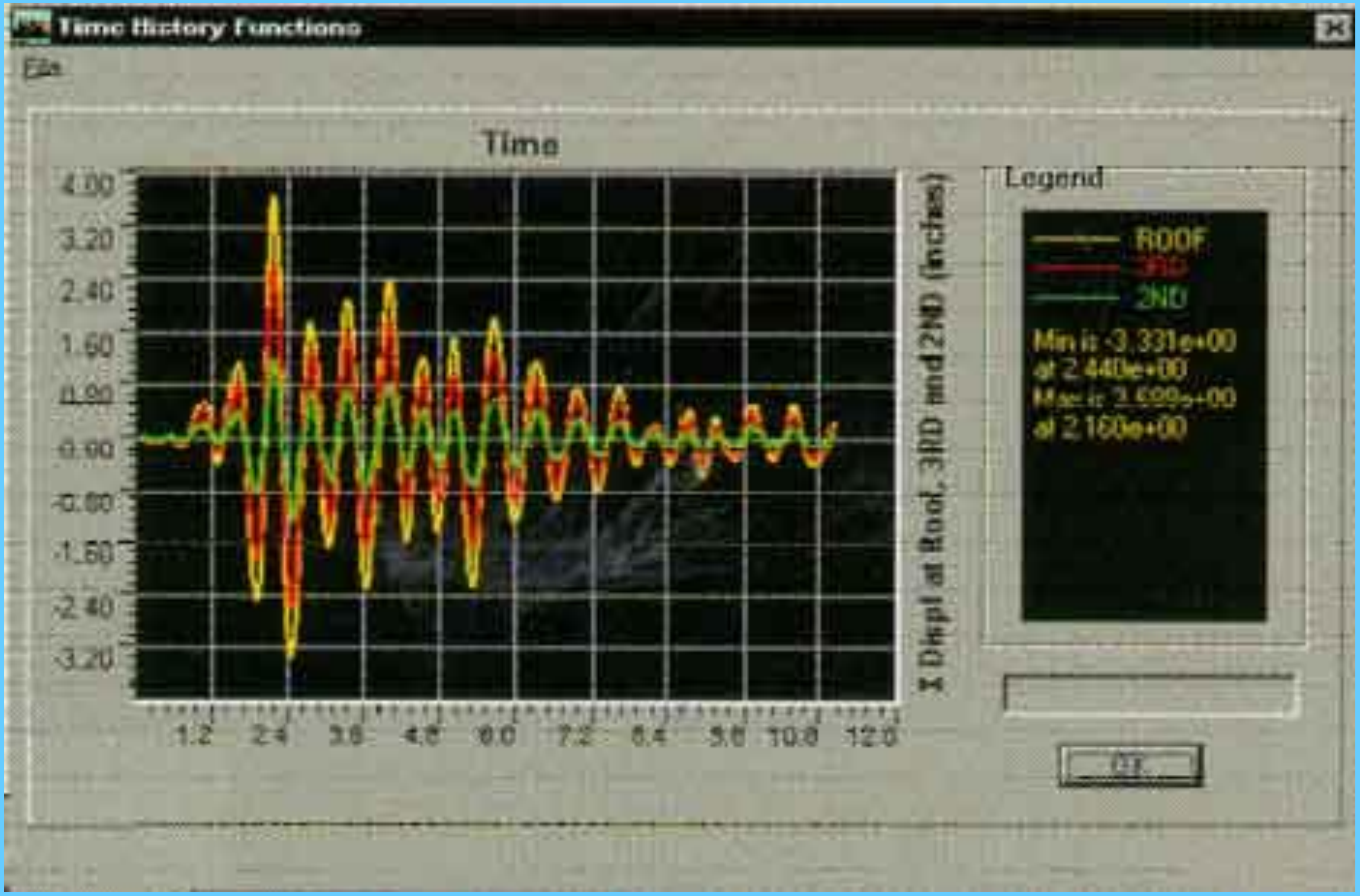


Modal analysis

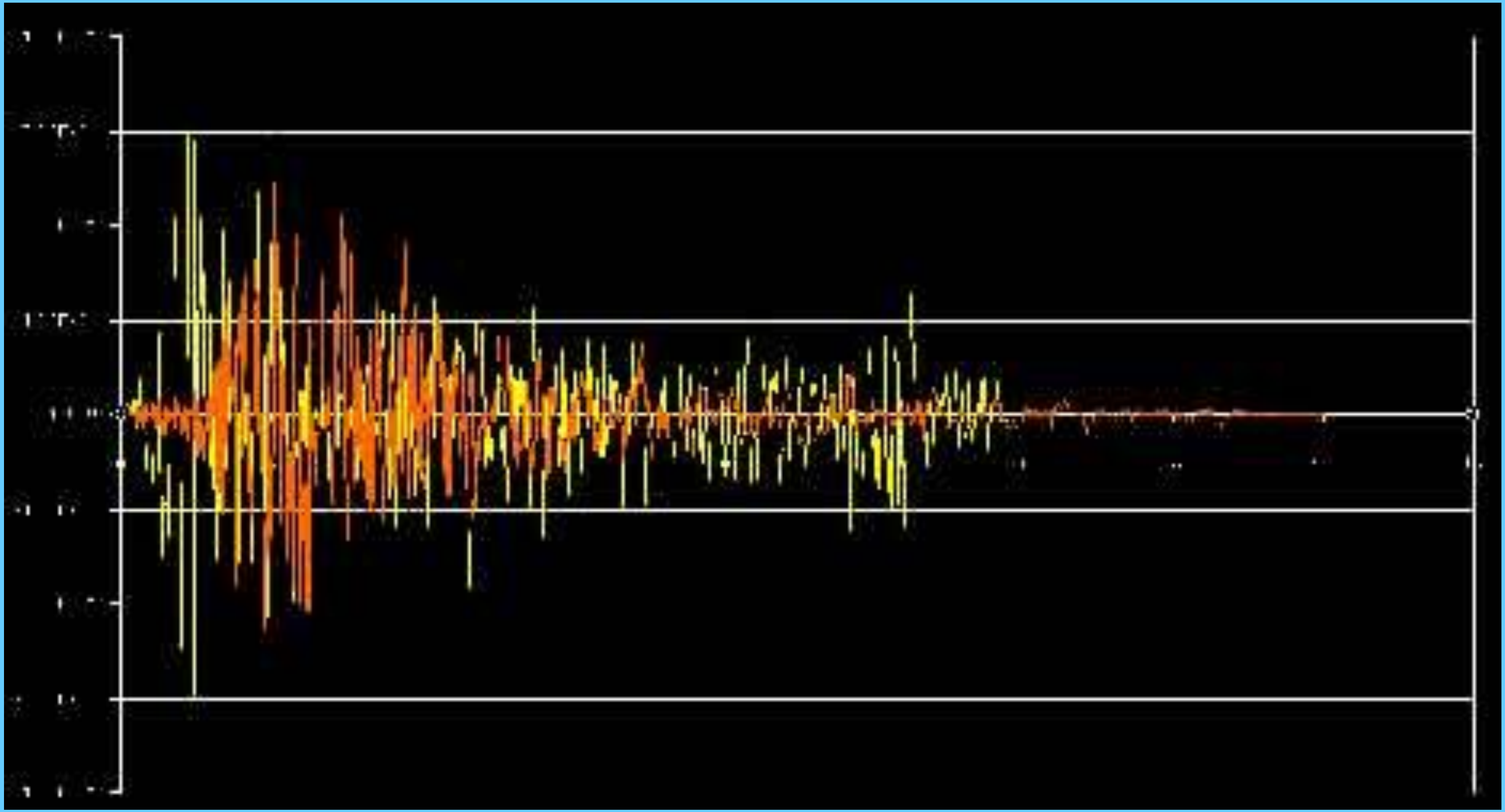
- Eigenvalue analysis with an accelerated subspace iteration algorithm
- Ritz analysis for optimal mode superposition basis
- Modes can include P-delta, large displacement and construction effects



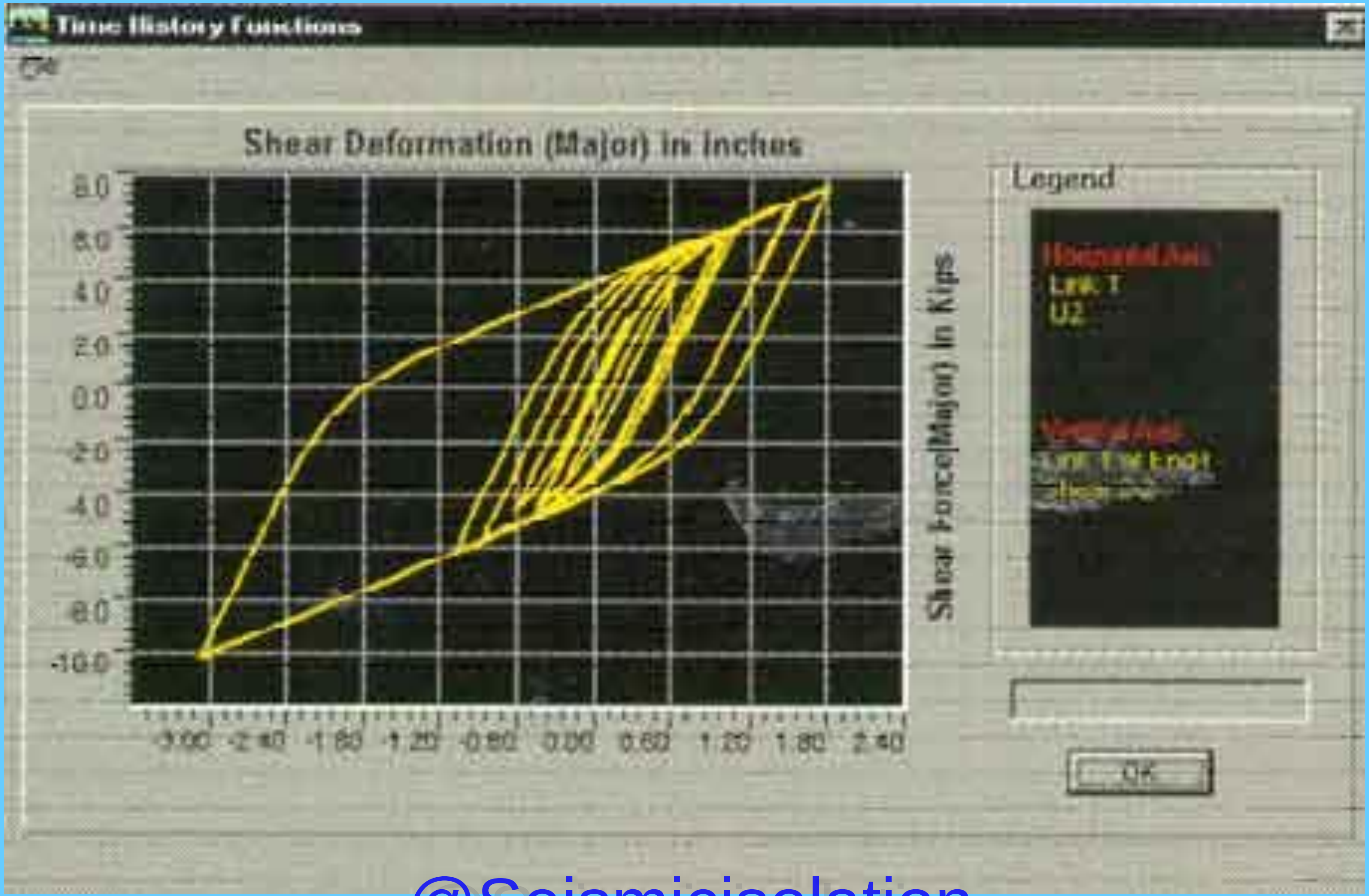
Time history analysis



Dynamic response spectrum analysis



Nonlinear element analysis



D. EVALUATE and CHECK RESULTS

The primary difficulty in using structural computer programs is to not truly understand the physical action and boundary conditions as well as the behavior of the structure elements of the real structure, which is especially true for the behavior of the joints. The true constraints (i.e. restraint to movement) that each member exerts on the adjacent ones, can only be approximated. It is apparent, that the mathematical model may not at all represent the reality and true character of the structure. Therefore it is necessary to evaluate whether the results are correct and check for errors and possibly develop an improved model. It is also necessary to verify computer answers with hand calculations.

Finite element computer programs report the results of *nodal displacements*, *support reactions* and *member forces or stresses* in graphical and numerical (tabular) form immediately. It is apparent that the meaning of the results must be checked as discussed above.

It is apparent that during the preliminary design stage the graphical results are more revealing. A check of the deformed shape superimposed upon the un-deflected shape gives an immediate indication whether there are any errors. Stress (or forces) are reported as stress components of principal stresses in contour maps, where the various colors clearly reflect the behavior of the structure as indicated by the intensity of stress flow and the distribution of stresses.

DISPLAY THE MODEL :

View the model and results of analysis for a typical frame structure,

- **REACTION FORCES**
- **MEMBER AXIAL FORCES**
- **MEMBER BENDING MOMENT**
- **DEFLECTED SHAPE**

E. DESIGN

Any type of material can be defined and assigned to a frame element. You can run the analysis and get forces for that frame element (not stresses). If the material type is Steel or Cold-Formed Steel or Concrete or Aluminum then you can design the element using the built-in design post processors. For Wood you can put in member sections and get forces but not stresses. You only get stresses for frame elements in a design post processor, which SAP2000 does not have for wood.

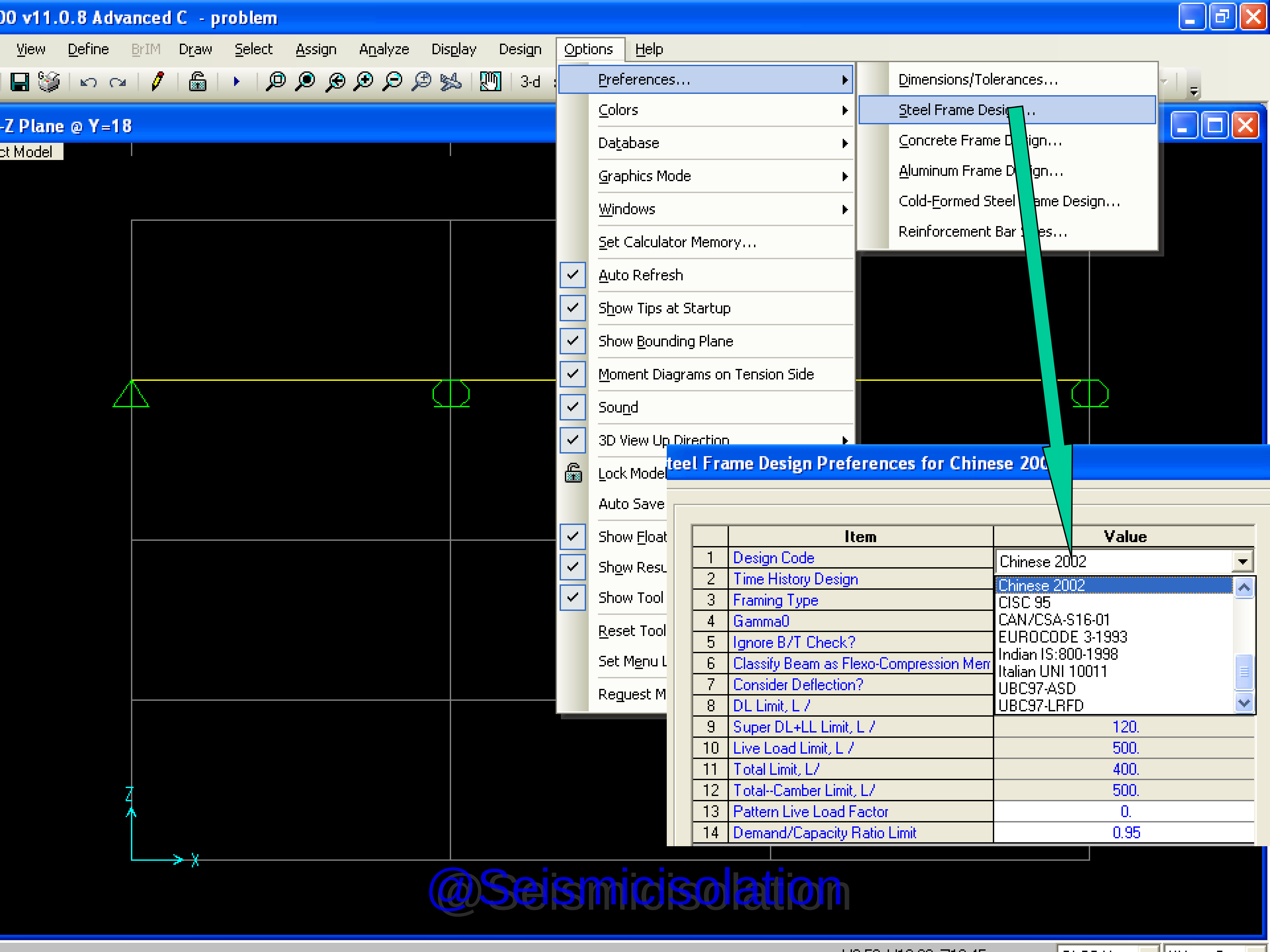
SAP2000 v. 8 is structured to support a wide variety of design codes for the automated design and check of aluminum, concrete and steel frame members. The program currently supports the following design codes (go Options/Preferences/Steel or Concrete):

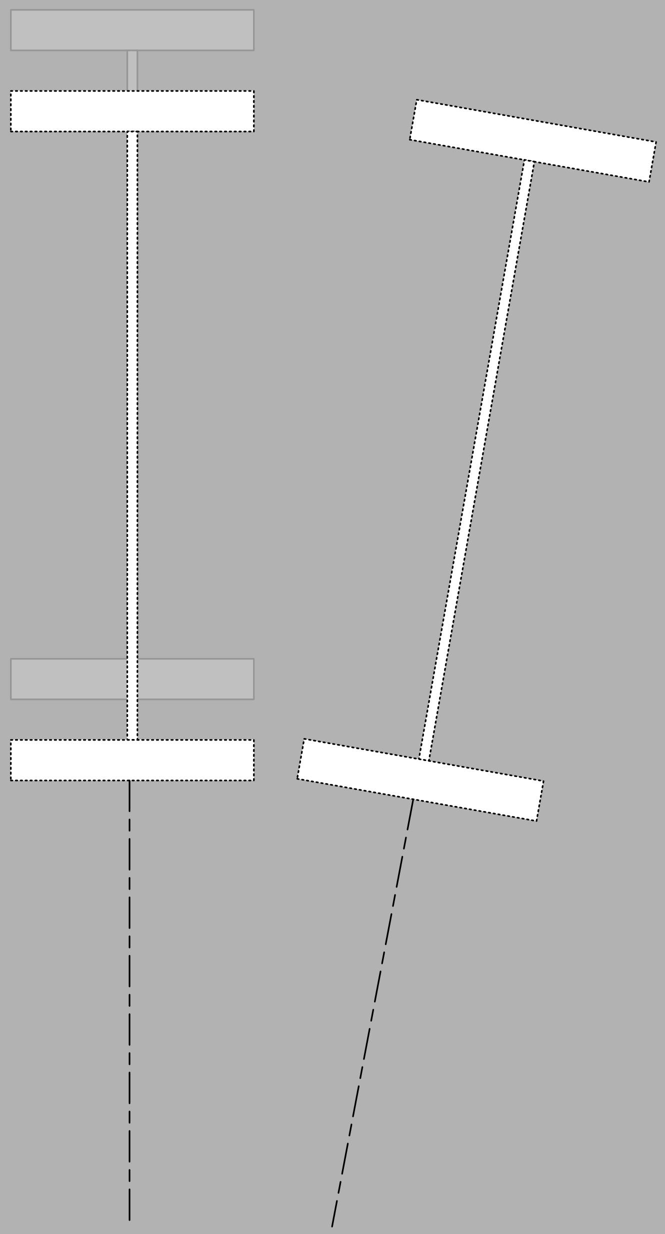
REINFORCED CONCRETE:

- **US: (ACI 318-99, AASHTO Concrete 97)**
- **Canadian: CSA-A23.3-94 (CSA 1994)**
- **British: (BS 8110-89)**
- **European: EUROCODE 2 – 1992 (CEN 1992)**
- **New Zealand design code: NZS 3101-95**
- **Indian IS**
- **Italian DM**
- **Mexican RCDF 2001**

STEEL:

- **US: (AISC-ASD89, AISC-LRFD93, AASHTO Steel 97)**
- **Canadian: CAN/CSA-S16.1-94 (CISC 1995)**
- **British: BS 5950 (BSI 1990)**
- **European: EUROCODE 3 – 1993 (CEN 1993)**
- **Italian UNI**


00 v11.0.8 Advanced C - problem
View Define BrIM Draw Select Assign Analyze Display Design Options Help
-Z Plane @ Y=18
Preferences...
Colors
Database
Graphics Mode
Windows
Set Calculator Memory...
Auto Refresh
Show Tips at Startup
Show Bounding Plane
Moment Diagrams on Tension Side
Sound
3D View Up Direction
Lock Model
Auto Save
Dimensions/Tolerances...
Steel Frame Design...
Concrete Frame Design...
Aluminum Frame Design...
Cold-Formed Steel Frame Design...
Reinforcement Bar Sizes...
Steel Frame Design Preferences for Chinese 200
Item
Value
1 Design Code
2 Time History Design
3 Framing Type
4 Gamma0
5 Ignore B/T Check?
6 Classify Beam as Flexo-Compression Mem
7 Consider Deflection?
8 DL Limit, L /
9 Super DL+LL Limit, L / 120.
10 Live Load Limit, L / 500.
11 Total Limit, L/ 400.
12 Total--Camber Limit, L/ 500.
13 Pattern Live Load Factor 0.
14 Demand/Capacity Ratio Limit 0.95
Chinese 2002
CISC 95
CAN/CSA-S16-01
EUROCODE 3-1993
Indian IS:800-1998
Italian UNI 10011
UBC97-ASD
UBC97-LRFD



Modeling Steel Members using SAP2000 (see also Appendix A)

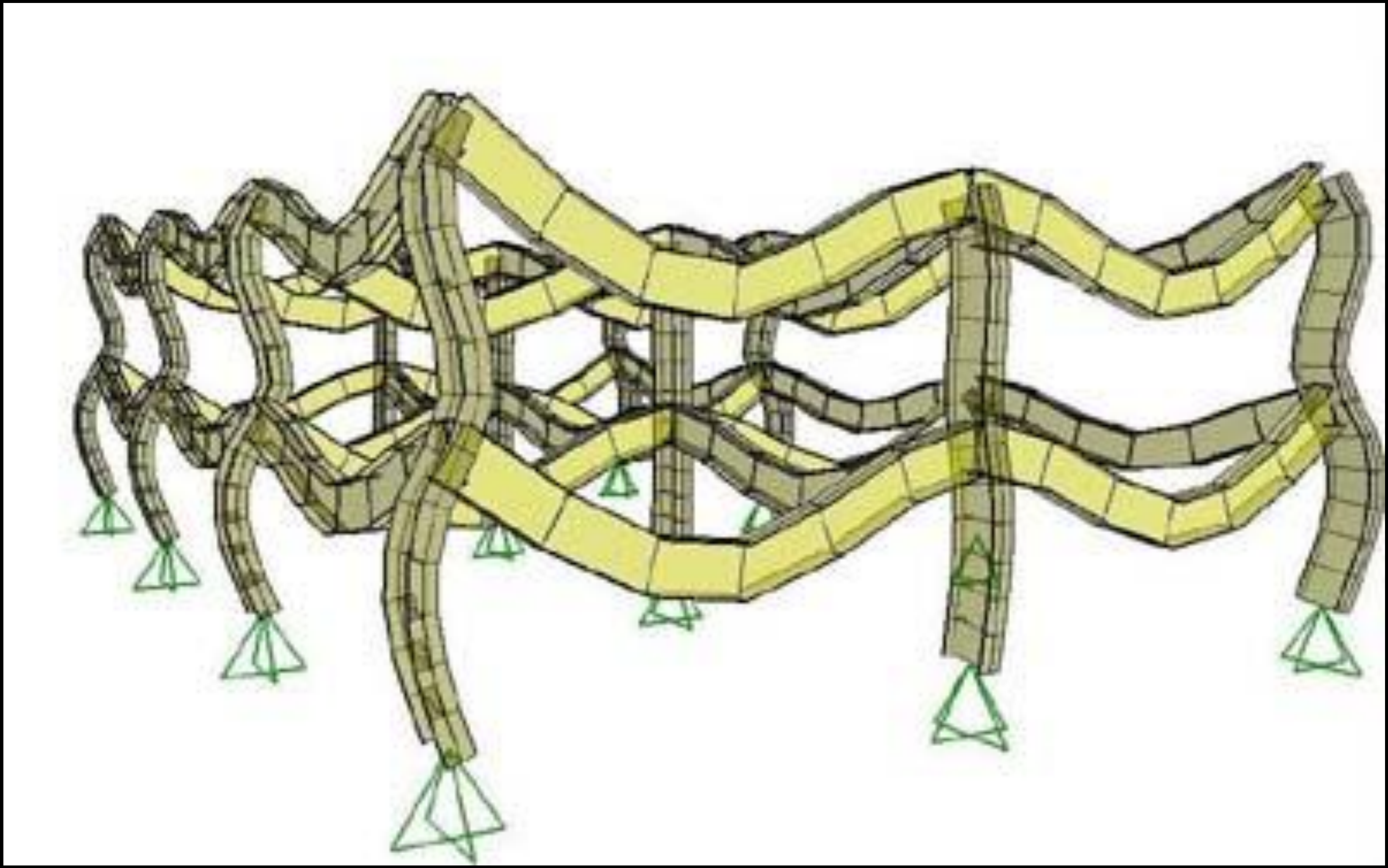
SAP2000 assumes by default that **frame elements** (i.e., beams and columns) **are laterally unsupported for their full length**. But beams are generally laterally supported by the floor structure (Fig. 4.1). Therefore, assume an unsupported length of say ***Lb*** = *2 ft* for preliminary design purposes, or when in doubt, take the spacing between the filler beams. For example, for a beam span of, *L* = *24 ft*, assume an **unbraced length ratio** about the minor axis of ***Lb /L*** = *2 ft/24 ft* = *0.083*, or say ***0.1***; that is, take the minor direction unbraced length as **10%** of the actual span length.

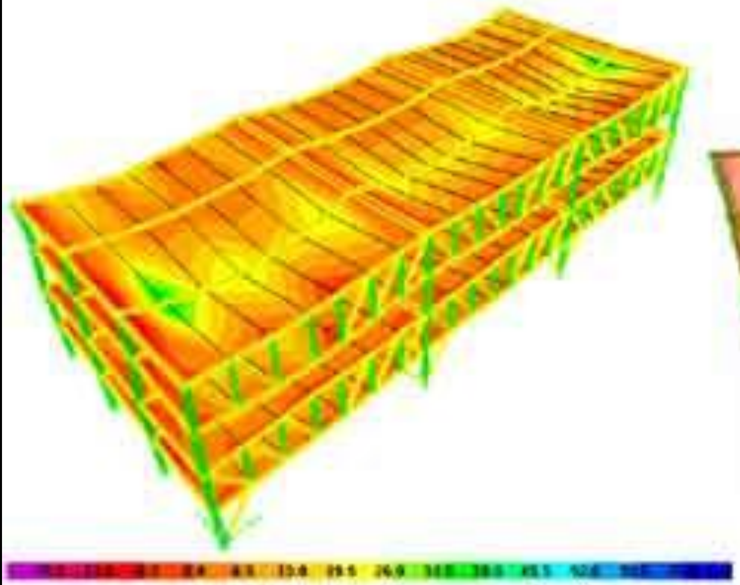
F. MODIFY MEMBER PROPERTIES

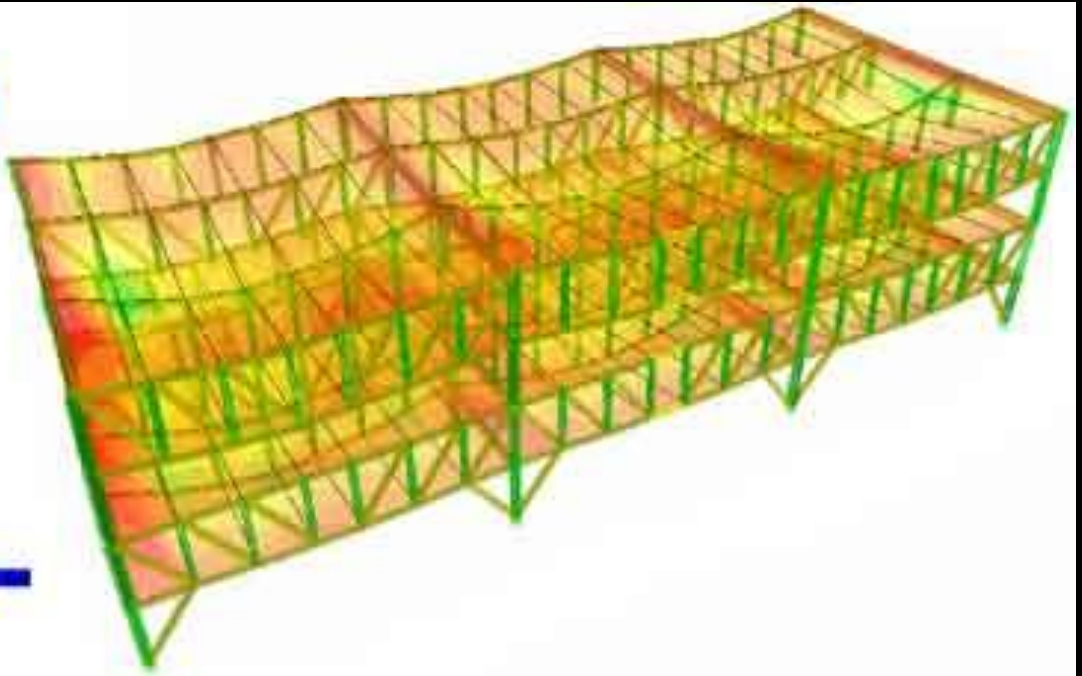
G. RE-ANALYZE and RE-DESIGN with updated model

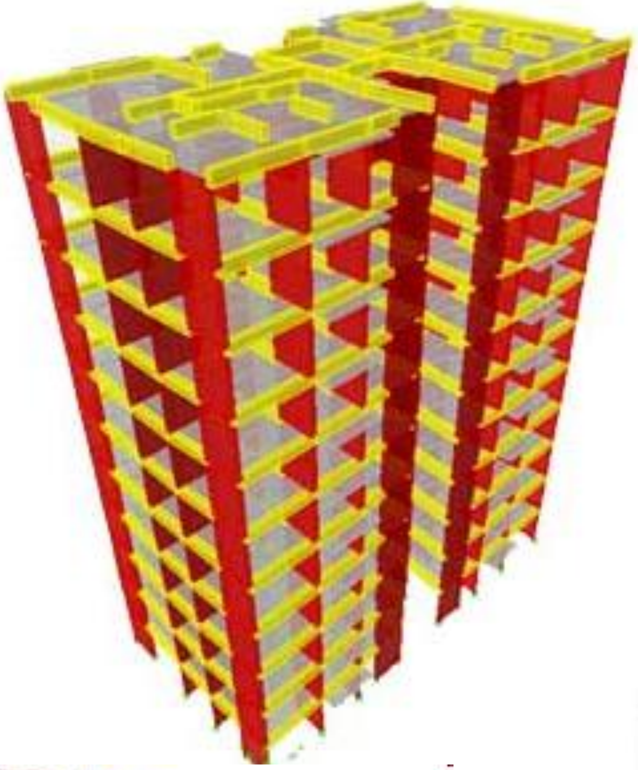
The design changes must be taken into account in the analysis portion that is the redistribution of member forces due to change of stiffness, therefore a re-run of the analysis is necessary

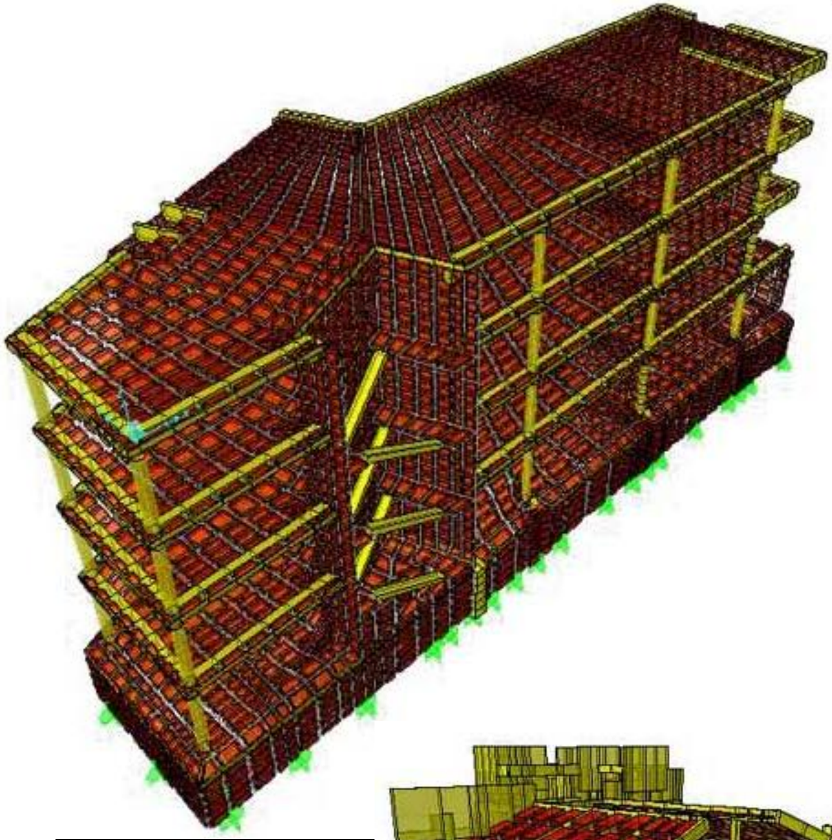
H. COMPUTER OUTPUT: display results (printing and plotting)

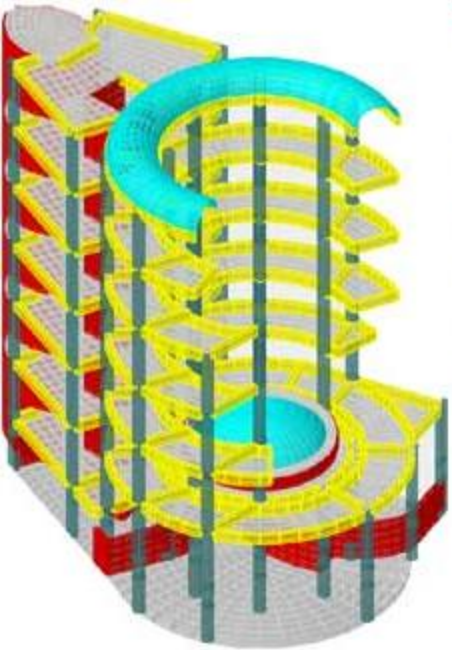


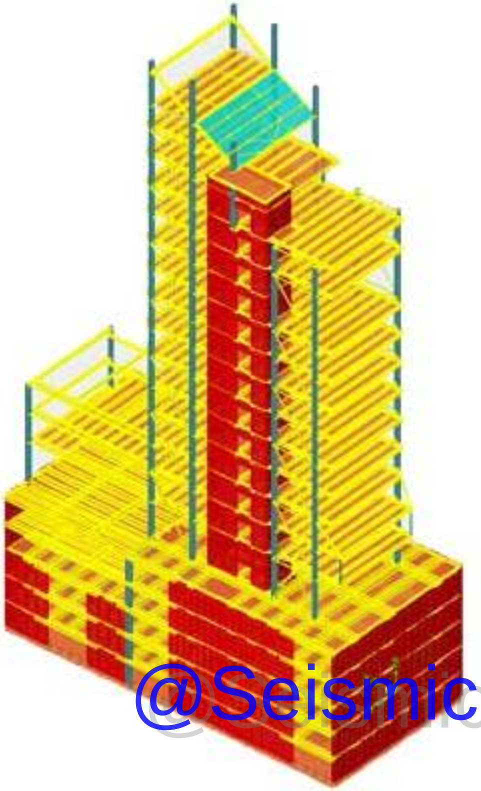


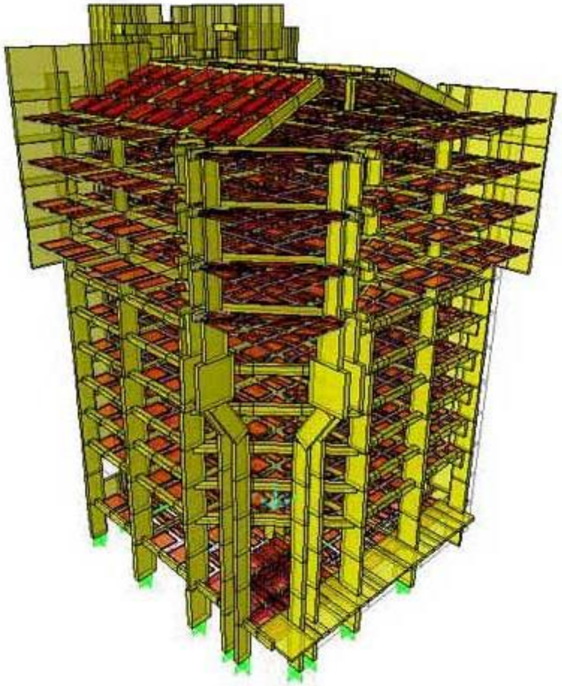


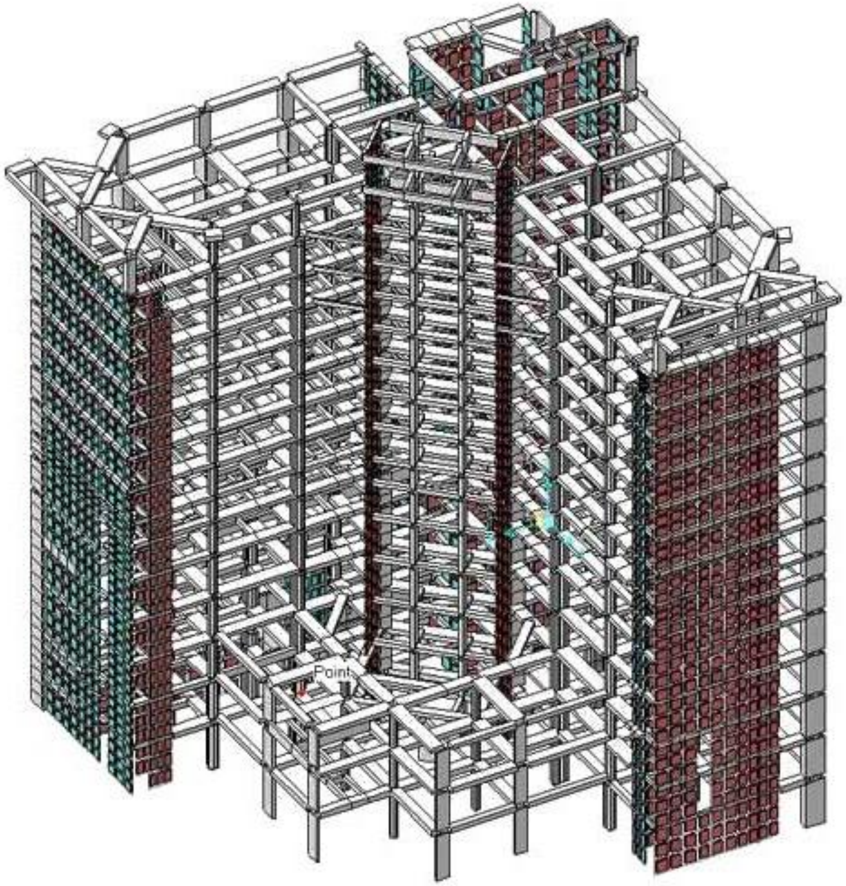


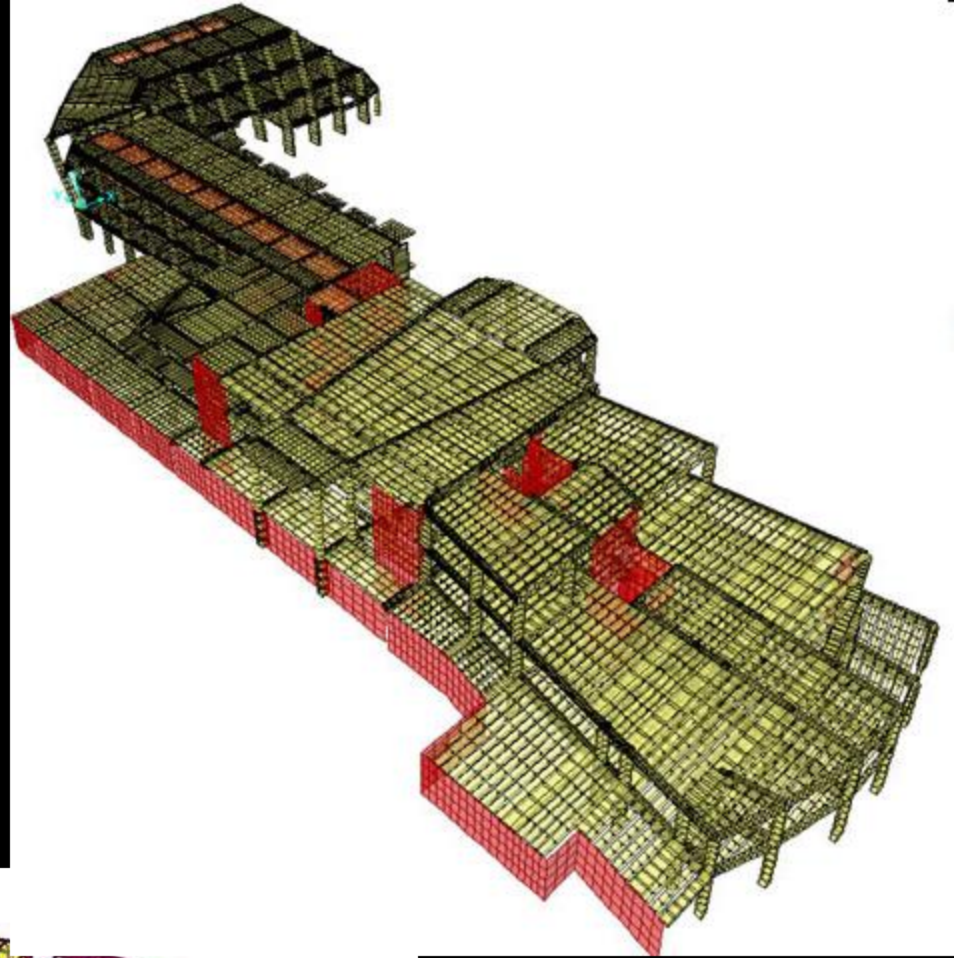


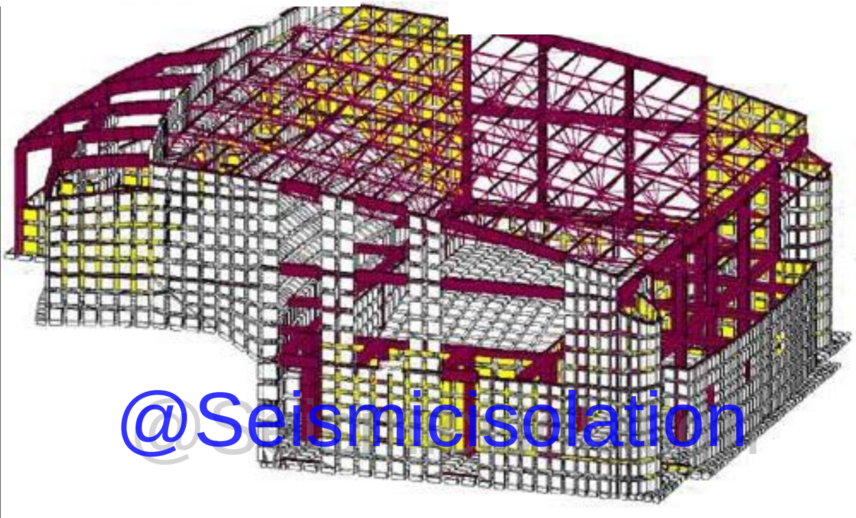


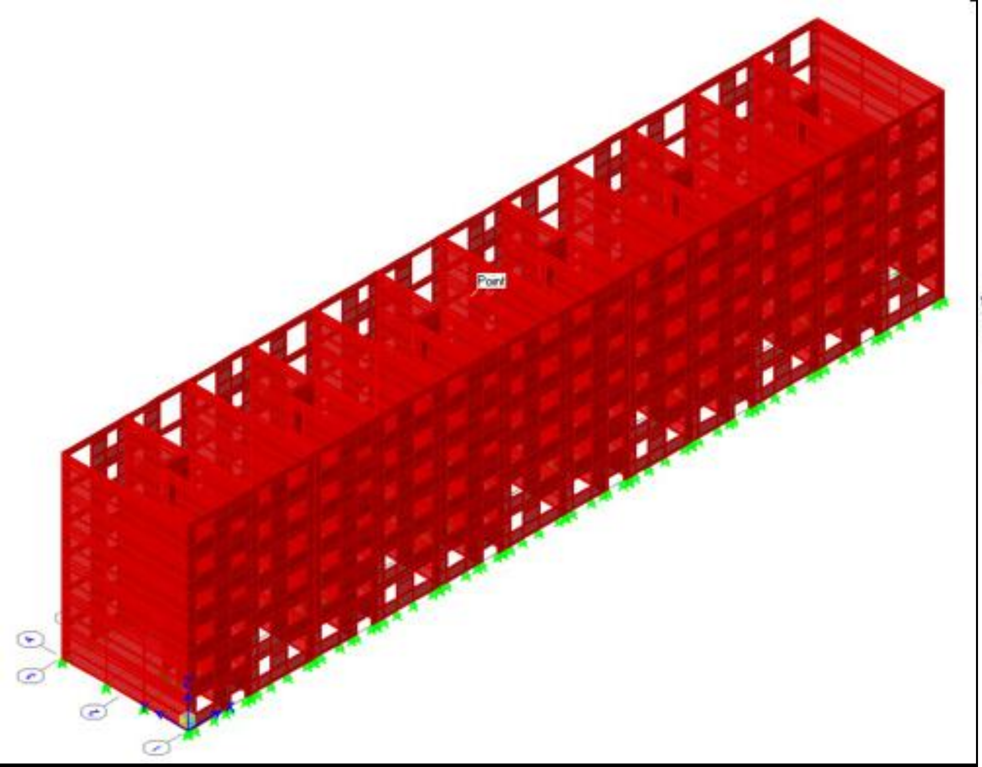


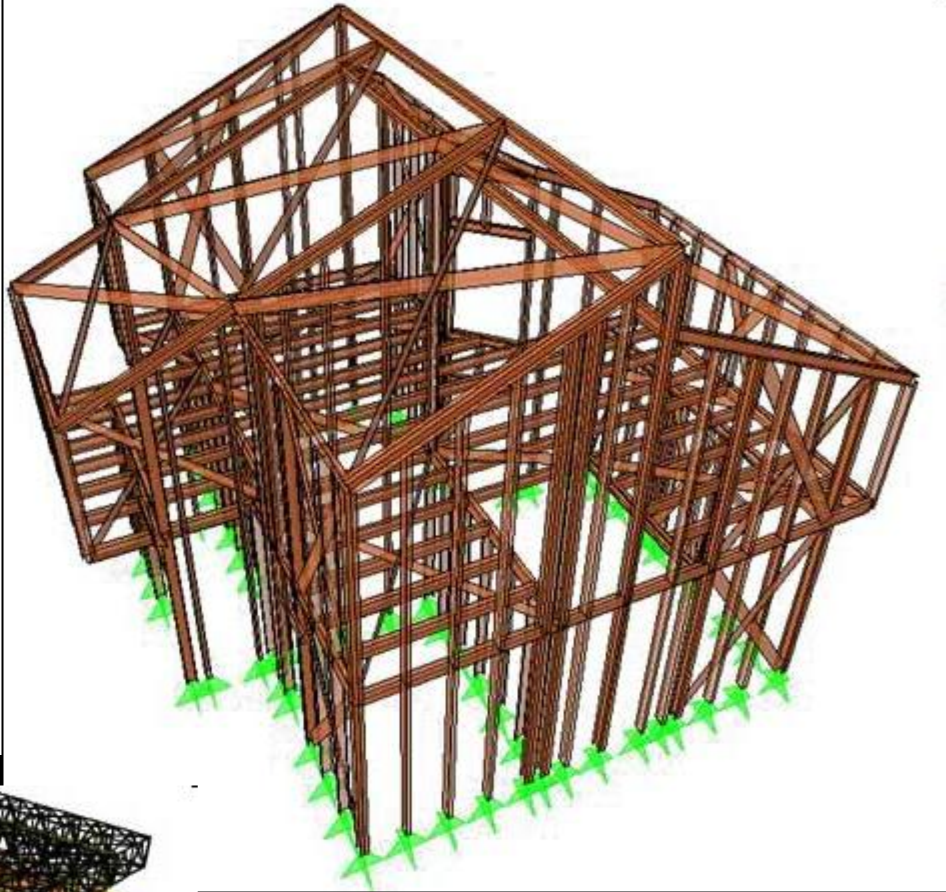


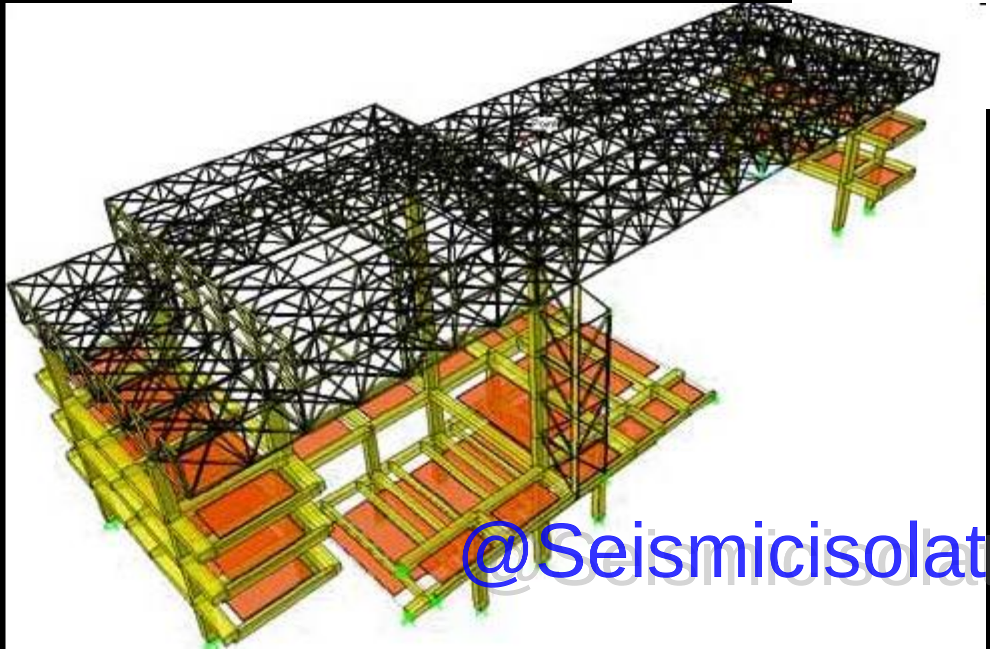

Point





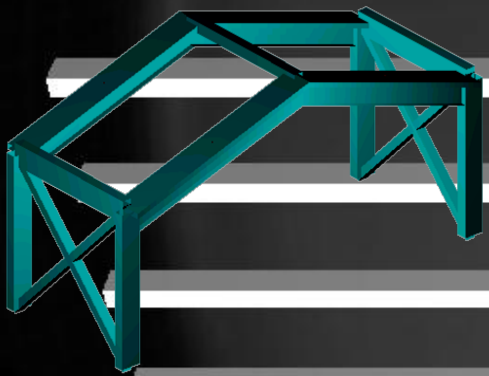

Point



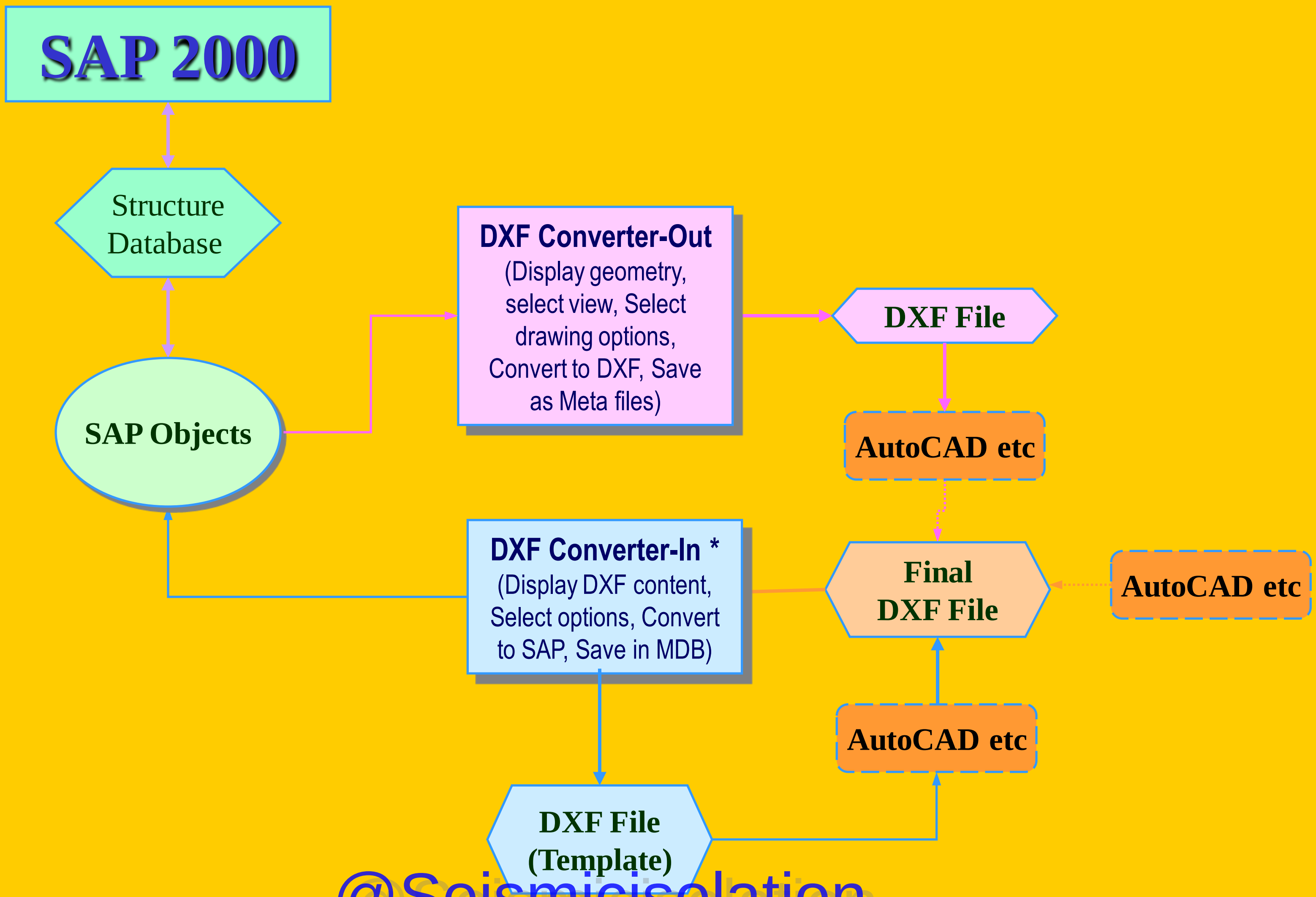


New Productivity Tools

DXF Interface and Reporter

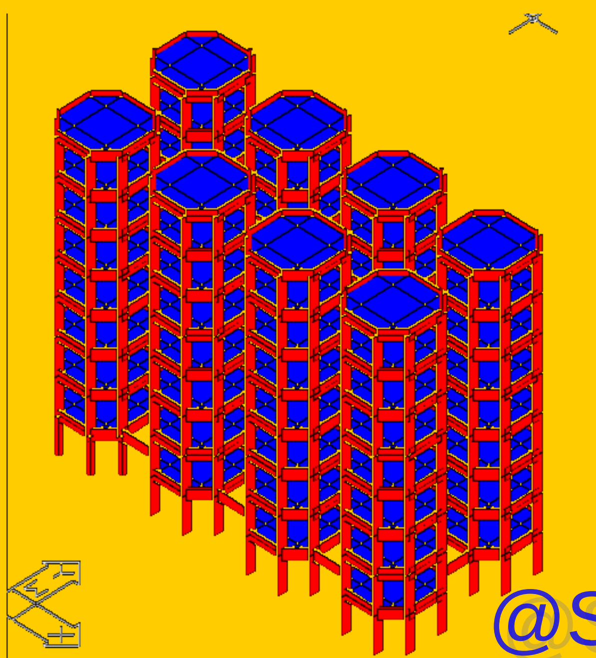


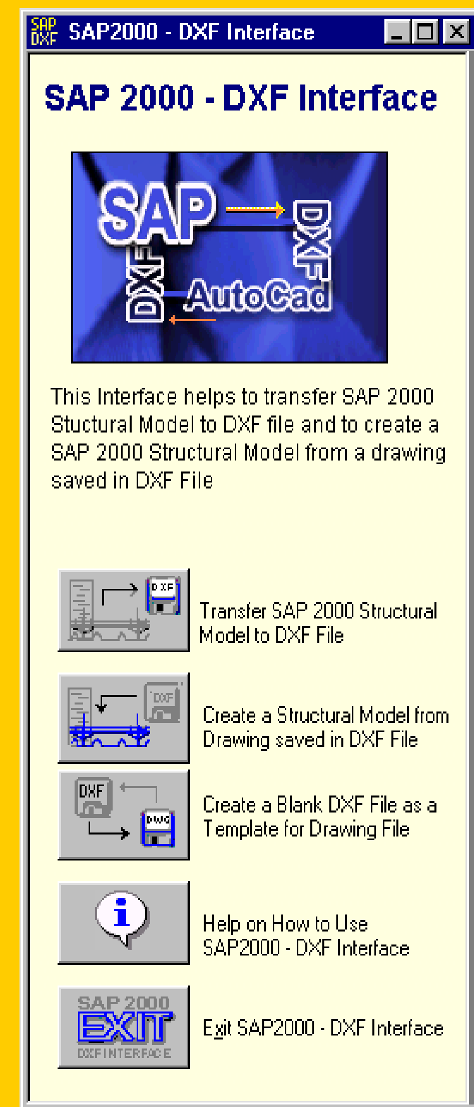
SAP200-DXF-AutoCAD Cycle



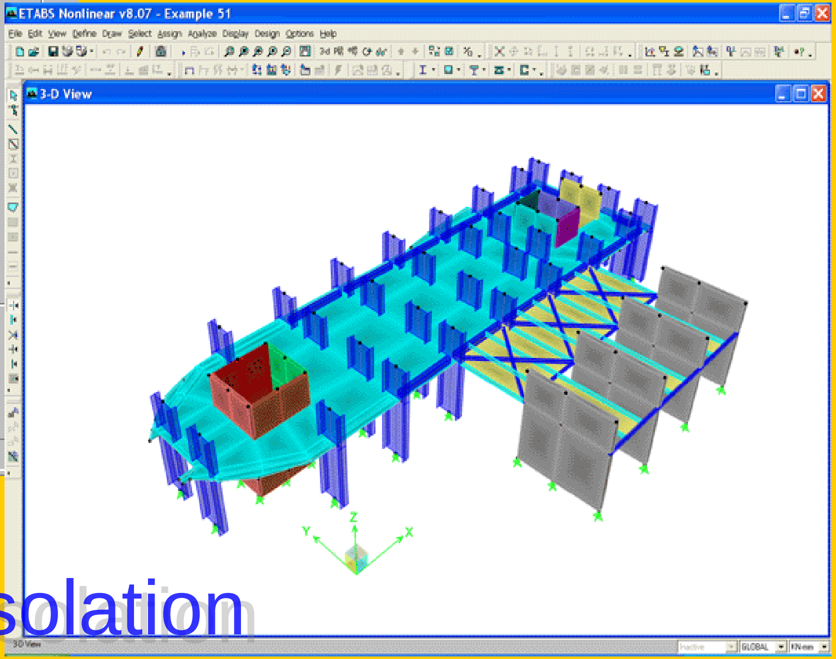
Transfer Model To AutoCAD

- Complete control of the elements and attributes to be transferred
- Generate wire-frame or full solid models





Get Model From AutoCAD



Complex geometry may be modeled first in SketchUp or Rhinoceros and the imported to AutoCAD.

Import AutoCAD into SAP2000.

For example for a frame system frame:

1-adjust your units an coordinates on AutoCAD

2-create layer for your lines. because when you export your shape on sap 2000 it will be important.

3-draw your shape only using line.

4-save your shape as a dxf file.

5-go to sap 2000 and file menu and go to import

6-you will see a list about what kind of drawing will import from AutoCAD to SAP2000

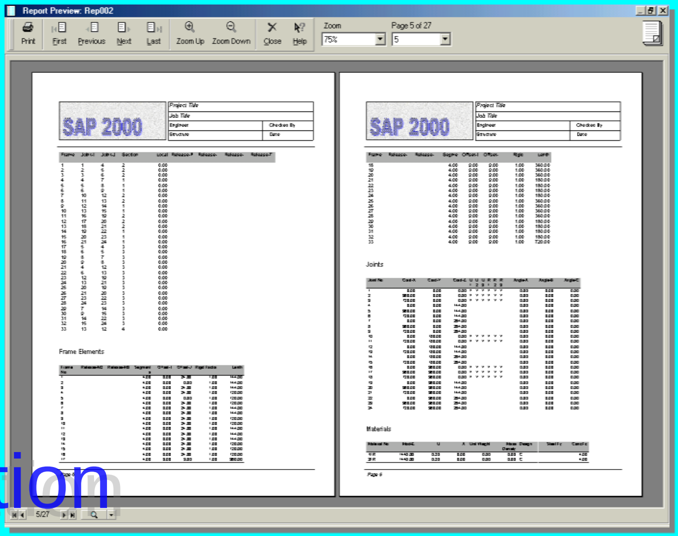
7-answer all questions and click ok.

you will see that all shape in AutoCAD you can import in SAP2000.

SAPReporter

- Create fully customized and formatted reports for selected items including text, tables and graphics.

- Output can be
  - *Previewed on screen*
  - *Printed on paper*
  - *Saved in RTF files*
  - *Saved in HTML format for viewing in web browsers.*



Structure Systems
Structure Behavior

A.. PLANAR STRUCTURES

Axial force systems

- TRUSSES
- STAYED STRUCTURES

Flexural force systems

- BEAMS

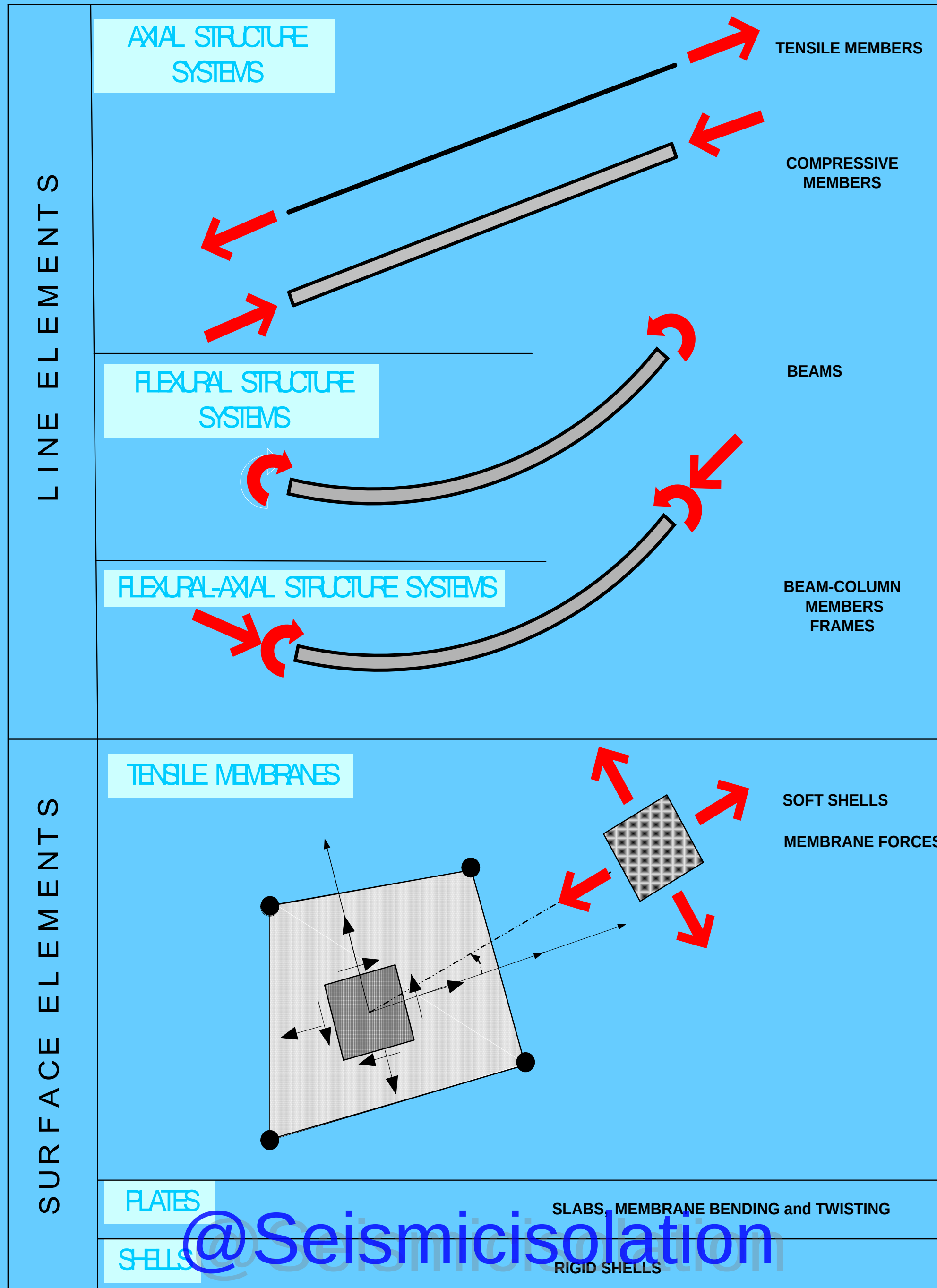
Flexural-axial force systems

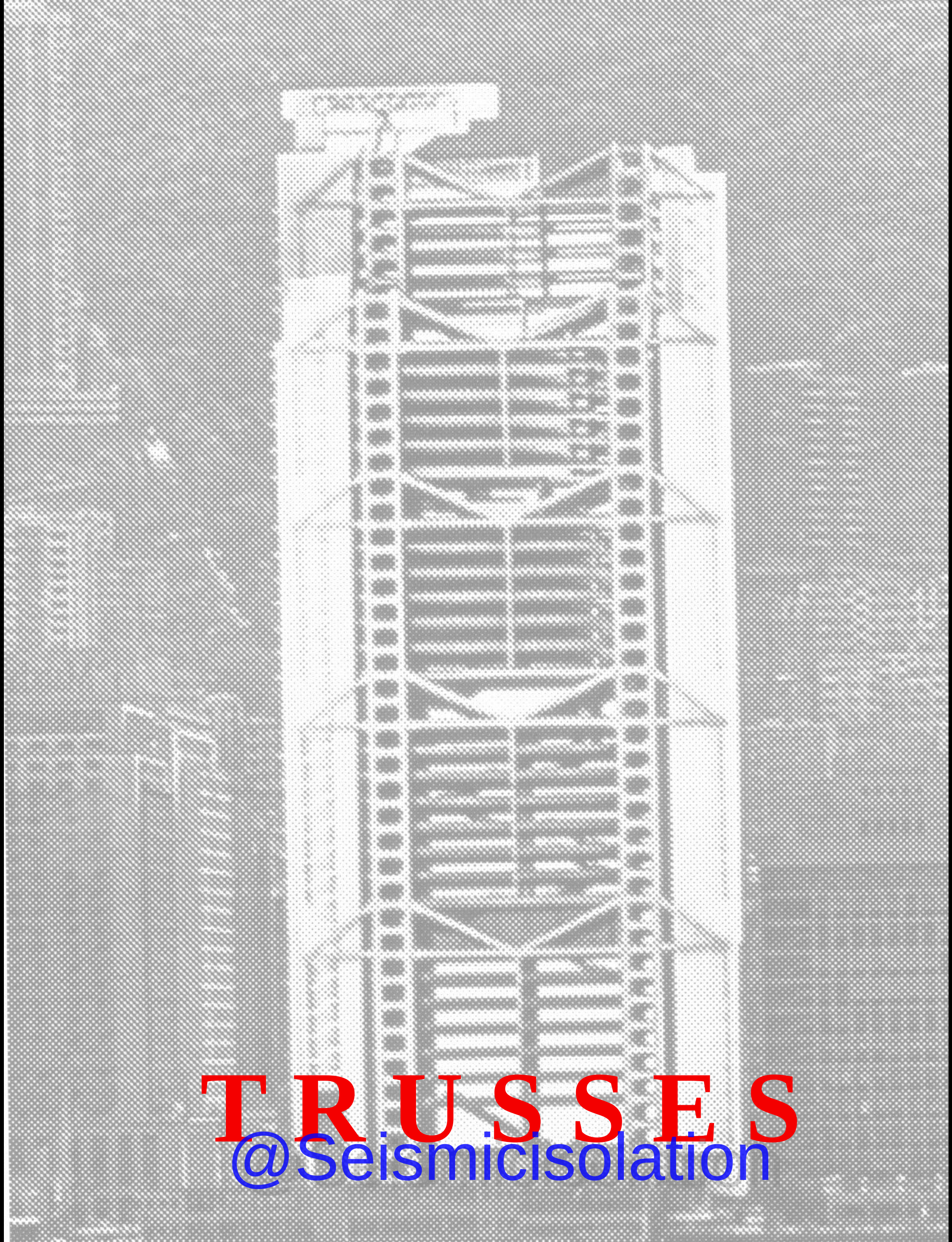
- SKELETONS: frames, arches

B. SPATIAL STRUCTURES

- LATERAL STABILITY OF BUILDINGS
- SPACE FRAMES
- CABLE STRUCTURES
- SURFACE STRUCTURES

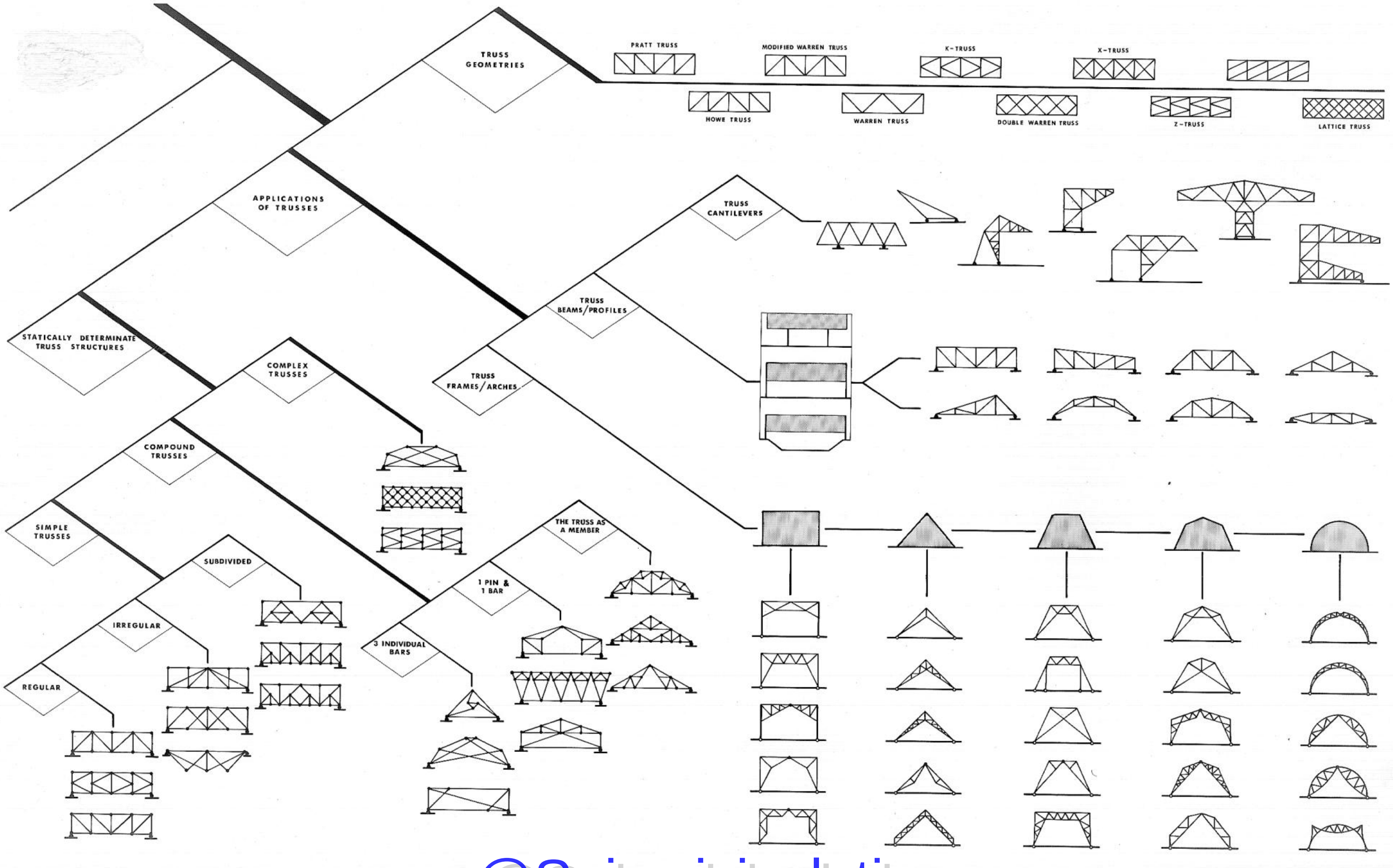
C. CASE STUDY


LINE ELEMENTS
AXIAL STRUCTURE SYSTEMS
TENSILE MEMBERS
COMPRESSIVE MEMBERS
FLEXURAL STRUCTURE SYSTEMS
BEAMS
FLEXURAL-AXIAL STRUCTURE SYSTEMS
BEAM-COLUMN MEMBERS FRAMES
SURFACE ELEMENTS
TENSILE MEMBRANES
SOFT SHELLS
MEMBRANE FORCES
PLATES
SLABS, MEMBRANE BENDING and TWISTING
SHELLS
RIGID SHELLS



T R U S S E S

Hongkong Bank, Hong Kong, 1985, Foster/Arup


TRUSS GEOMETRIES
PRATT TRUSS
MODIFIED WARREN TRUSS
K-TRUSS
X-TRUSS
HOWE TRUSS
WARREN TRUSS
DOUBLE WARREN TRUSS
Z-TRUSS
LATTICE TRUSS
APPLICATIONS OF TRUSSES
TRUSS CANTILEVERS
TRUSS BEAMS/PROFILES
TRUSS FRAMES/ARCHES
STATICALLY DETERMINATE TRUSS STRUCTURES
COMPLEX TRUSSES
COMPOUND TRUSSES
THE TRUSS AS A MEMBER
1 PIN & 1 BAR
3 INDIVIDUAL BARS
SIMPLE TRUSSES
SUBDIVIDED
IRREGULAR
REGULAR

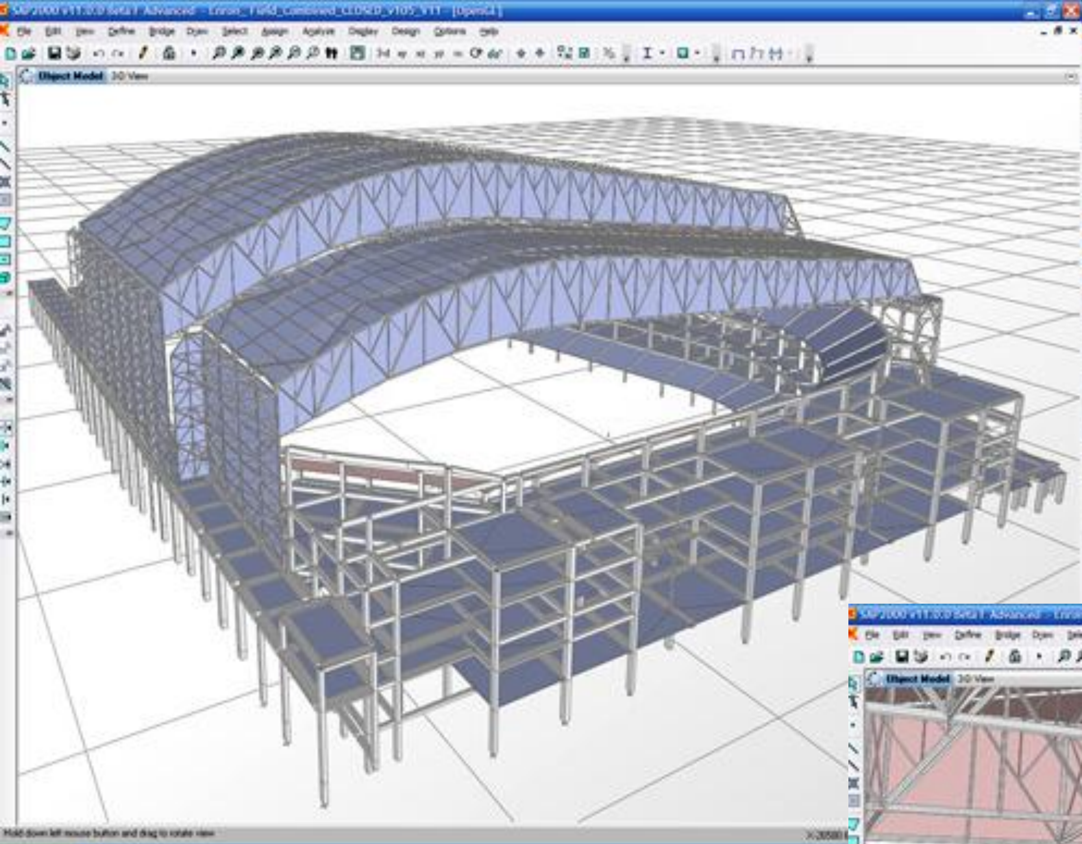
John Hancock Center (100 stories, 344 m), Chicago, 1968, Bruce Graham/ Fazlur Kahn of SOM

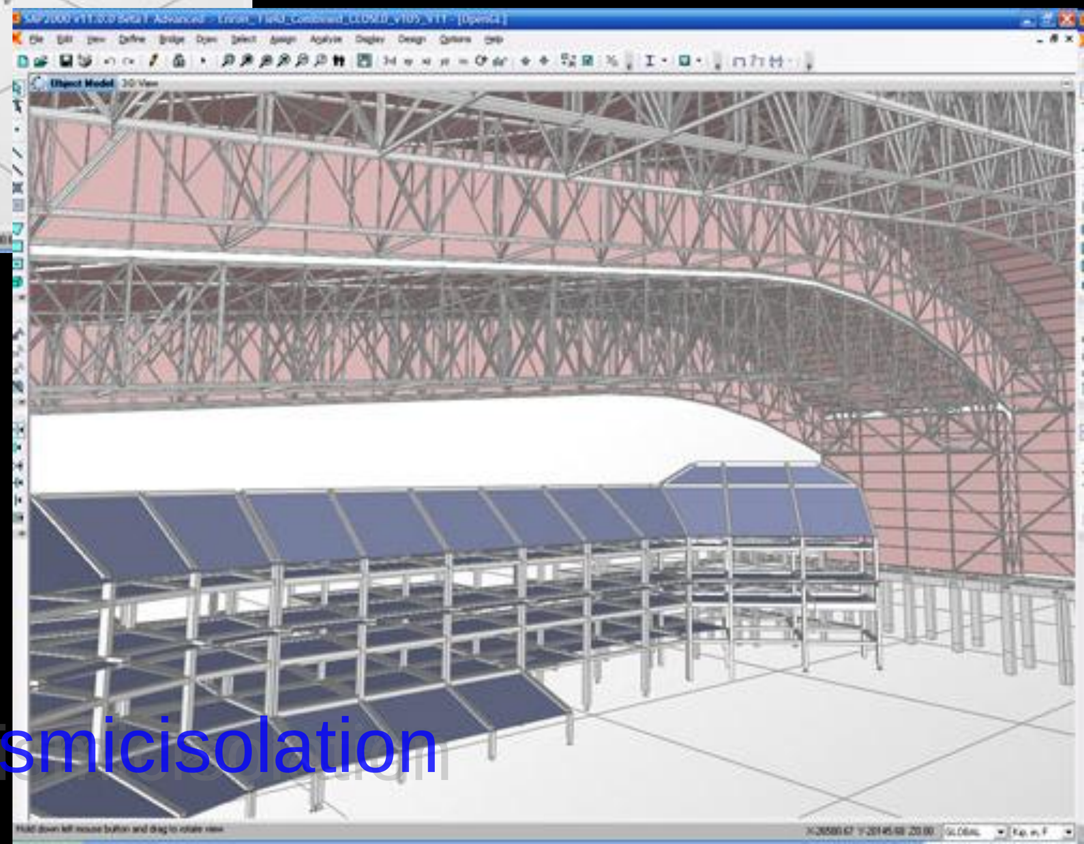
Library Gainesville, FL

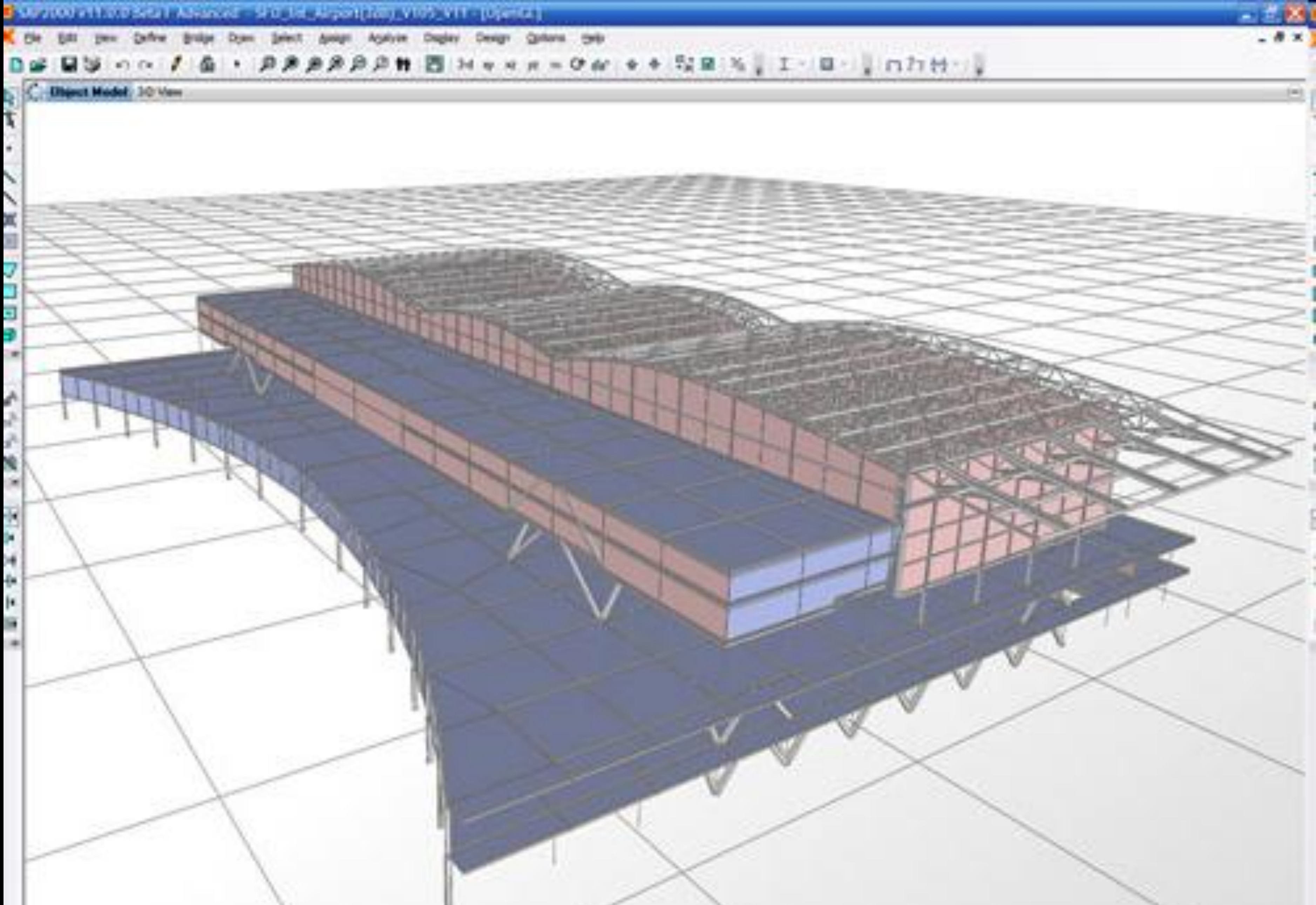
Merzedes-Benz Zentrale, Berlin, 2000, Lamm, Weber, Donath und Partne

Sony Center, **Potzdamer Platz, Berlin, 2000, Helmut Jahn Arch., Ove Arup USA Struct. Eng**

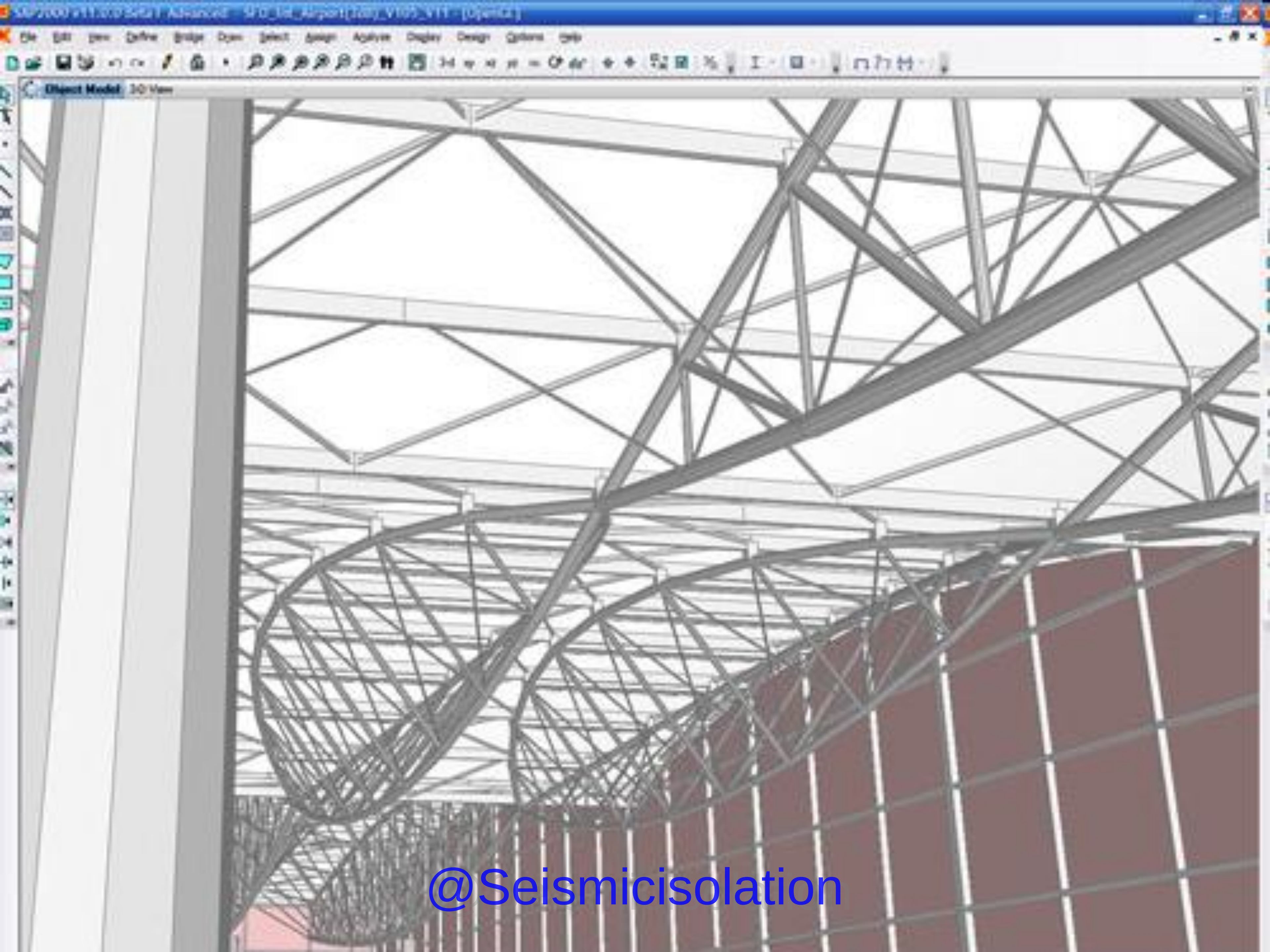

24 9 2001

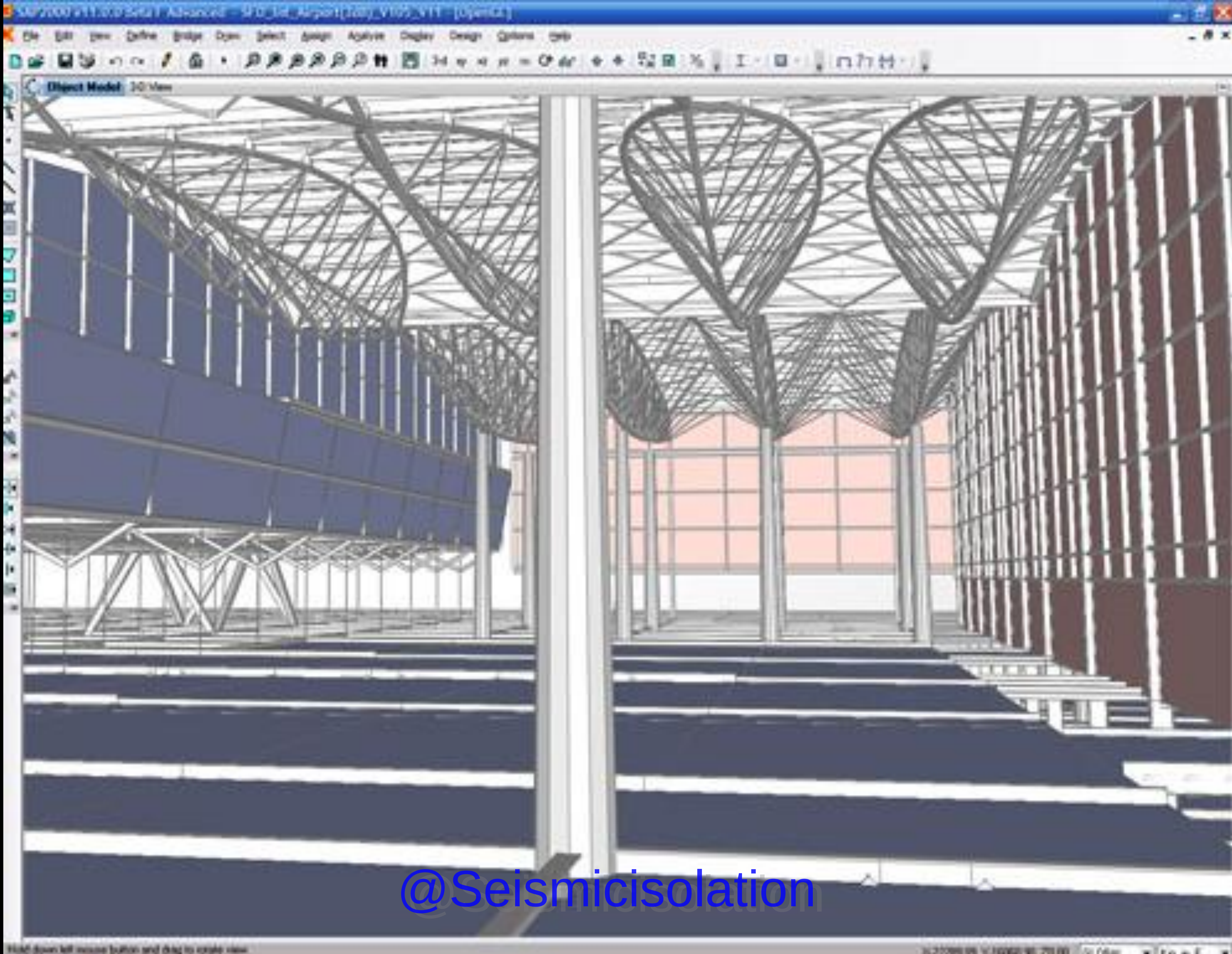


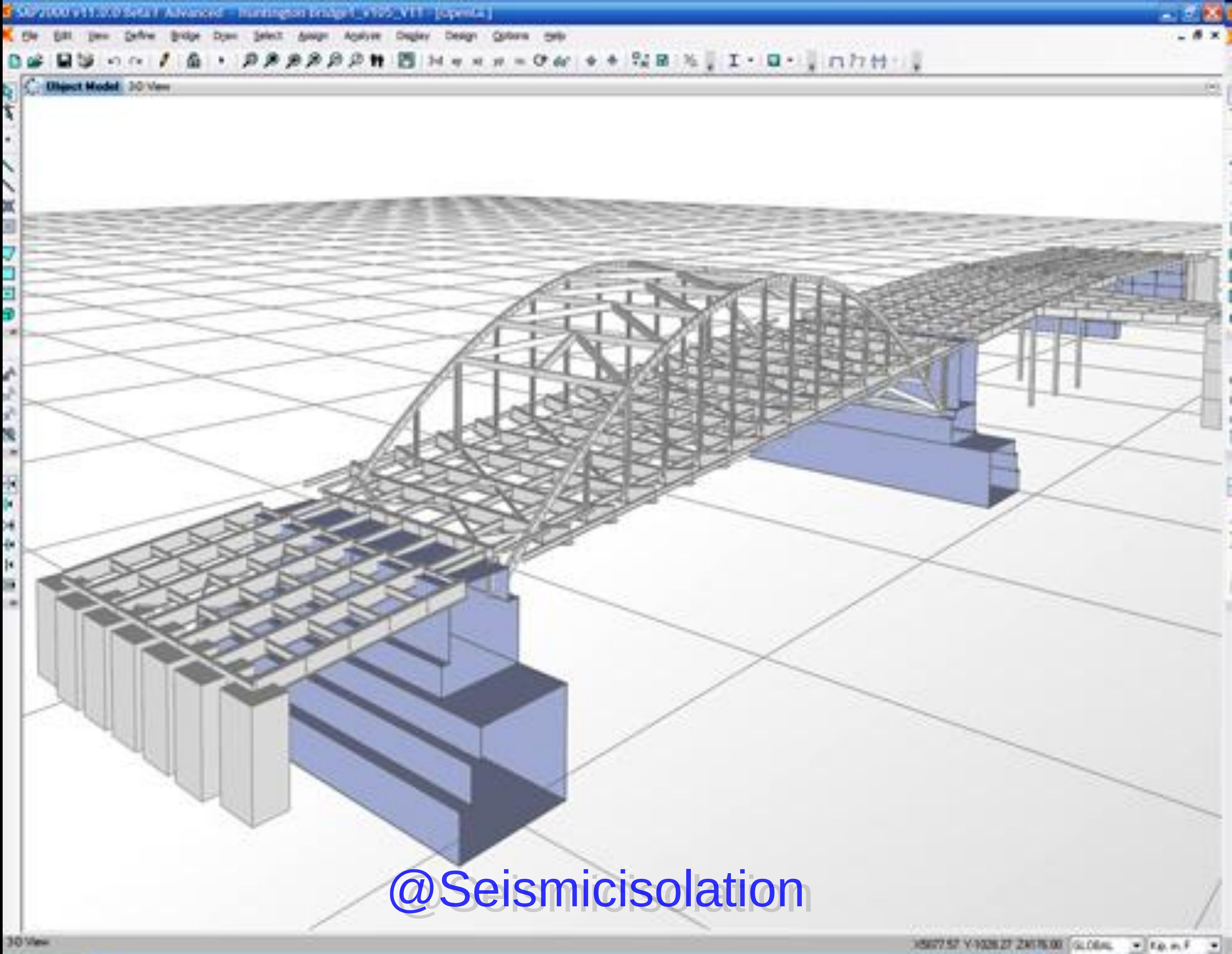


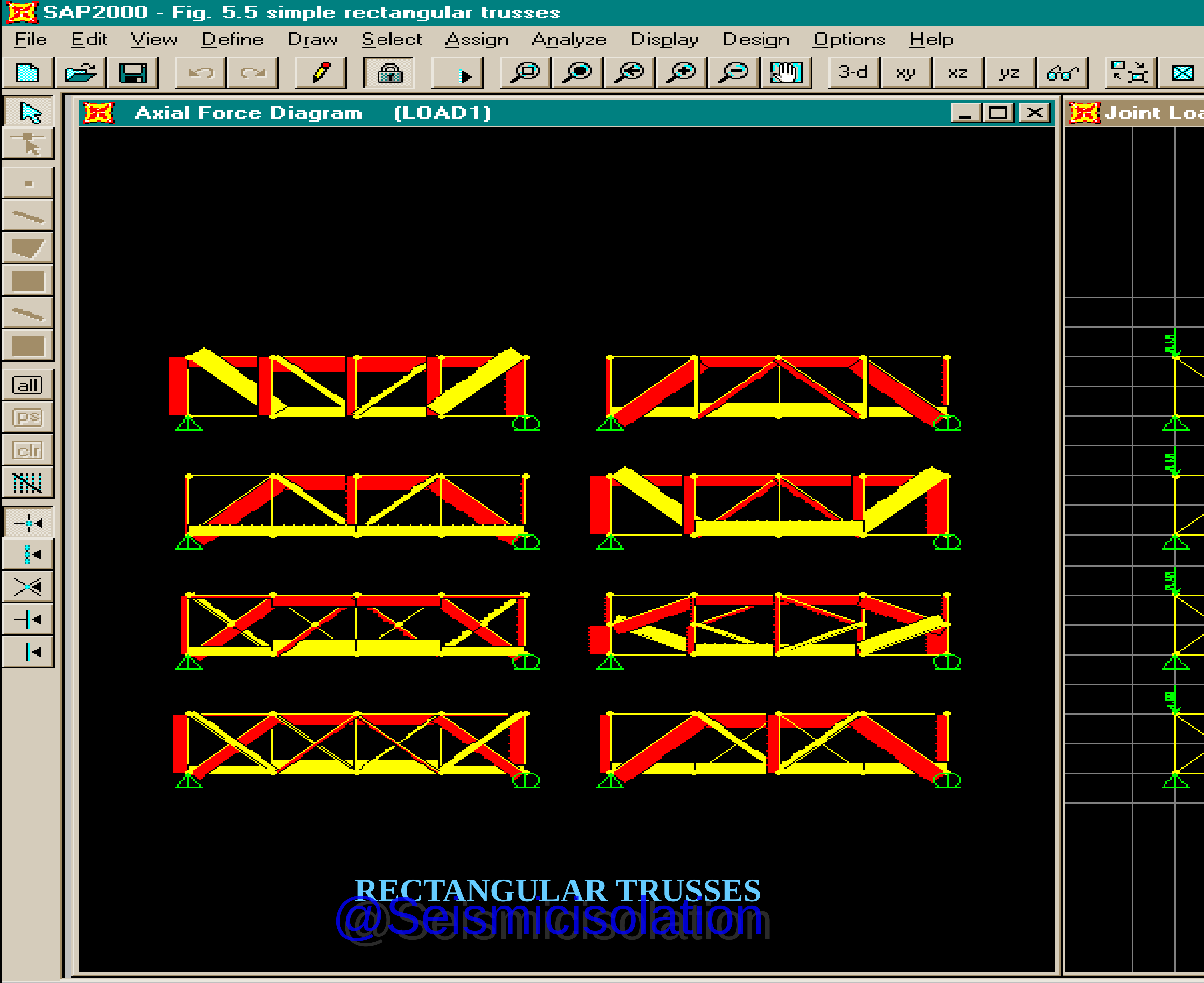


San Francisco Terminal, 2001, SOM, SAP2000 drawing

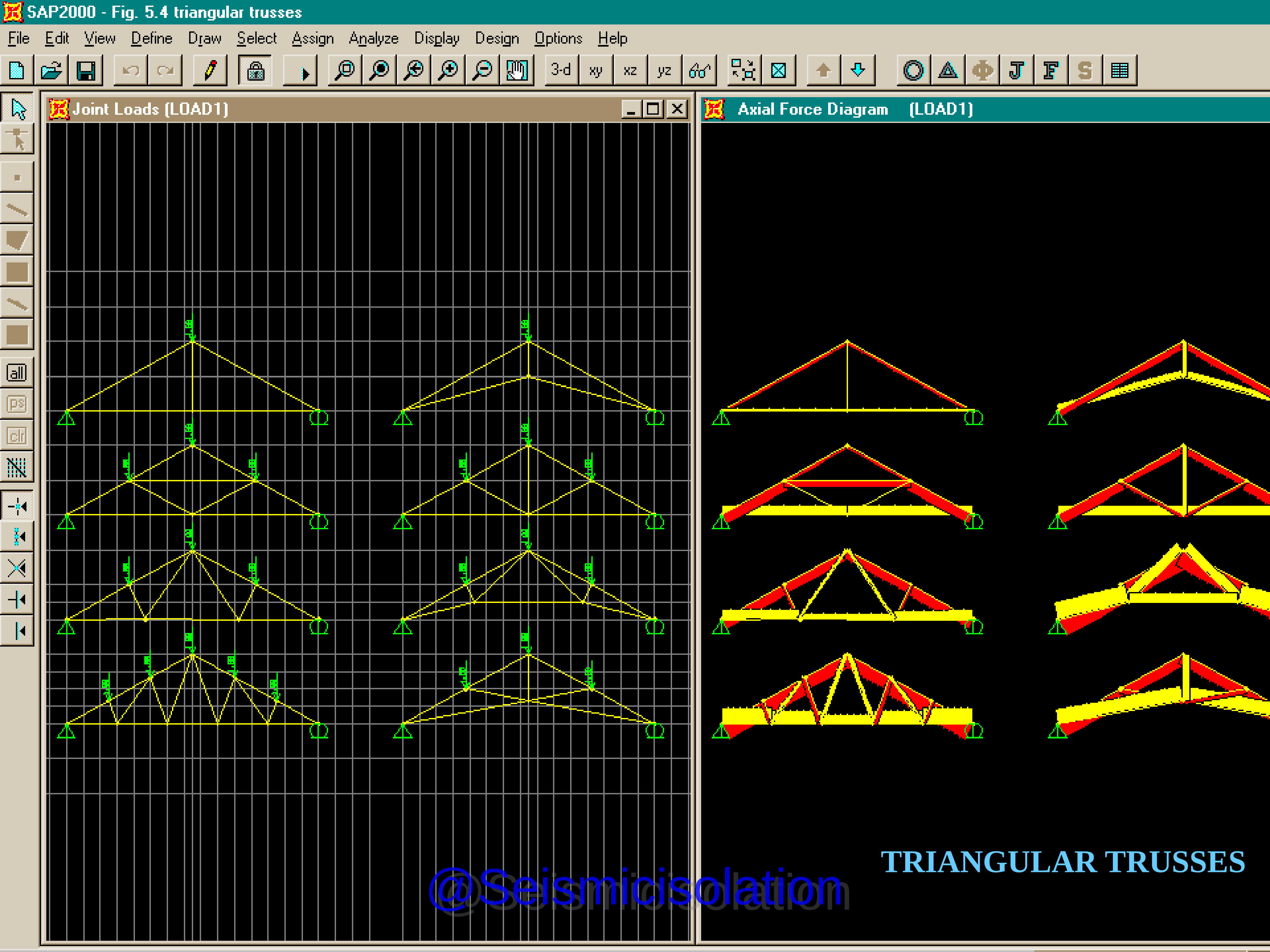

@Seismicisolation

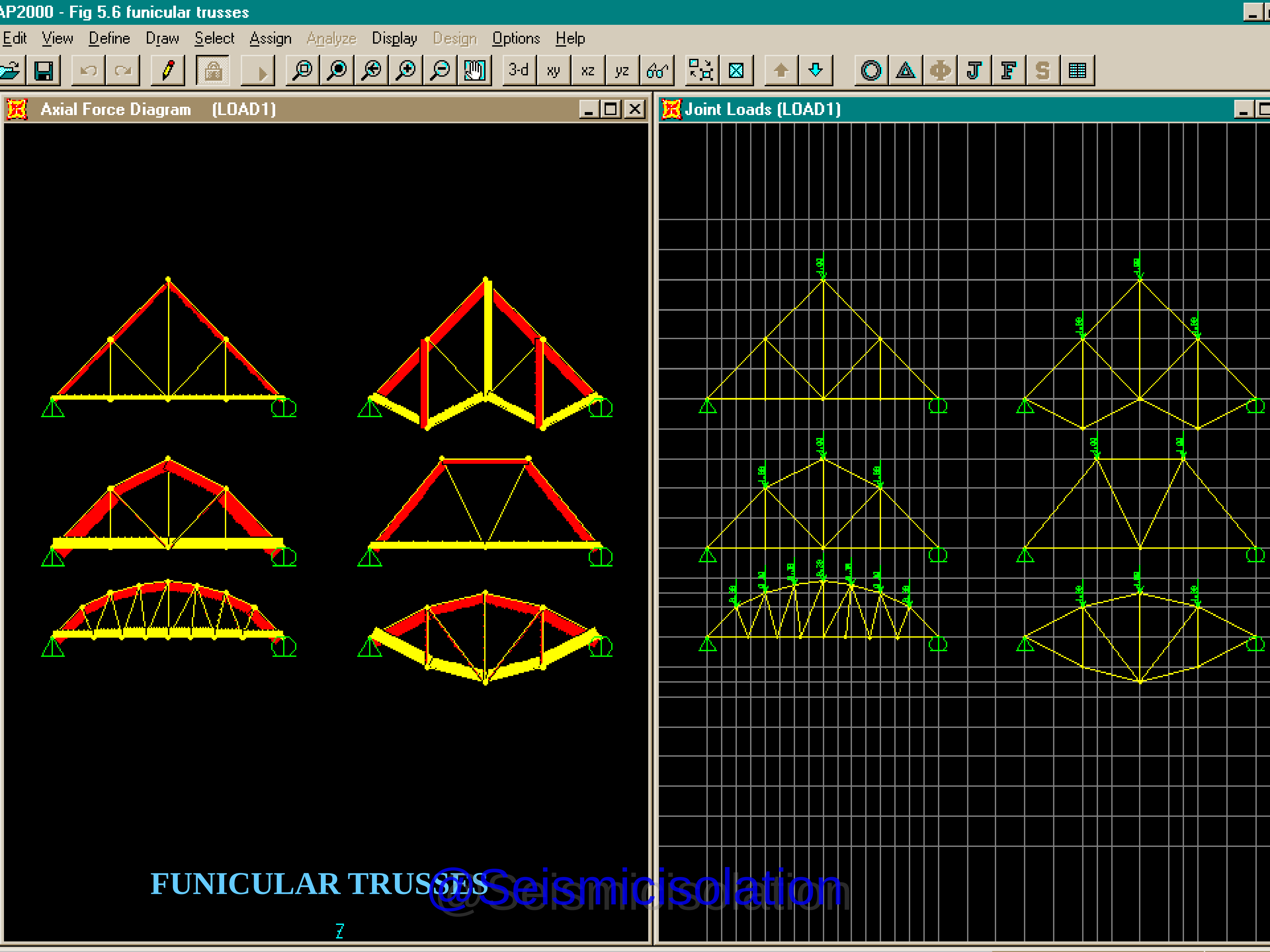


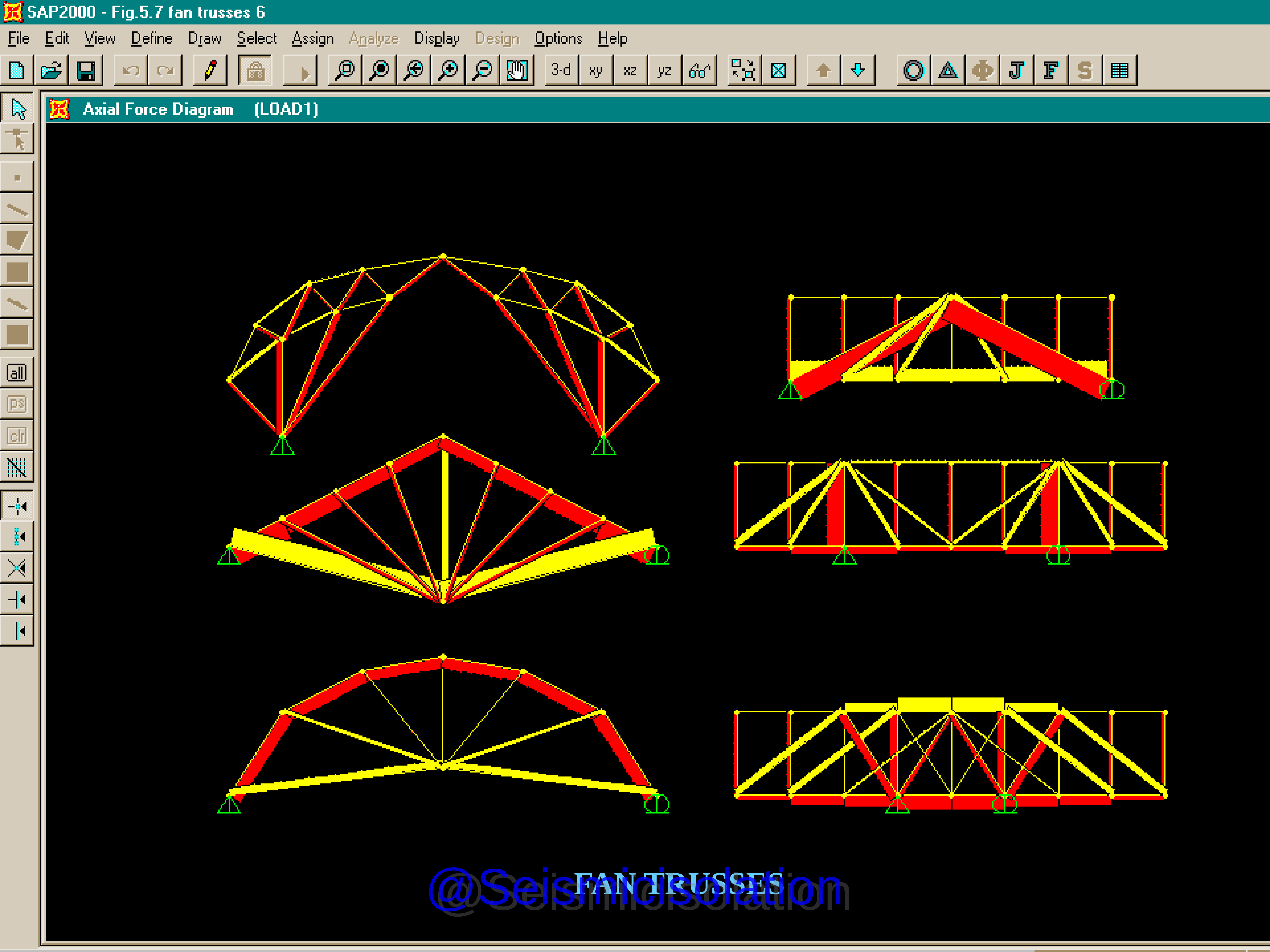


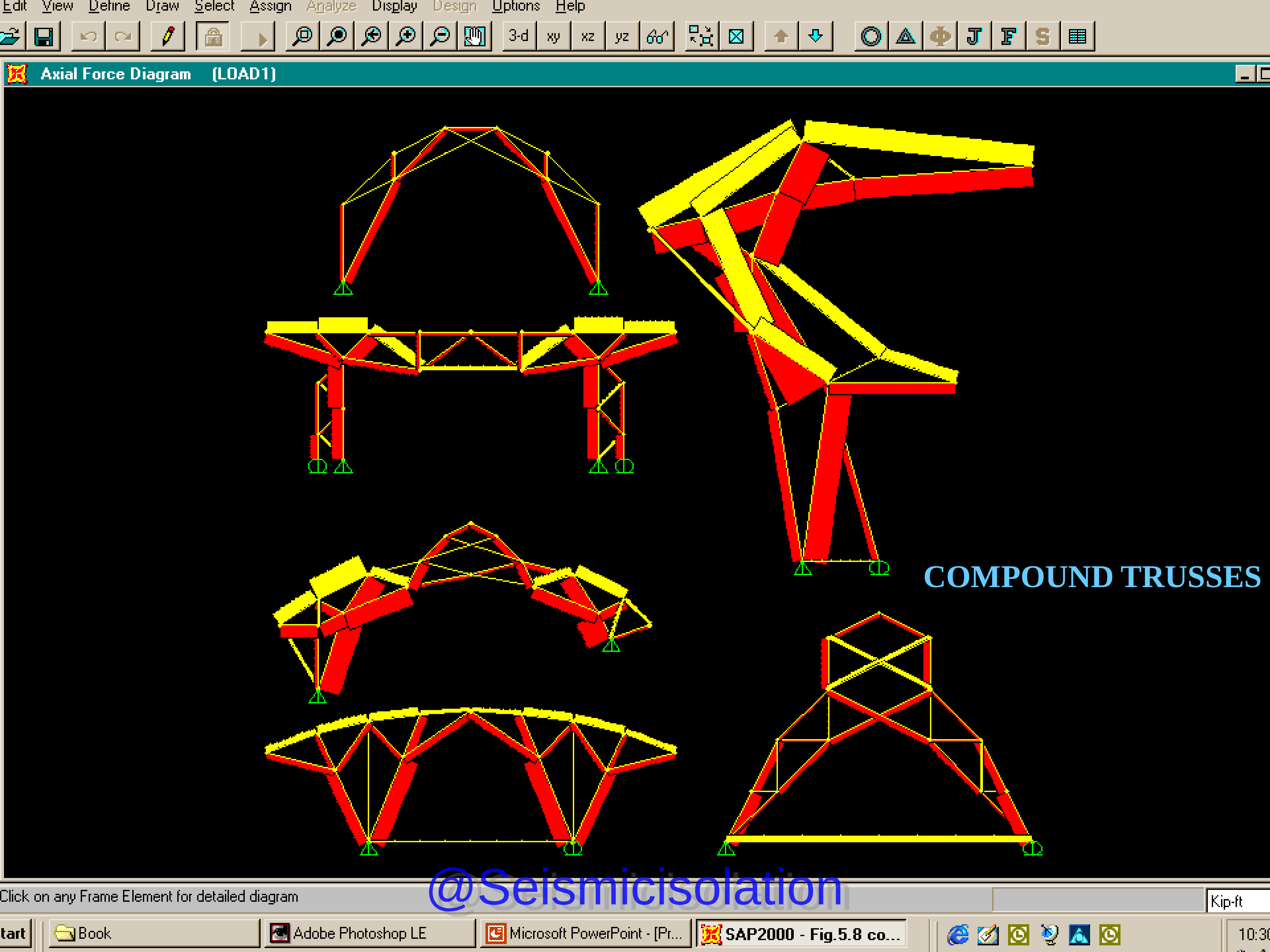


RECTANGULAR TRUSSES

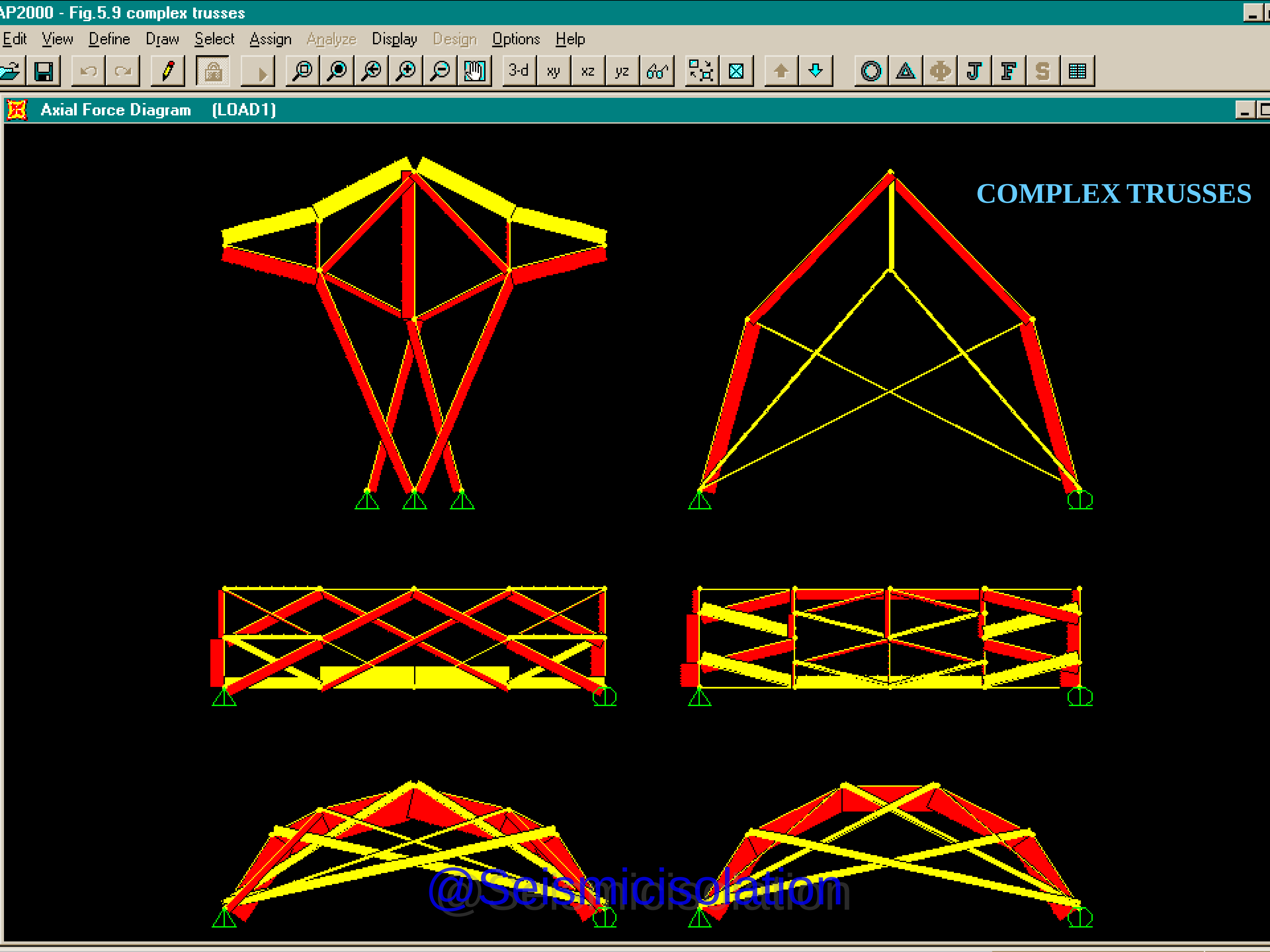

SAP2000 - Fig. 5.4 triangular trusses
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Joint Loads (LOAD1)
Axial Force Diagram (LOAD1)
TRIANGULAR TRUSSES

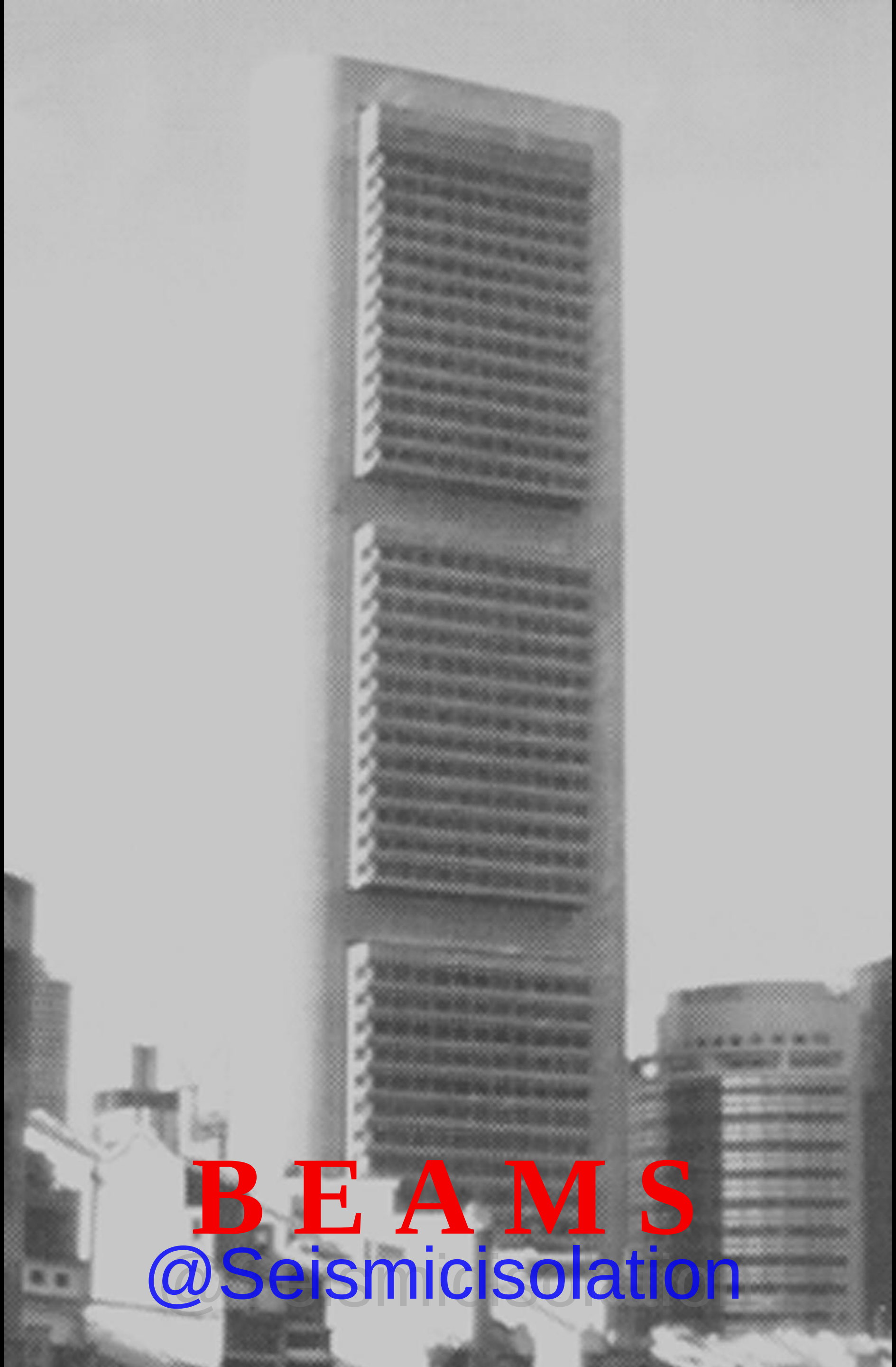

AP2000 - Fig 5.6 funicular trusses
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (LOAD1)
Joint Loads (LOAD1)
FUNICULAR TRUSSES
Z


SAP2000 - Fig.5.7 fan trusses 6
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (LOAD1)
FAN TRUSSES

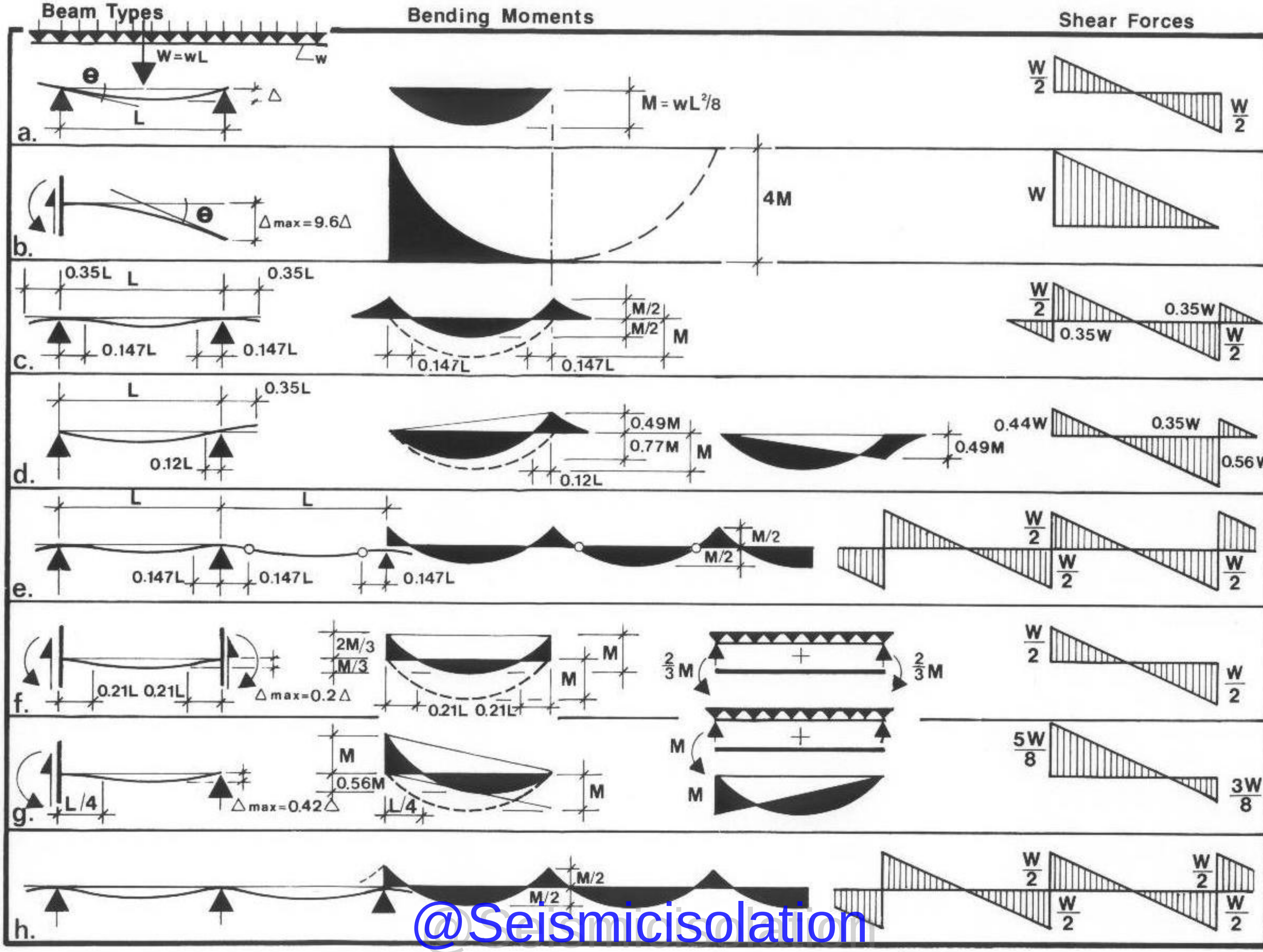


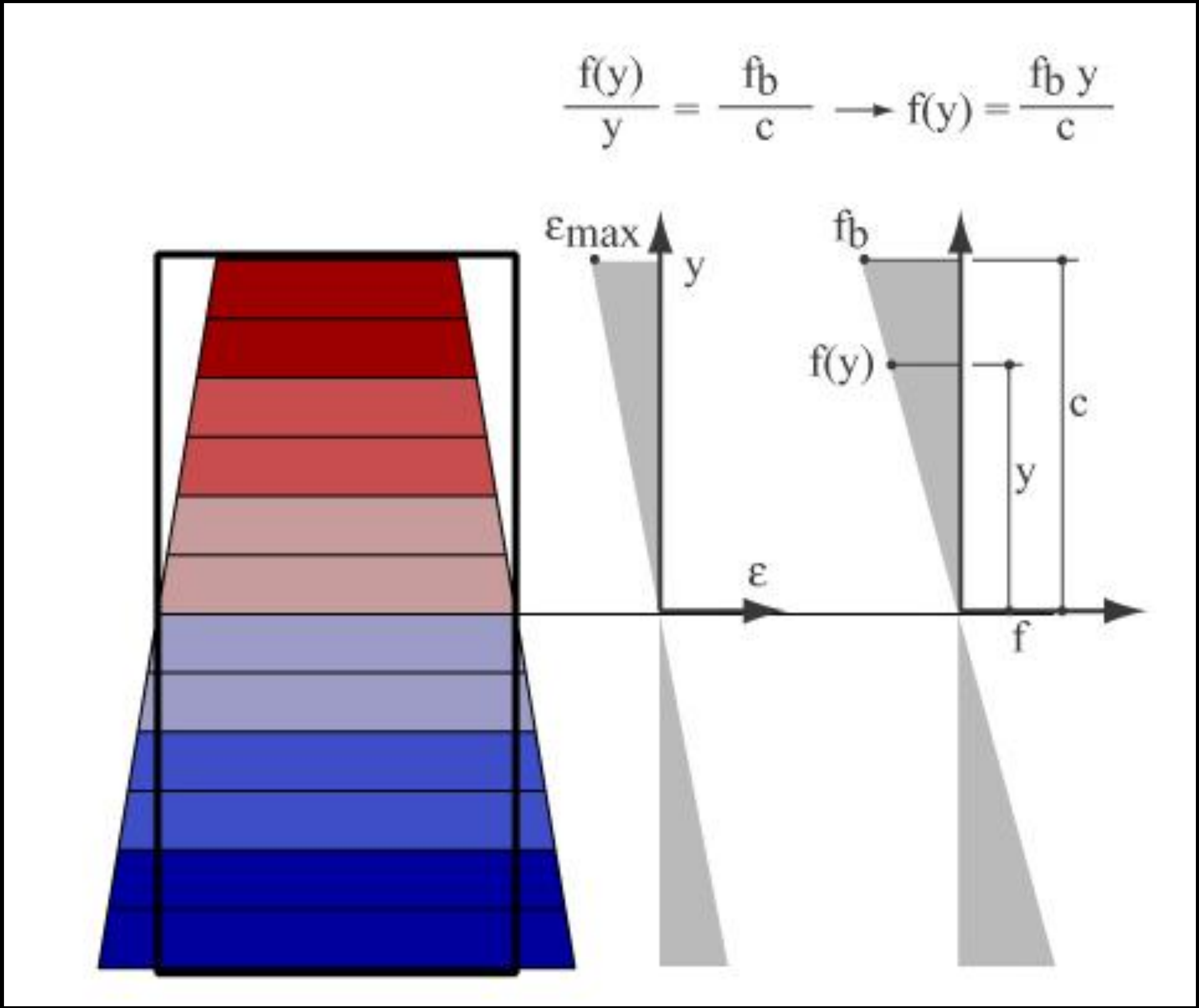
COMPOUND TRUSSES


AP2000 - Fig.5.9 complex trusses
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (LOAD1)
COMPLEX TRUSSES



OBCB Center, Singapore, 1976, I.M. Pei/ Ove Arup


Beam Types
Bending Moments
Shear Forces
W=wL
w
θ
Δ
L
a.
M=wL²/8
W
2
W
2
Δ max=9.6Δ
θ
b.
4M
W
0.35L
L
0.35L
0.147L
0.147L
c.
M/2
M/2
M
0.147L
0.147L
0.35W
0.35W
L
0.35L
0.12L
d.
0.49M
0.77M
M
0.12L
0.49M
0.44W
0.35W
0.56W
L
L
0.147L
0.147L
0.147L
e.
M/2
M/2
2M/3
M/3
M
M
0.21L 0.21L
Δ max=0.2Δ
f.
0.21L 0.21L
2/3 M
2/3 M
M
0.56M
L/4
Δ max=0.42Δ
g.
L/4
M
M
M
5W
8
3W
8
h.

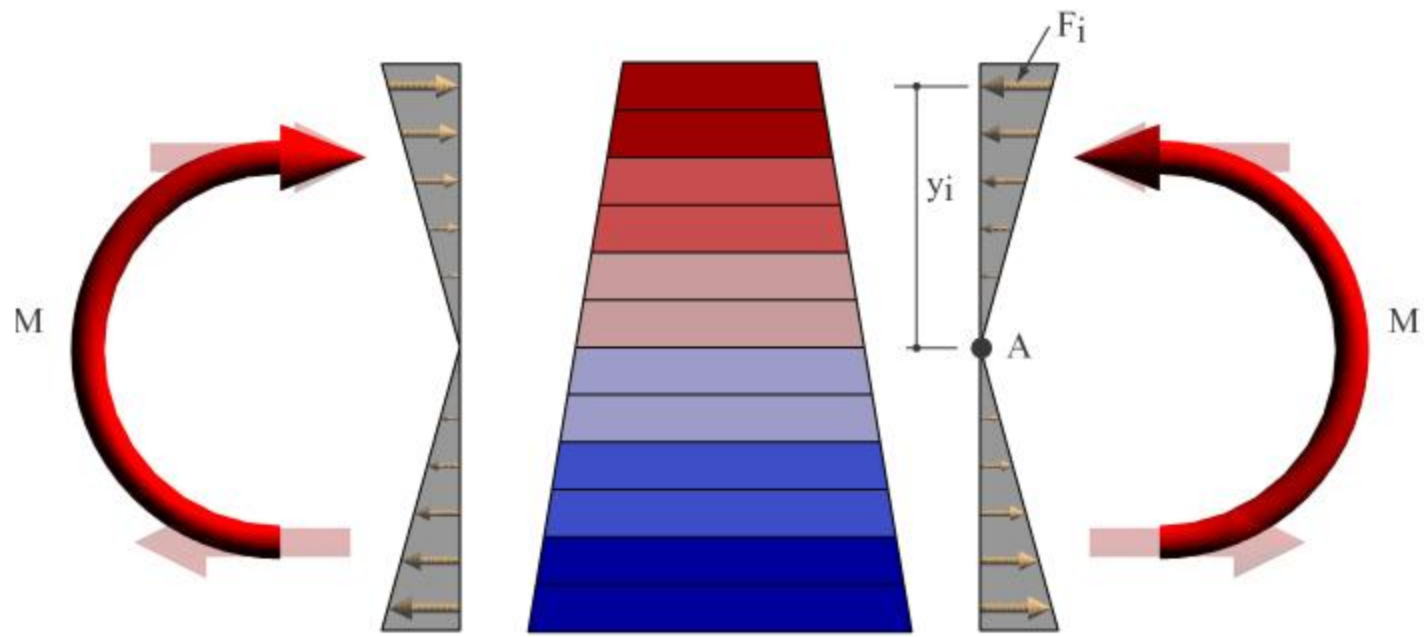


$\frac{f(y)}{y} = \frac{f_b}{c} \rightarrow f(y) = \frac{f_b\, y}{c}$
ε_{max}
y
ε
f_b
f(y)
c
y
f

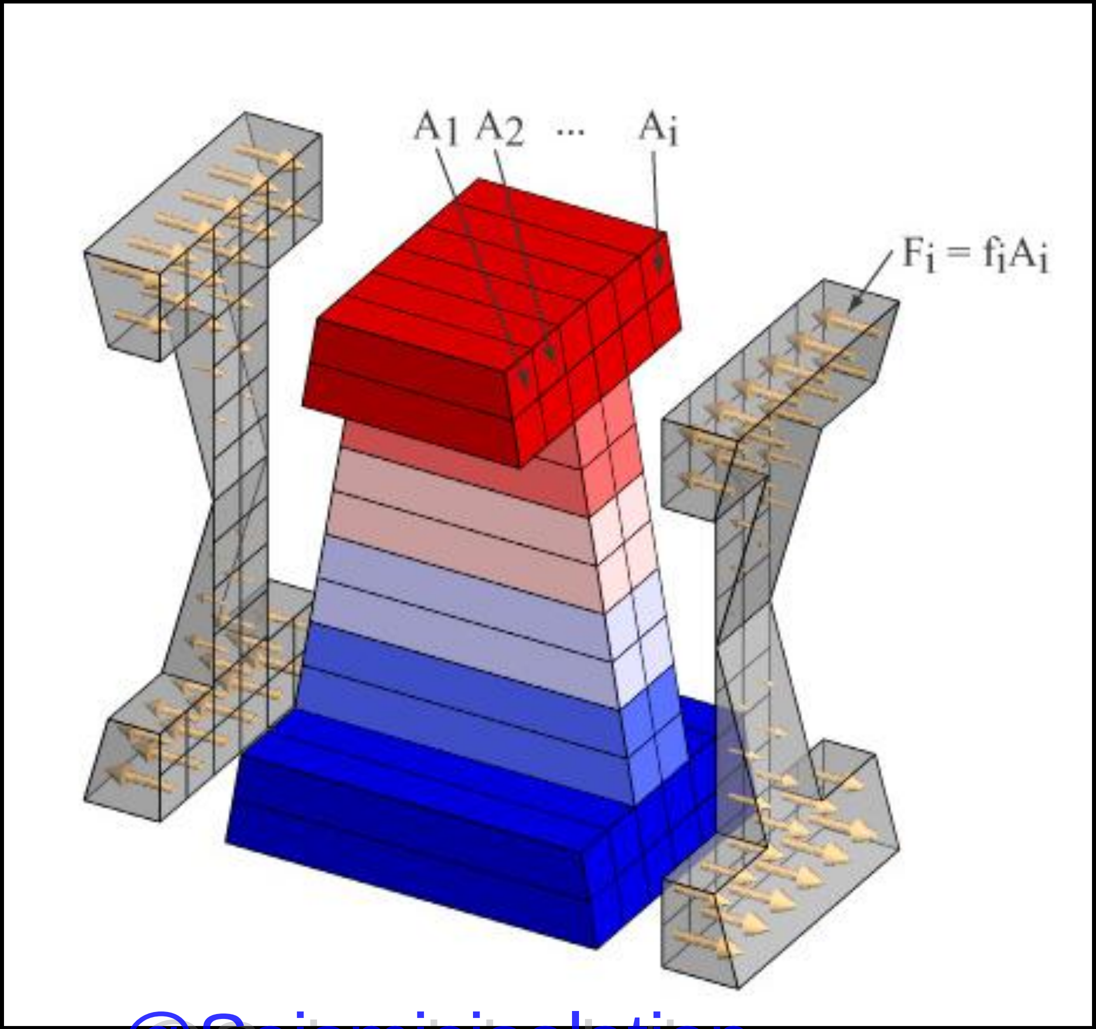
$$M = \sum_i F_i y_i \qquad M = \frac{f_b}{c}\sum_i A_i y_i^2 \qquad I = \sum_i A_i y_i^2$$

$$M = \sum_i (f_i A_i) y_i \qquad M = \frac{f_b}{c} I \qquad S = \frac{I}{c}$$

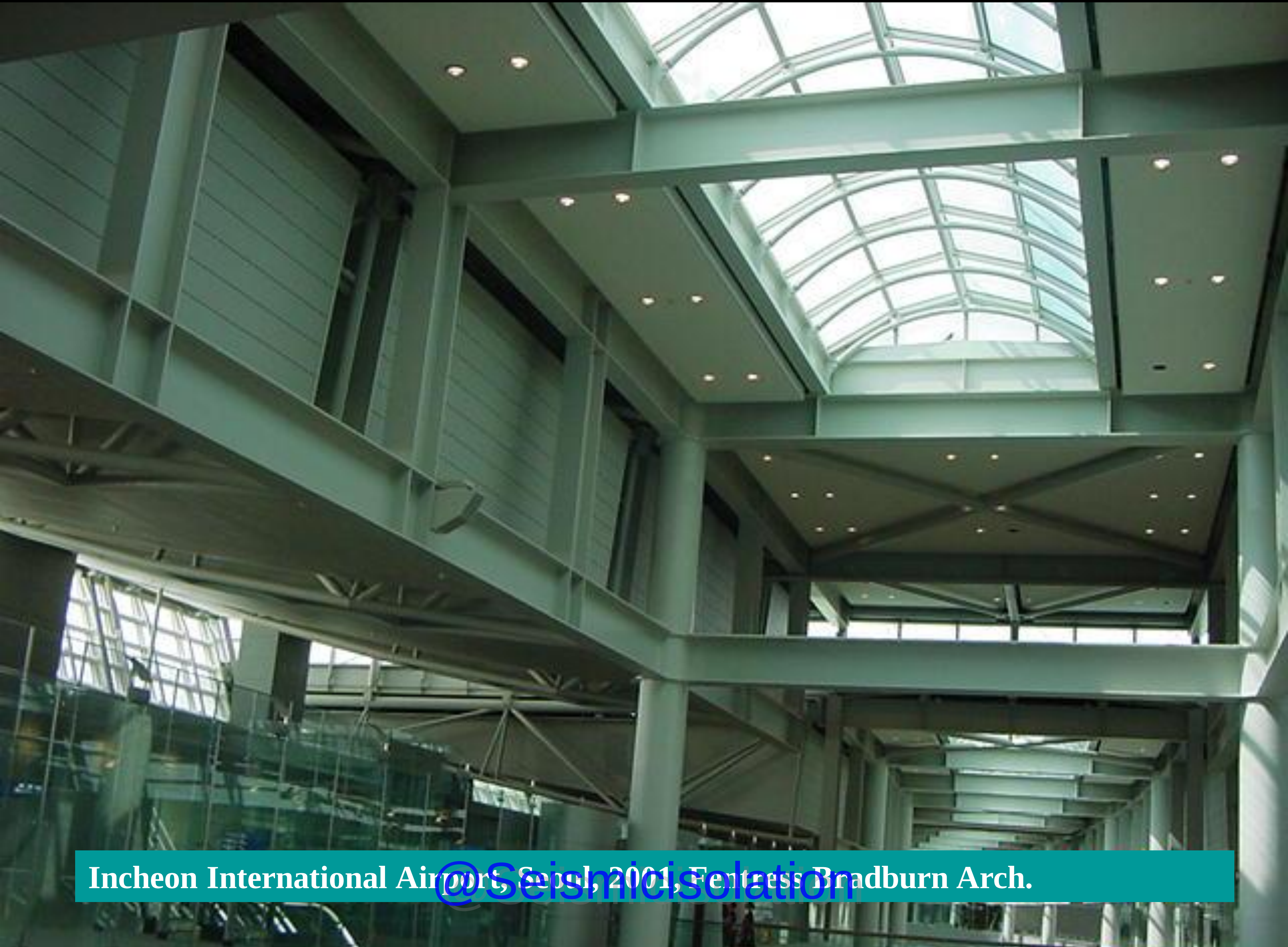
$$M = \sum_i \left(\frac{f_b y_i}{c} A_i\right) y_i \qquad f_b = \frac{Mc}{I} \qquad f_b = \frac{M}{S}$$



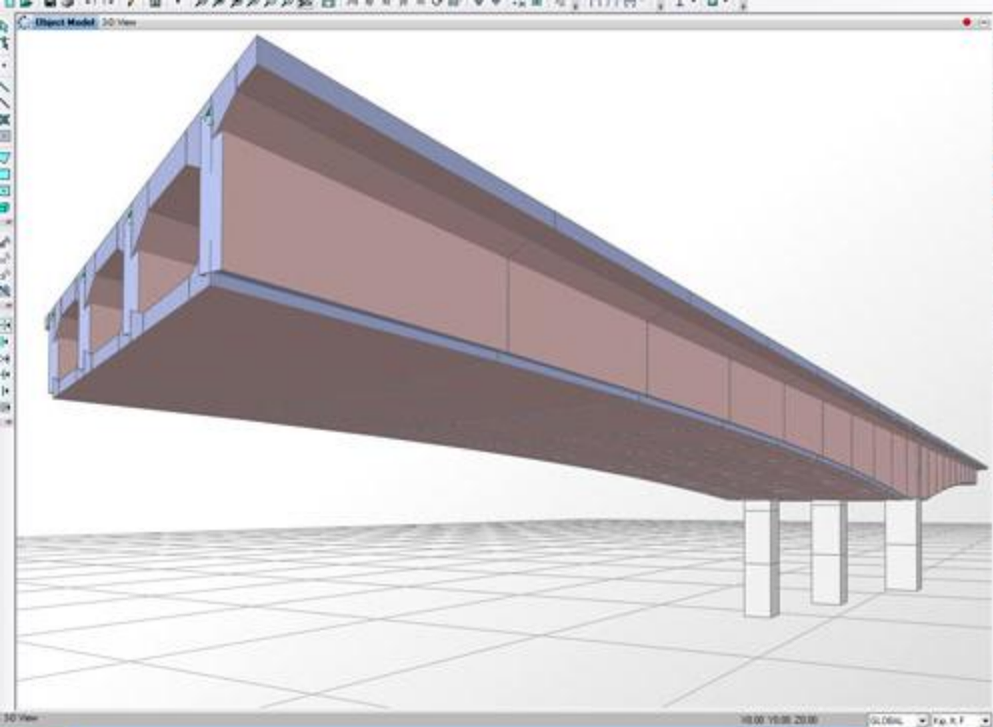
General Form of the Flexure Formula

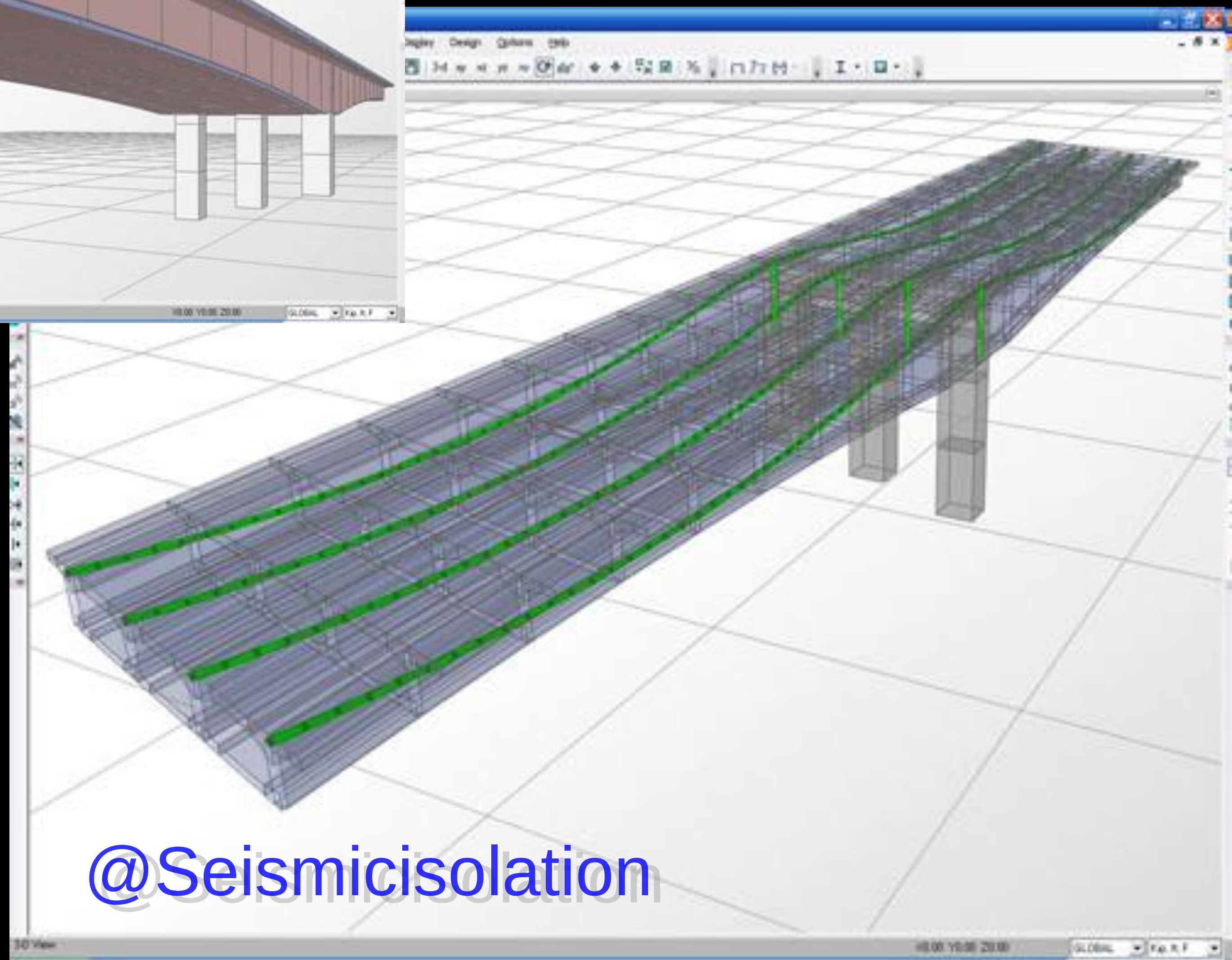


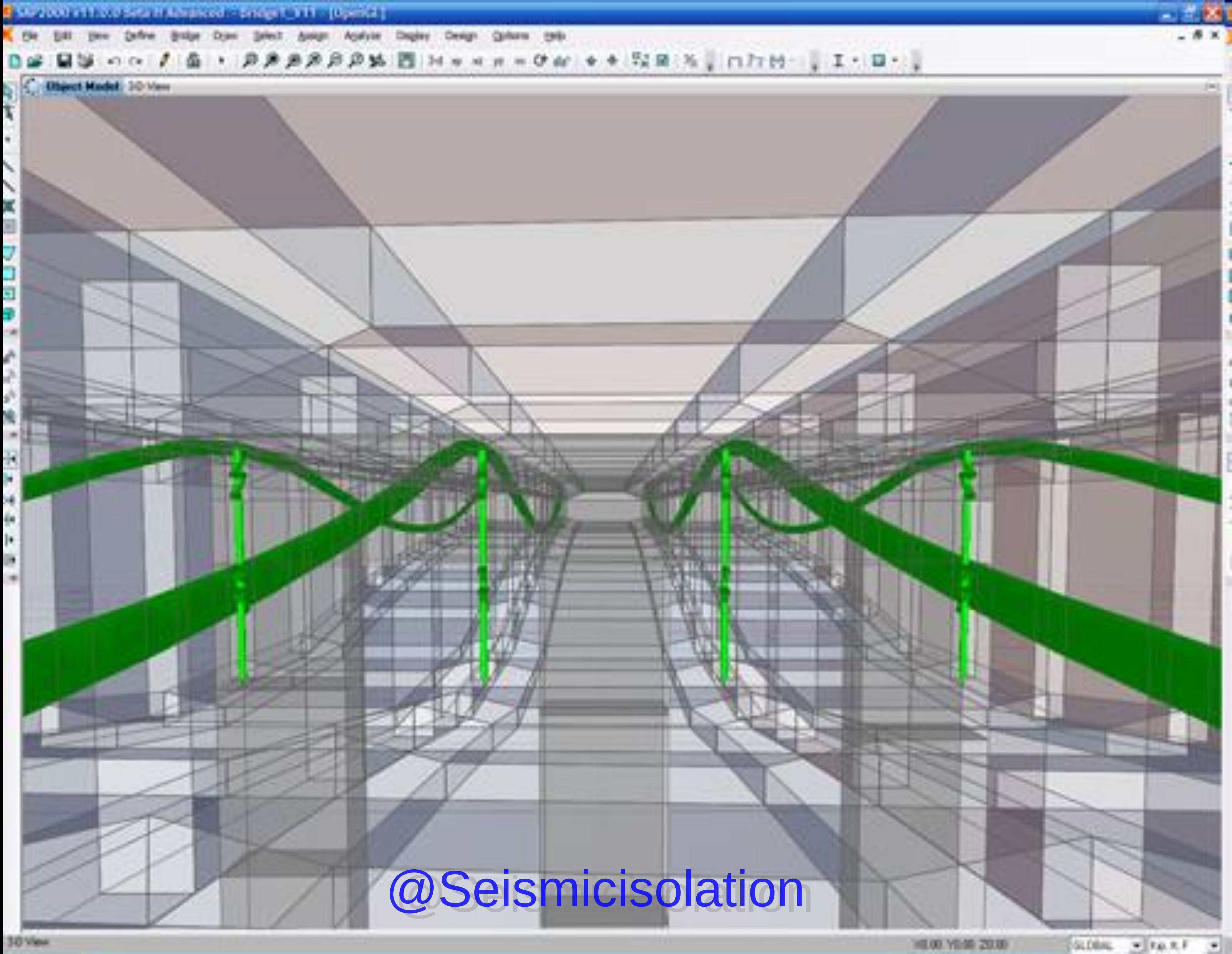
@Seismicisolation

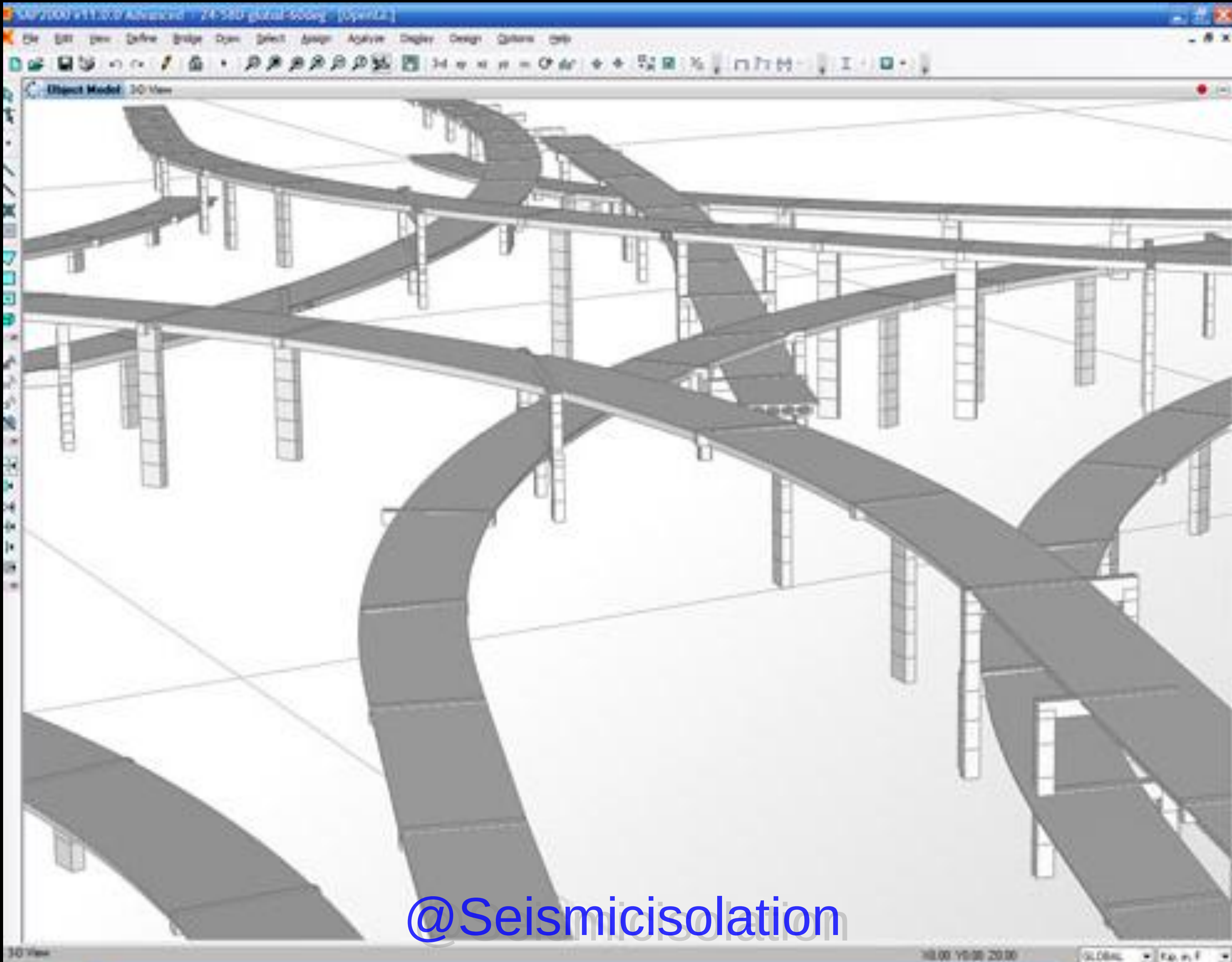


Incheon International Airport, Seoul, 2001, Fentress Bradburn Arch.



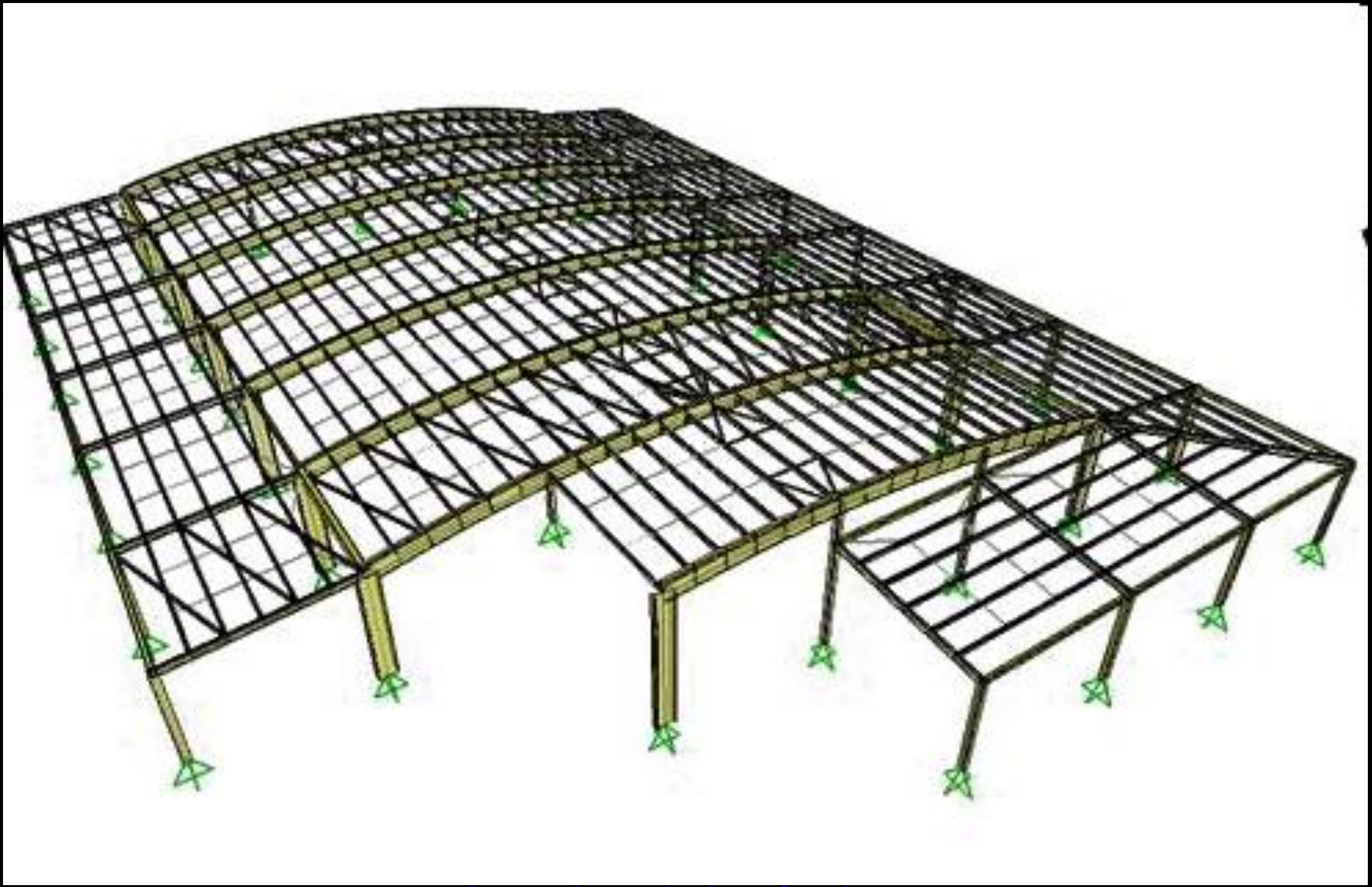






Ningbo downtown, **2002, Qingyun Ma**

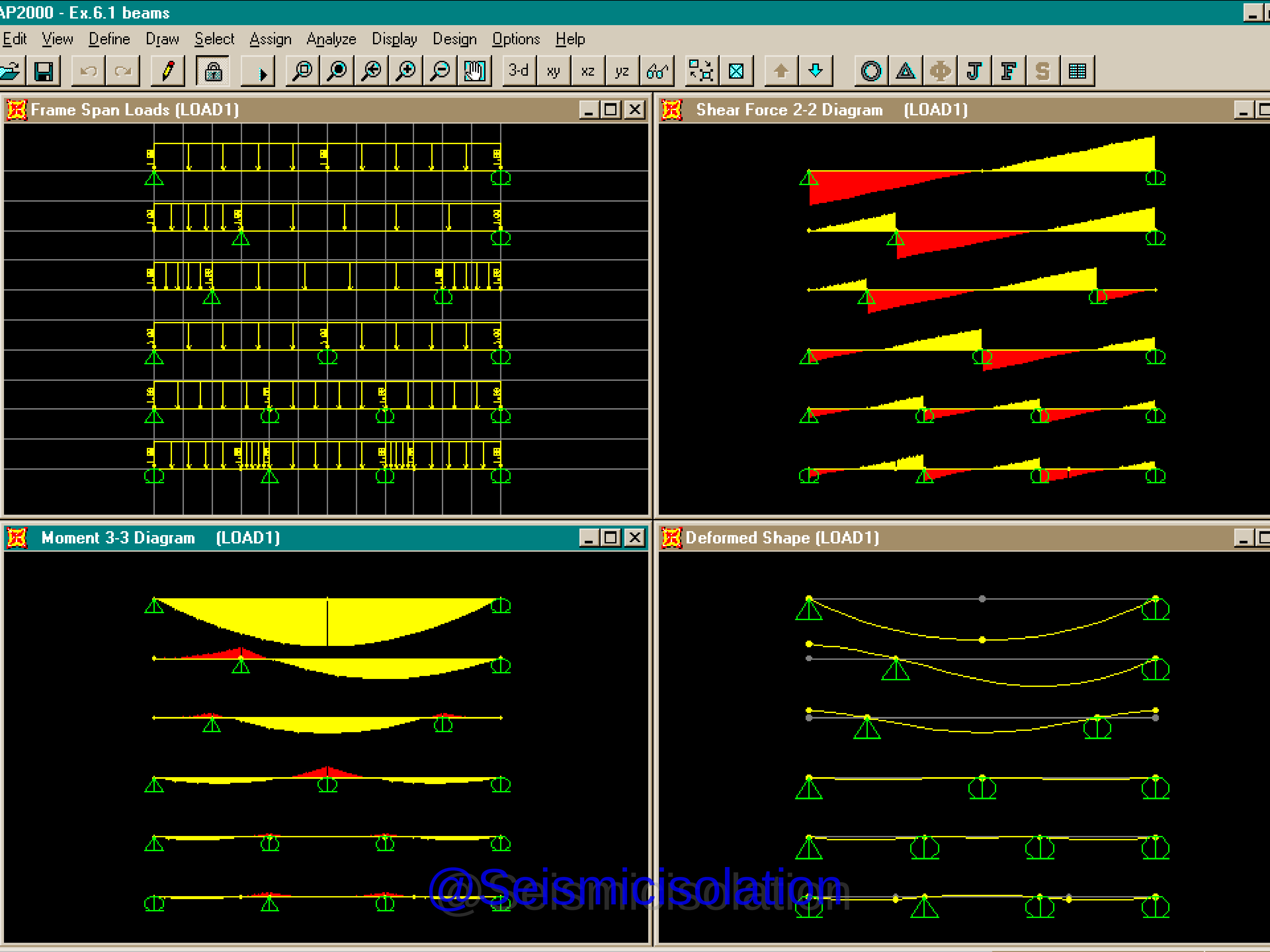
Dresdner Bank, **Verwaltungszentrum, Leipzig, 1997, Engel und Zimmermann**



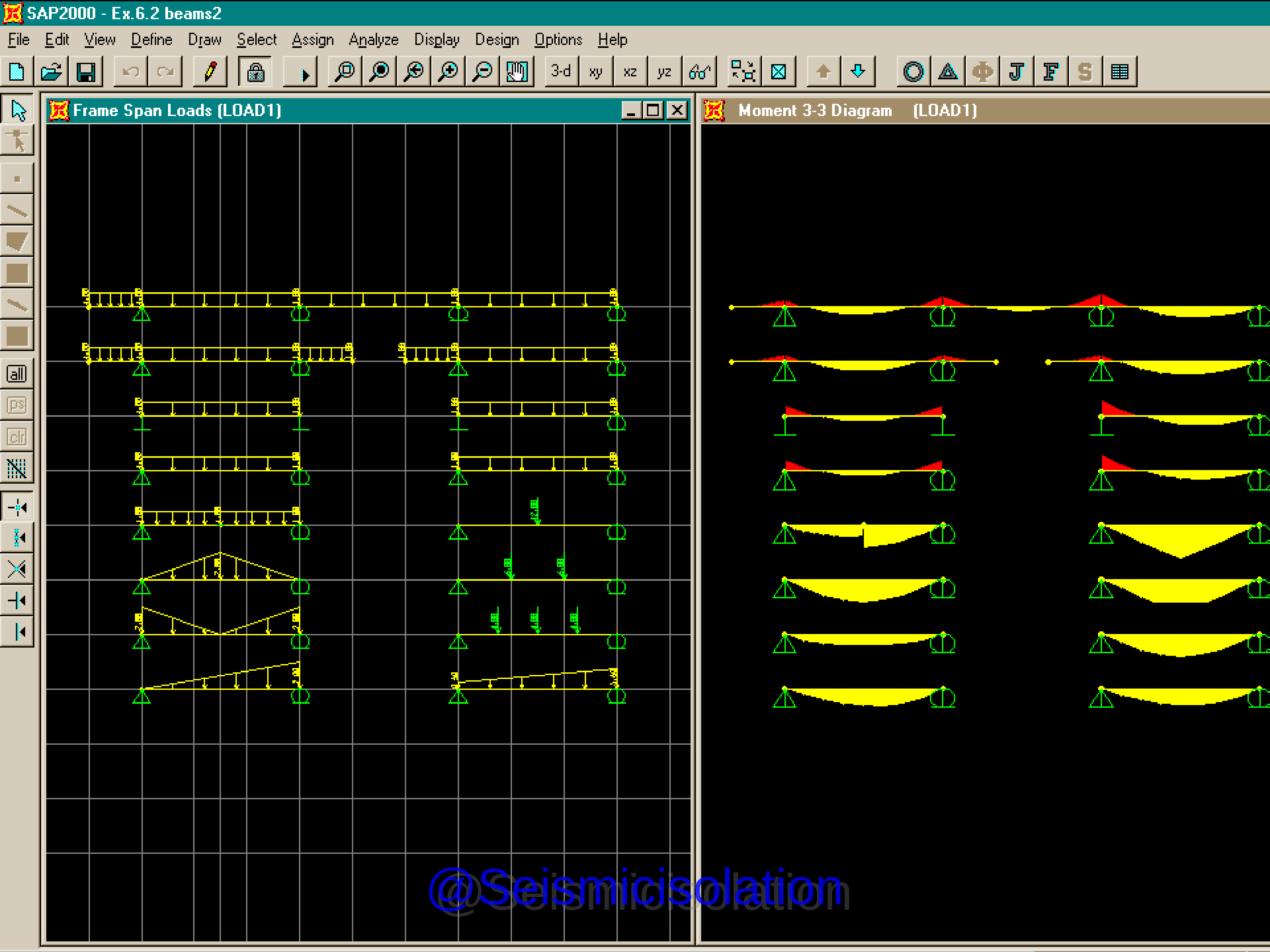
National Gallery of Art, East Wing, Washington, 1978, I.M. Pei

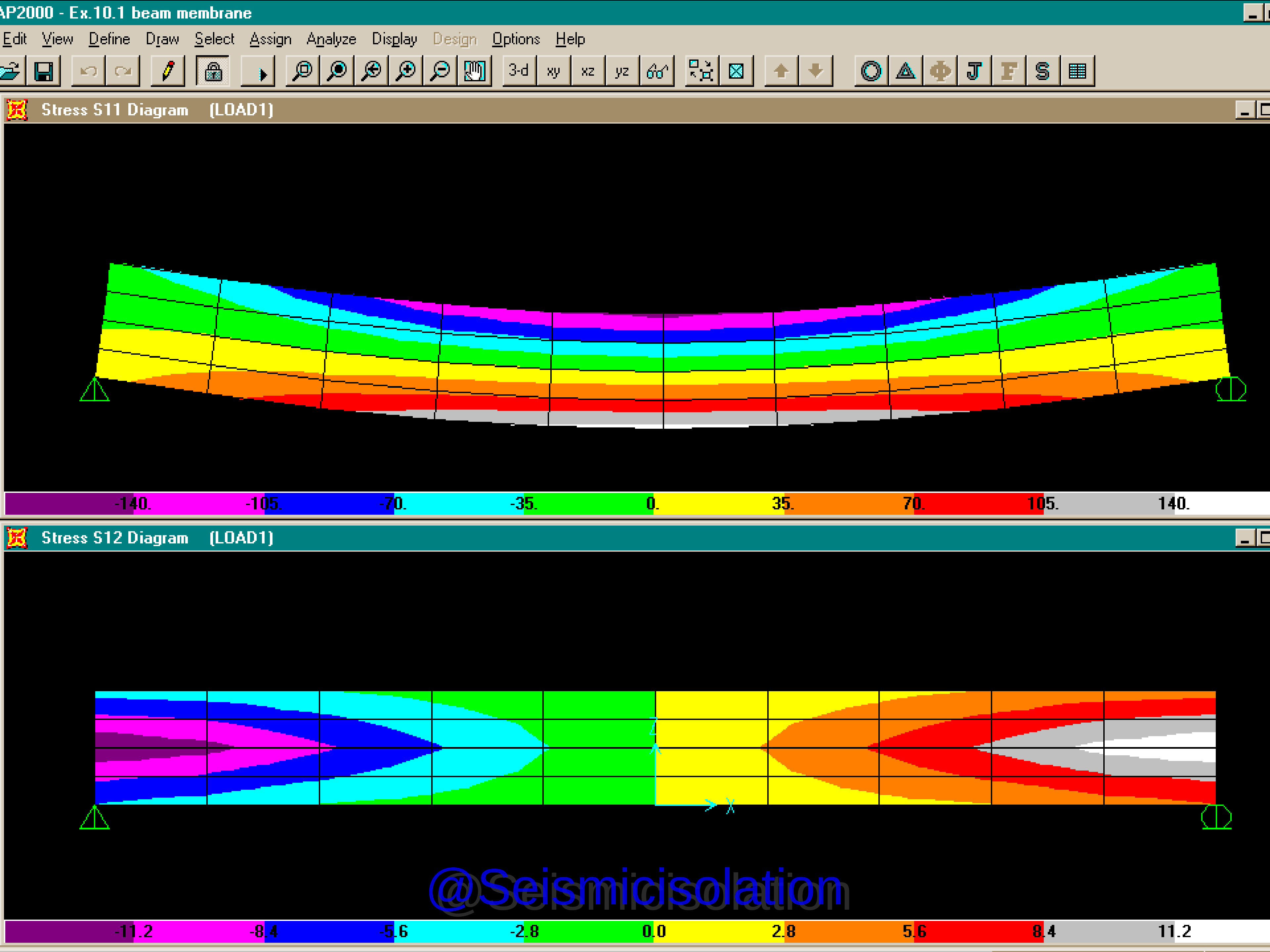
Atrium, Germanisches Museum, Nuremberg, Germany

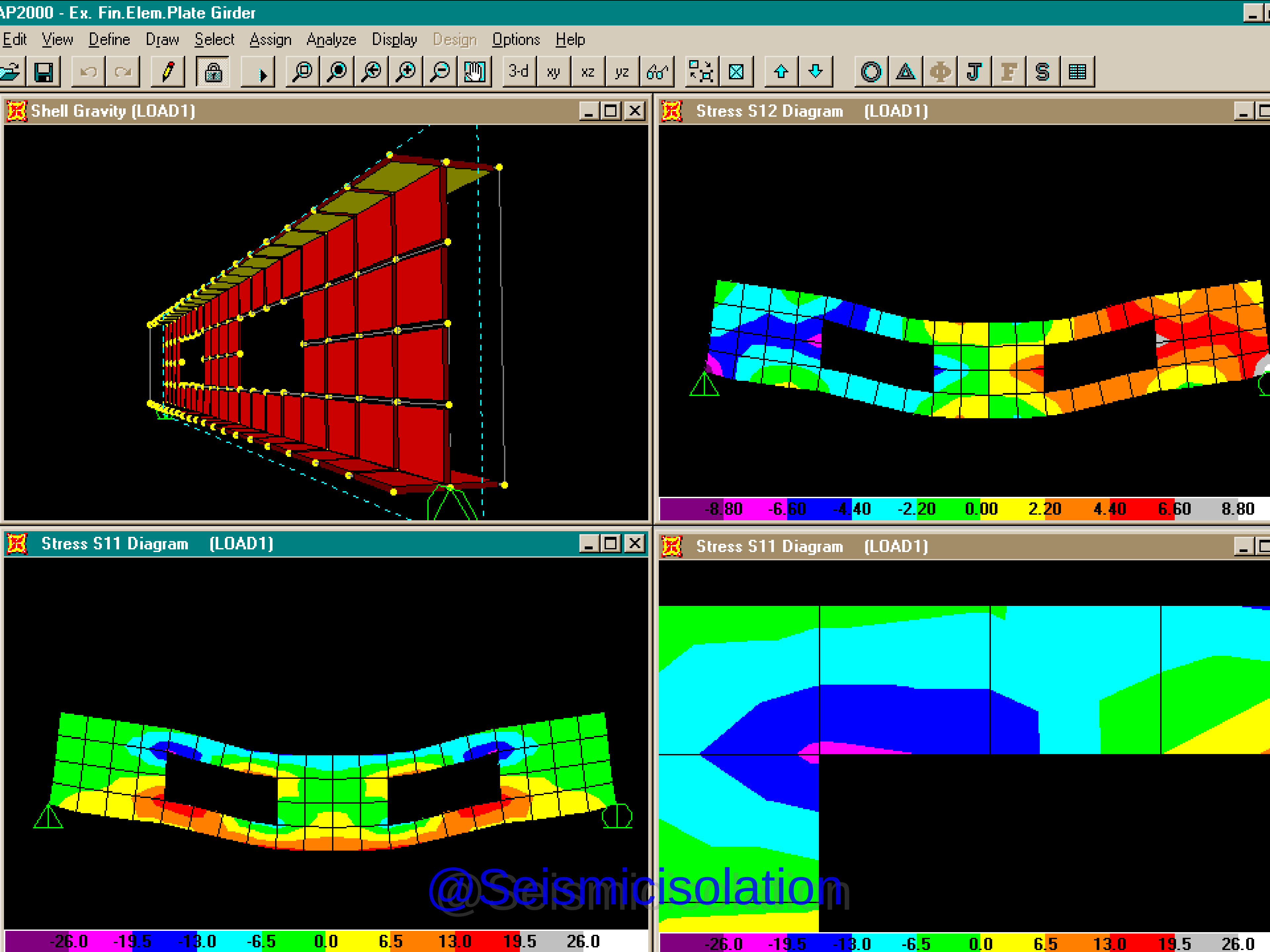
Shanghai-Pudong International Airport, 2001, Paul Andreu principal architect, Coyne et Bellier structural engineers

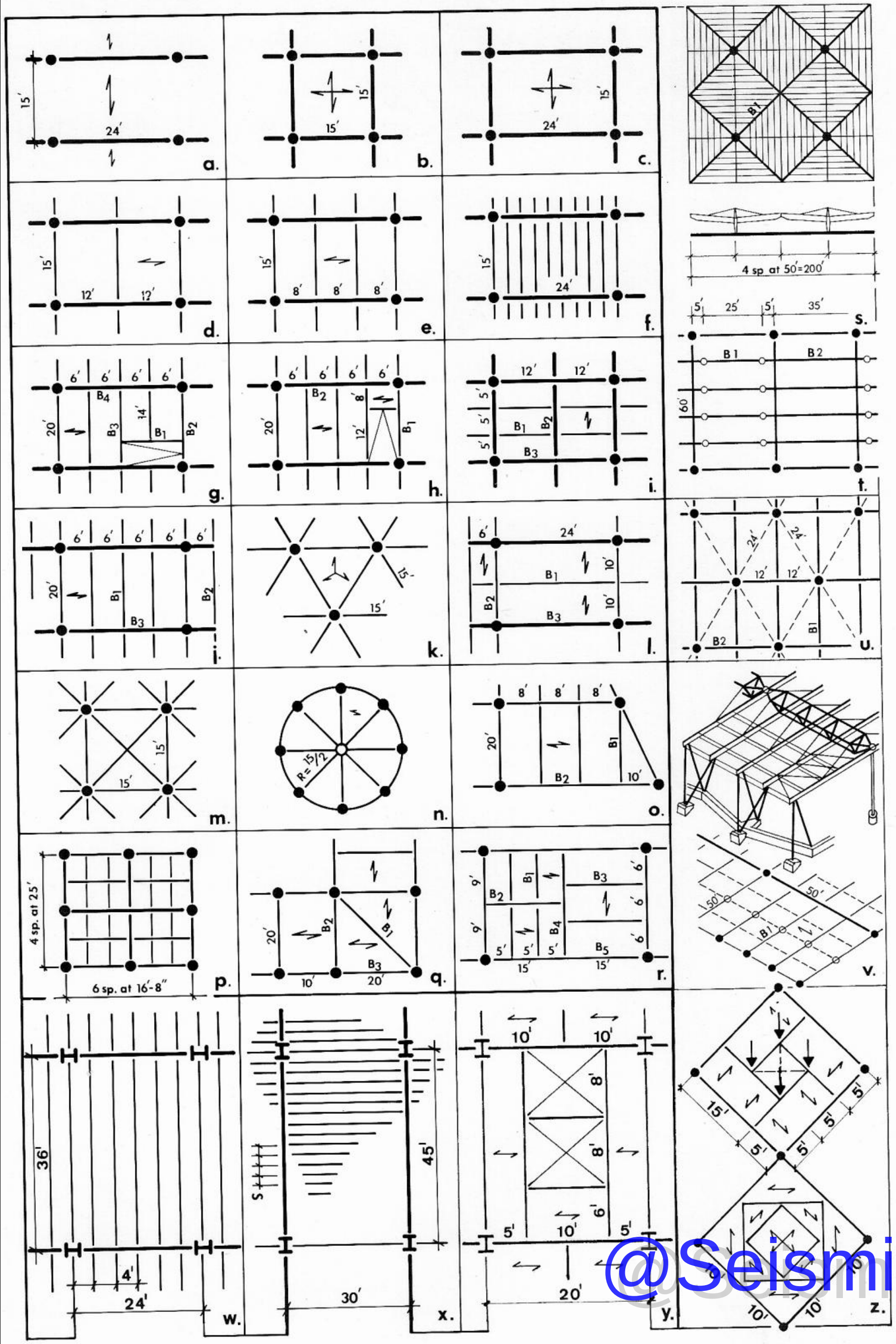


AP2000 - Ex.6.1 beams
Edit View Define Draw Select Assign Analyze Display Design Options Help
Frame Span Loads (LOAD1)
Shear Force 2-2 Diagram (LOAD1)
Moment 3-3 Diagram (LOAD1)
Deformed Shape (LOAD1)
@Seismicisolation

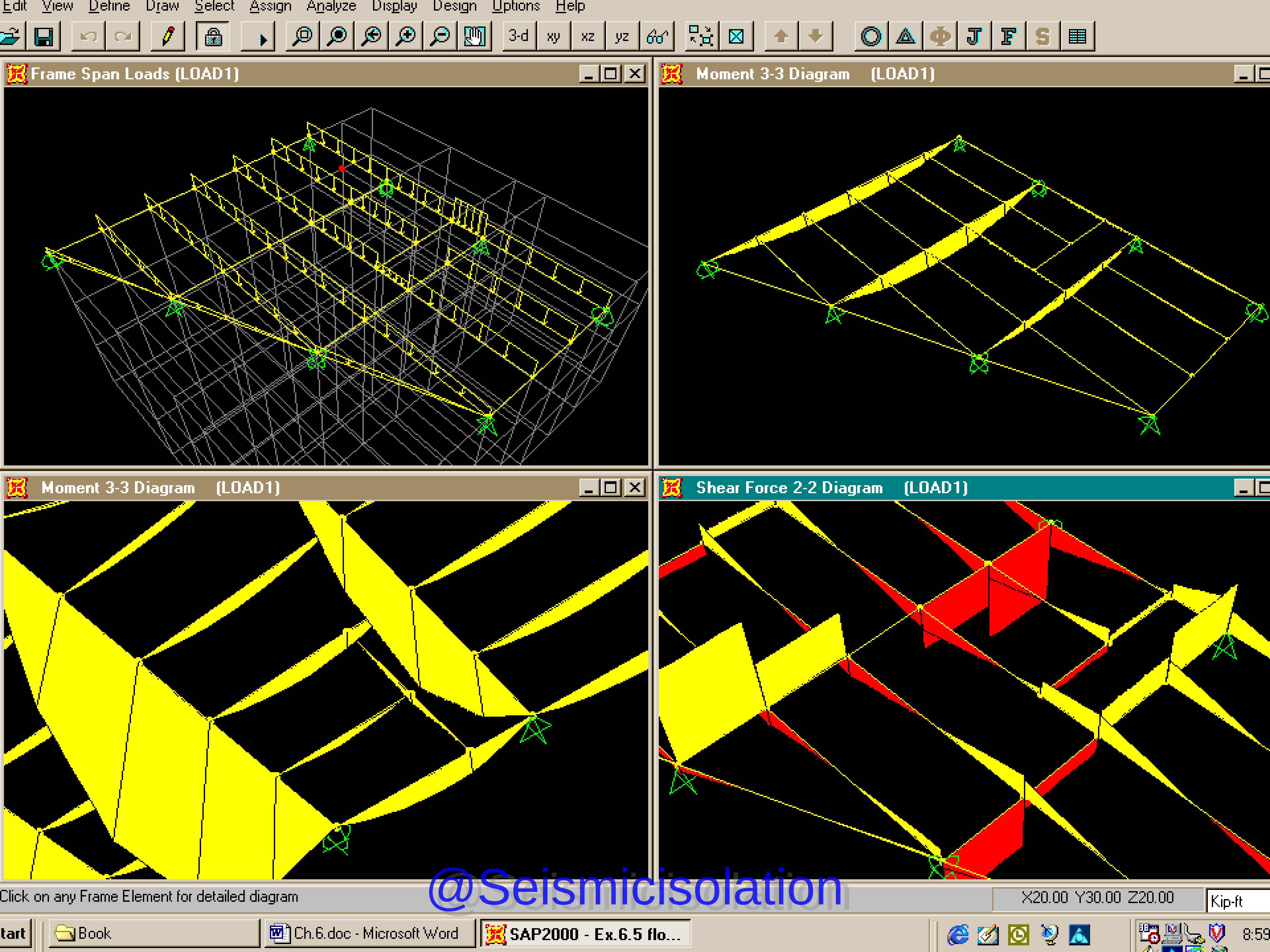

SAP2000 - Ex.6.2 beams2
File Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
all
ps
clr
Frame Span Loads (LOAD1)
Moment 3-3 Diagram (LOAD1)

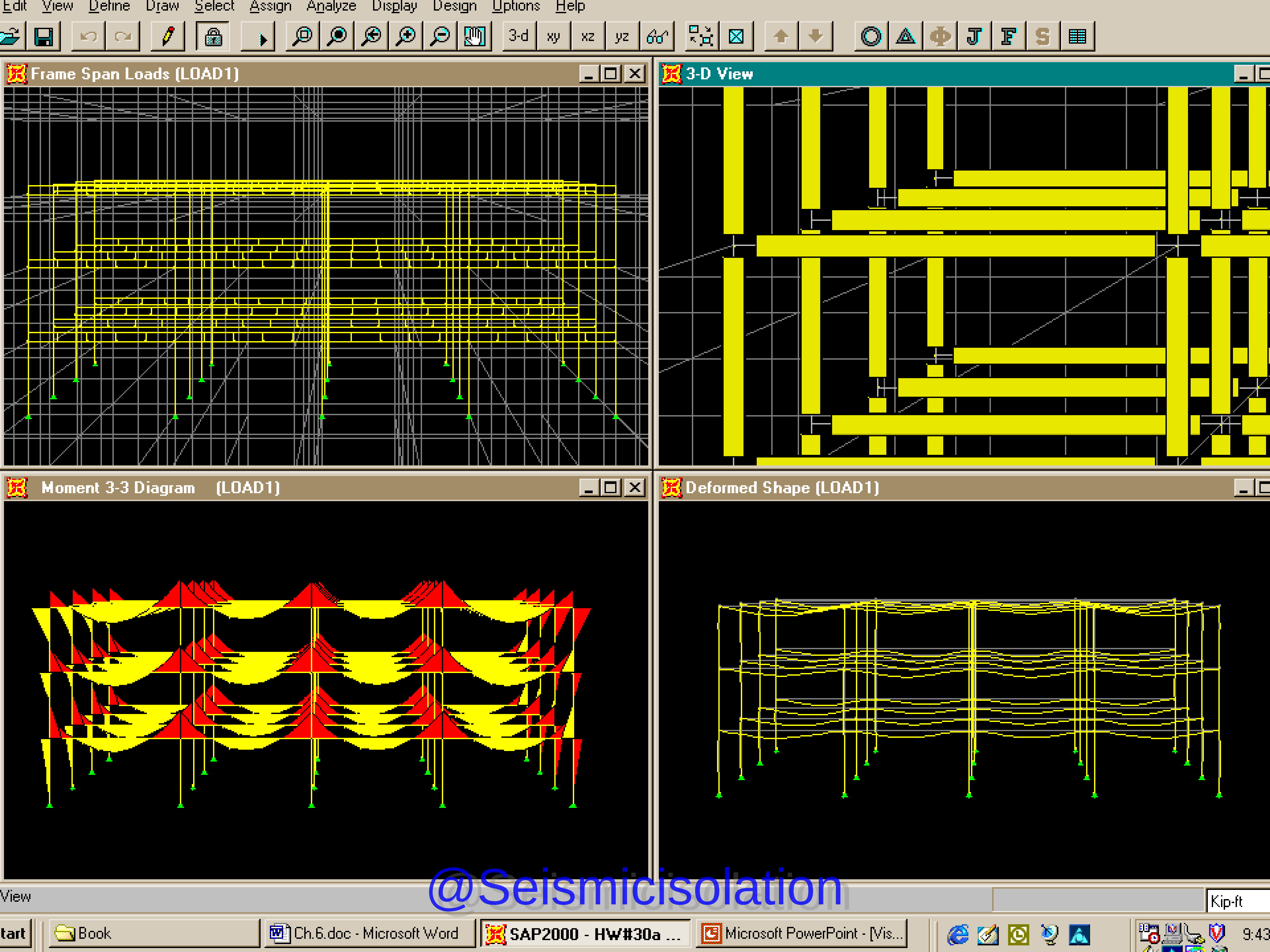

AP2000 - Ex.10.1 beam membrane
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Stress S11 Diagram (LOAD1)
-140. -105. -70. -35. 0. 35. 70. 105. 140.
Stress S12 Diagram (LOAD1)
X
@Seismicisolation
-11.2 -8.4 -5.6 -2.8 0.0 2.8 5.6 8.4 11.2

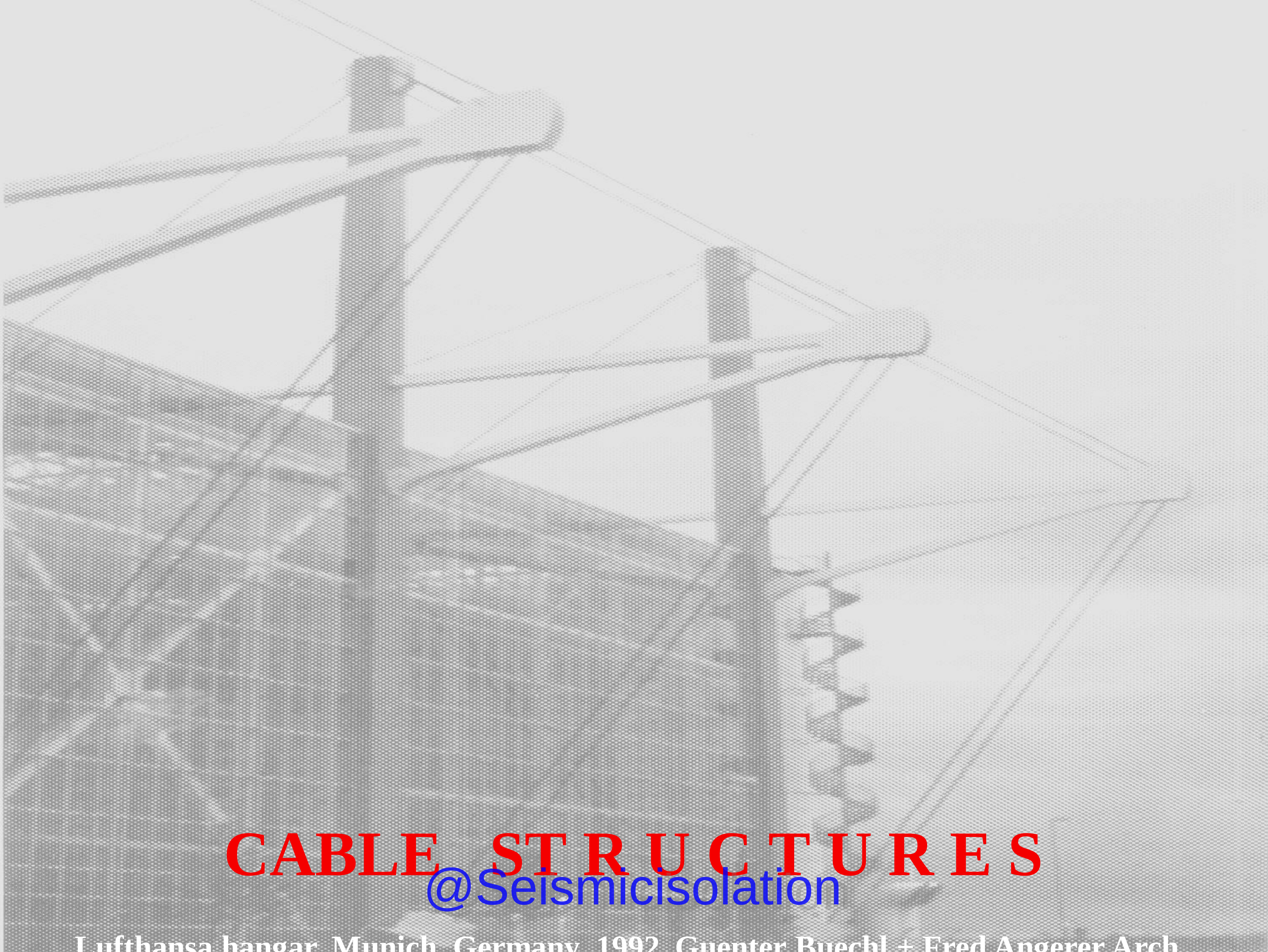

AP2000 - Ex. Fin.Elem.Plate Girder
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Shell Gravity (LOAD1)
Stress S12 Diagram (LOAD1)
-8.80 -6.60 -4.40 -2.20 0.00 2.20 4.40 6.60 8.80
Stress S11 Diagram (LOAD1)
-26.0 -19.5 -13.0 -6.5 0.0 6.5 13.0 19.5 26.0
Stress S11 Diagram (LOAD1)
-26.0 -19.5 -13.0 -6.5 0.0 6.5 13.0 19.5 26.0



FLOOR-ROOF FRAMING SYSTEMS


Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
Moment 3-3 Diagram (LOAD1)
Moment 3-3 Diagram (LOAD1)
Shear Force 2-2 Diagram (LOAD1)
Click on any Frame Element for detailed diagram
X20.00 Y30.00 Z20.00
Kip-ft
tart
Book
Ch.6.doc - Microsoft Word
SAP2000 - Ex.6.5 flo...
8:59


Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
3-D View
Moment 3-3 Diagram (LOAD1)
Deformed Shape (LOAD1)
View
Kip-ft
tart
Book
Ch.6.doc - Microsoft Word
SAP2000 - HW#30a ...
Microsoft PowerPoint - [Vis...
9:43



C A B L E S T R U C T U R E S

Lufthansa hangar, Munich, Germany, 1992, Guenter Buechl + Fred Angerer Arch.

CABLE STRUCTURES

Single-layer, simply suspended cable roofs

Single-curvature and dish-shaped (synclastic) hanging roofs

Prestressed tensile membranes and cable nets

Edge-supported saddle roofs
Mast-supported conical saddle roofs
Arch-supported saddle roofs
Air supported structures and air-inflated structures (air members)

Cable-supported structures

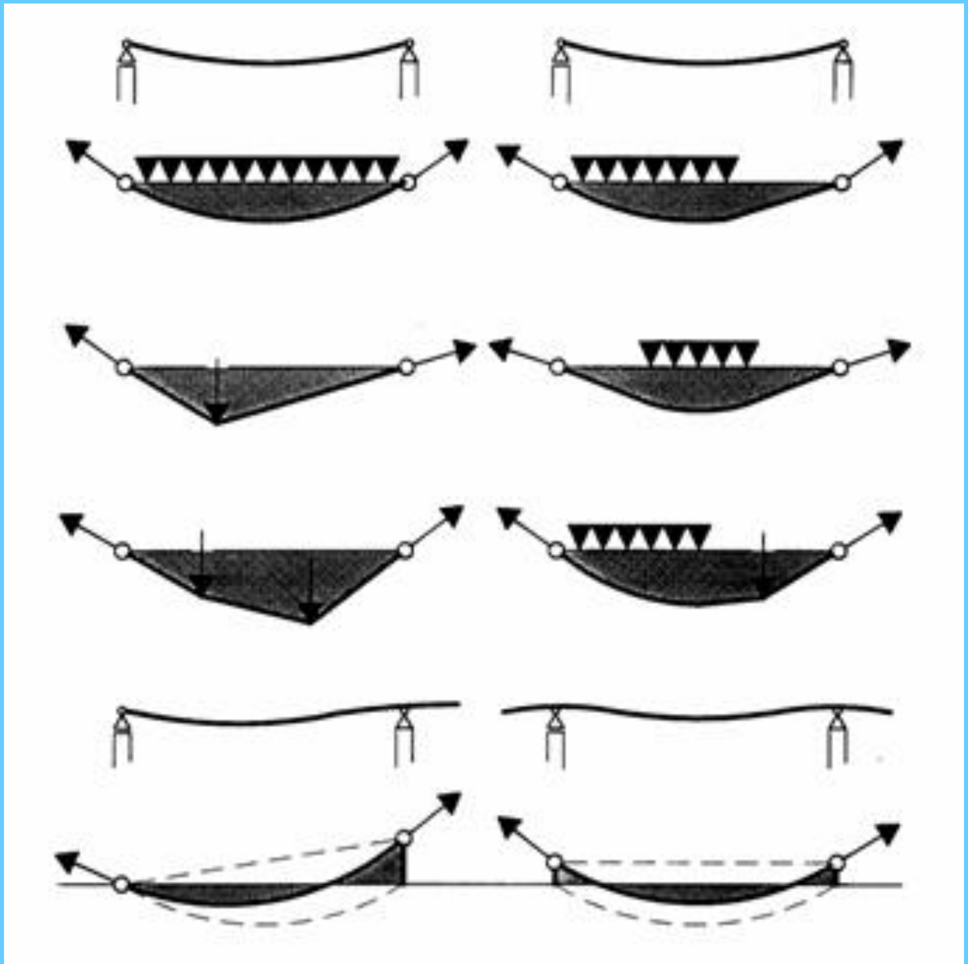
cable-supported beams and arched beams
cable-stayed bridges
cable-stayed roof structures

Tensegrity structures

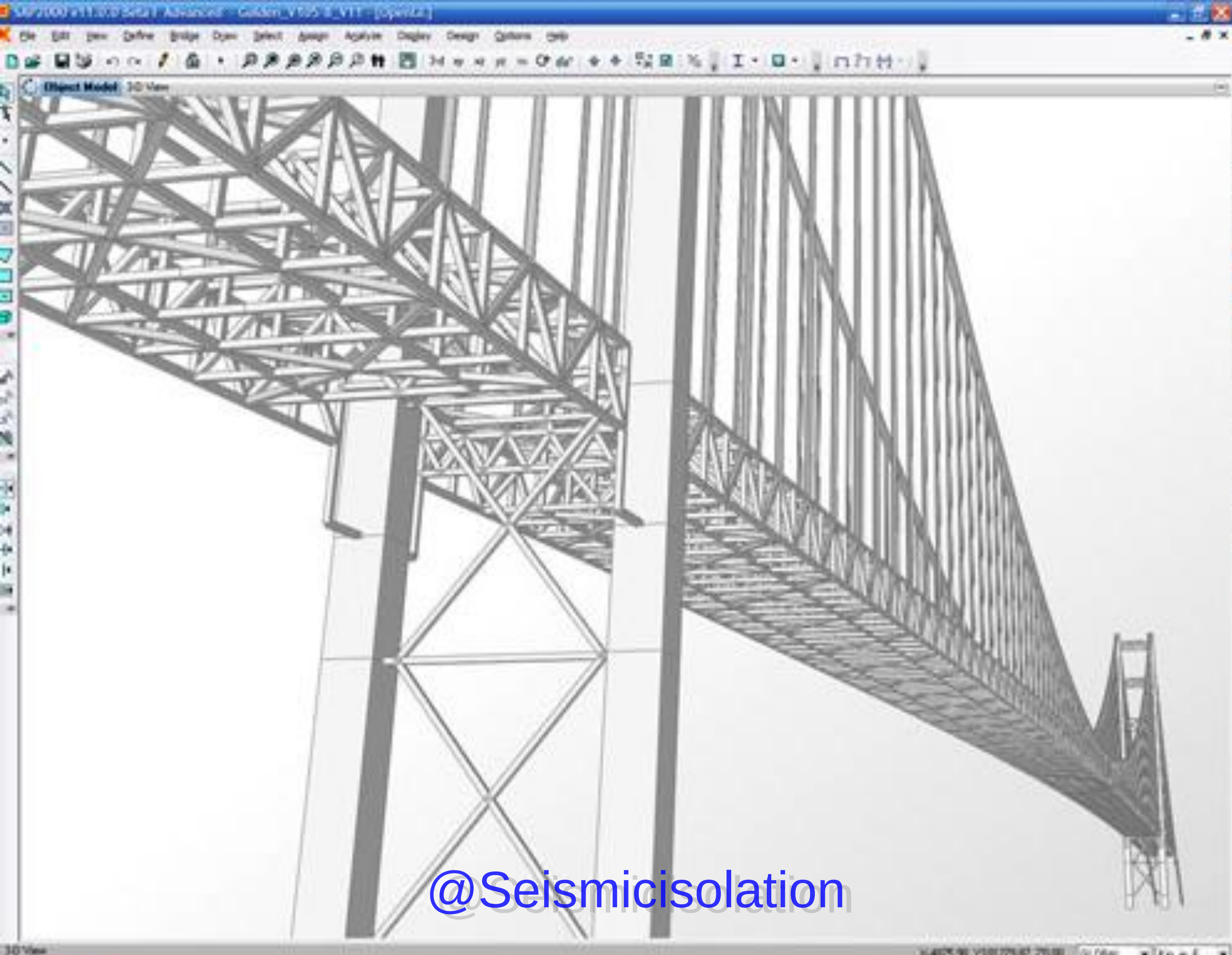
Planar open and closed tensegrity systems: cable beams, cable trusses, cable frames
Spatial open tensegrity systems: cable domes
Spatial closed tensegrity systems: polyhedral twist units

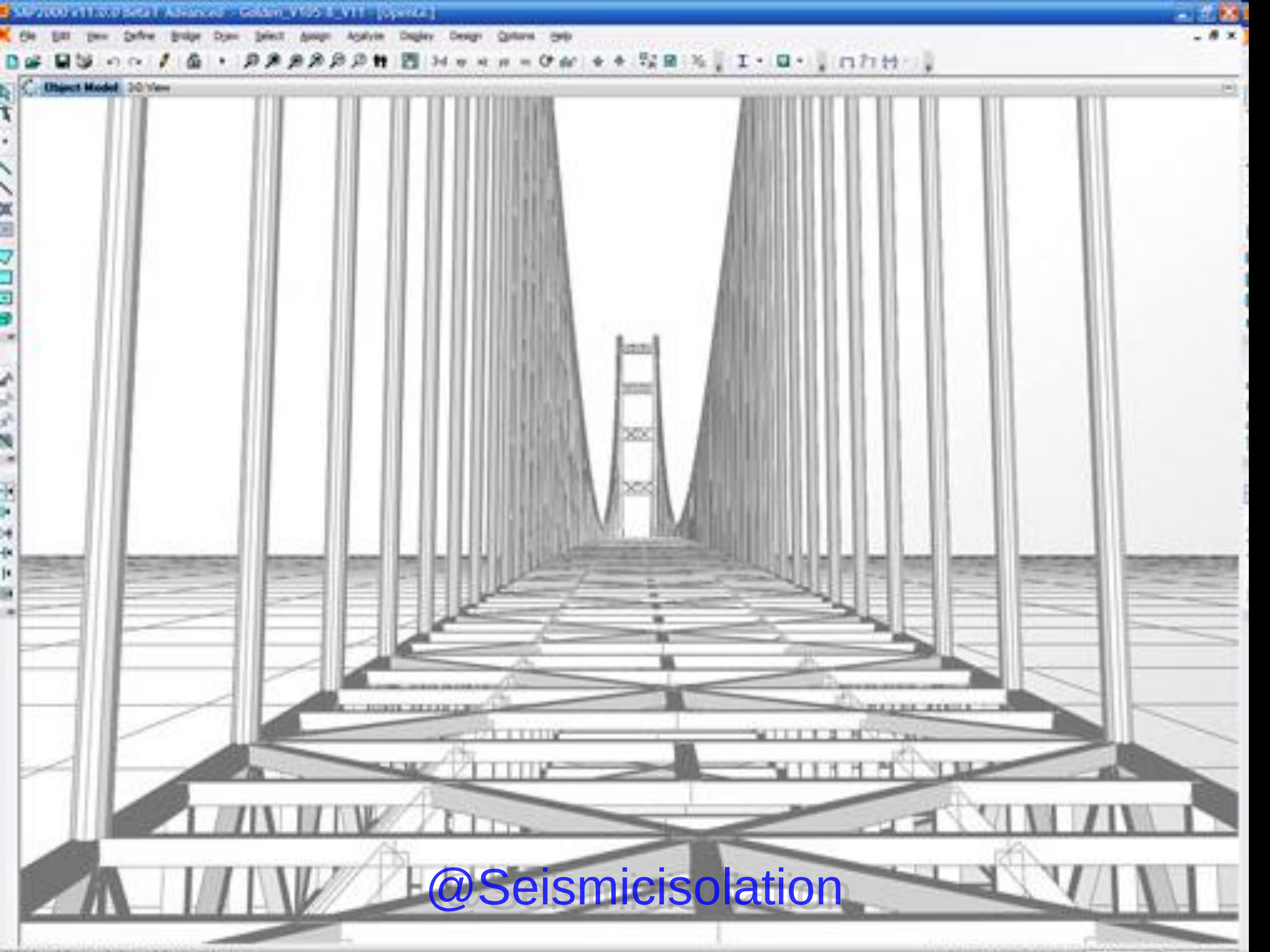
Hybrid structures

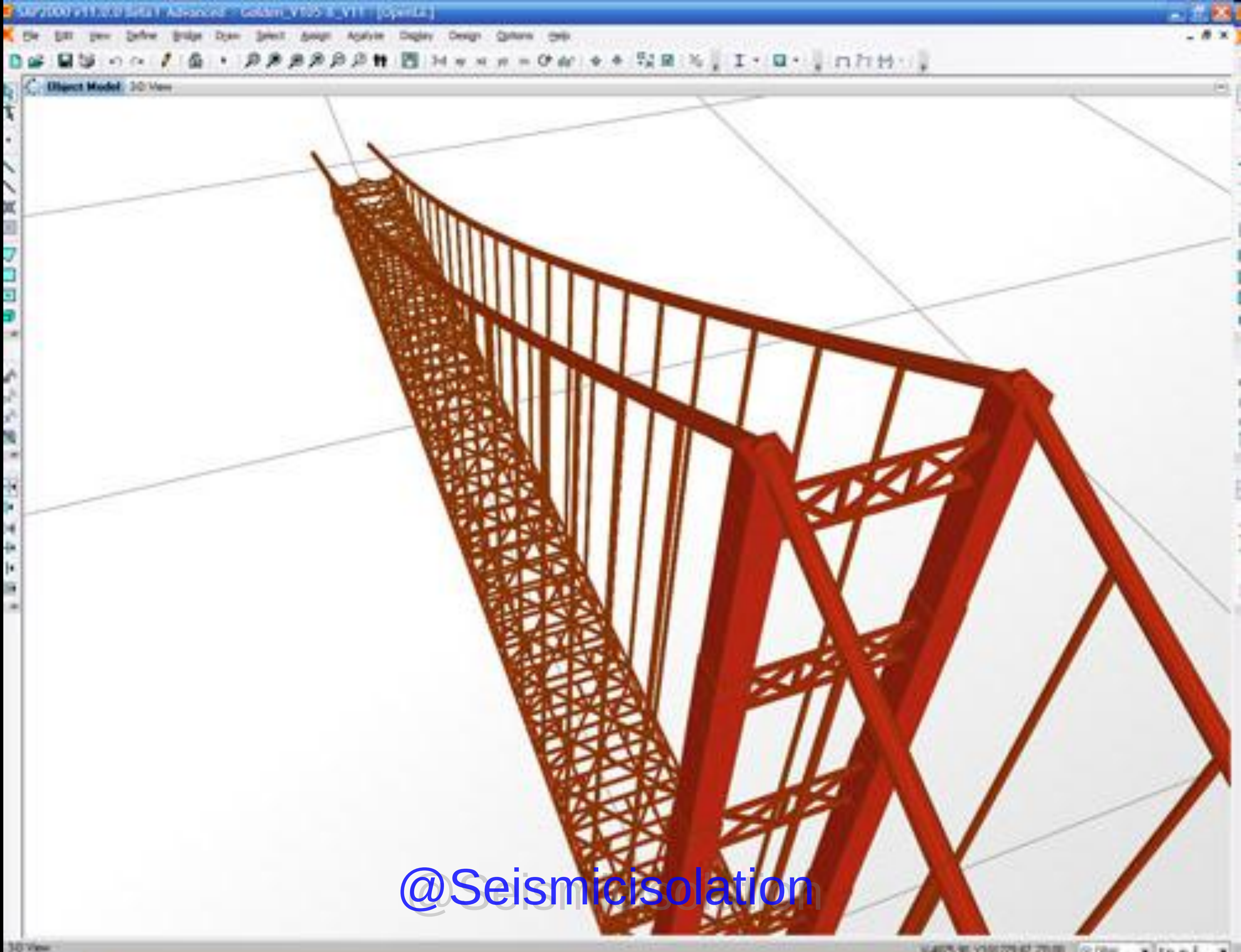
Combination of the above systems

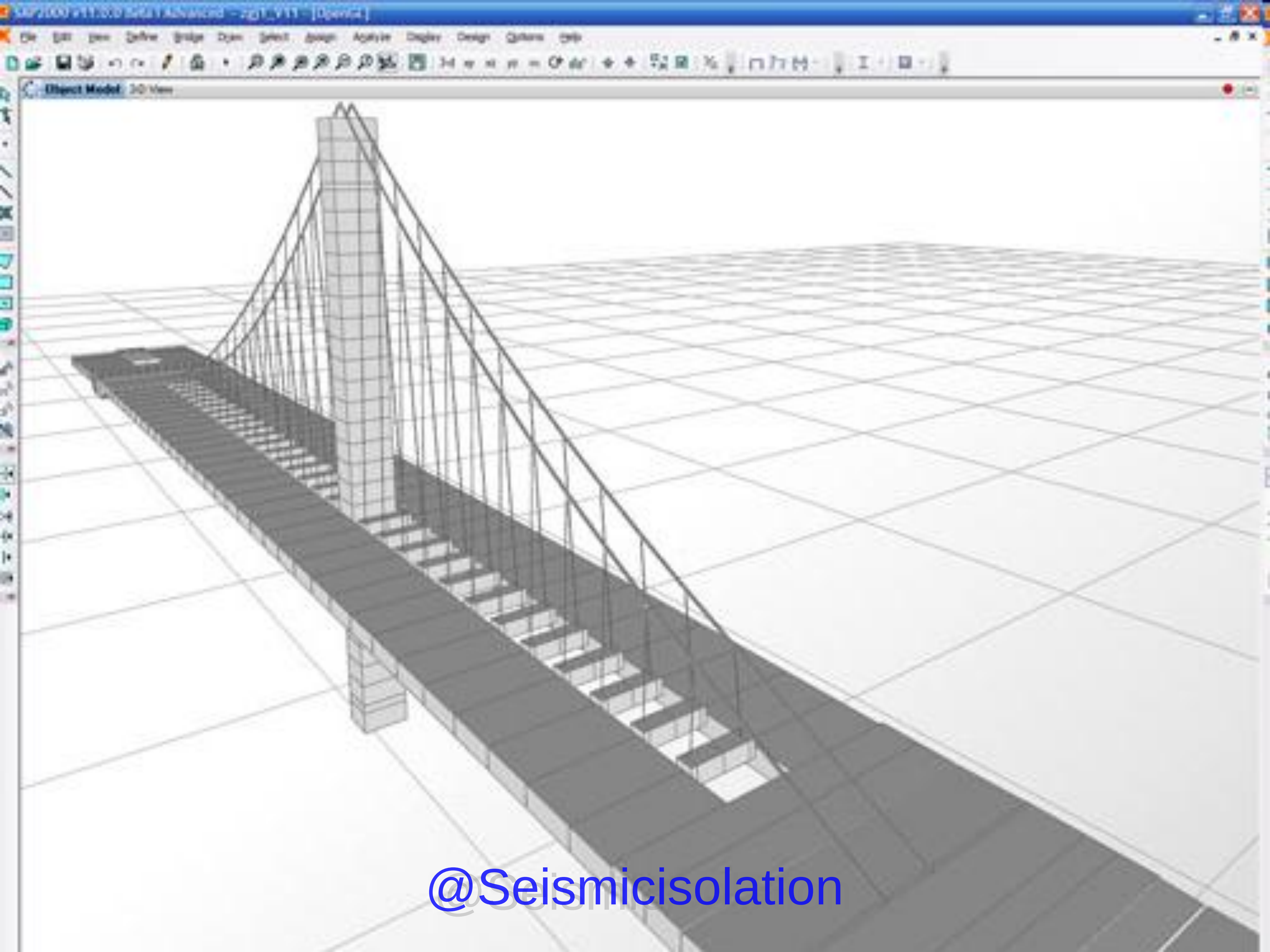


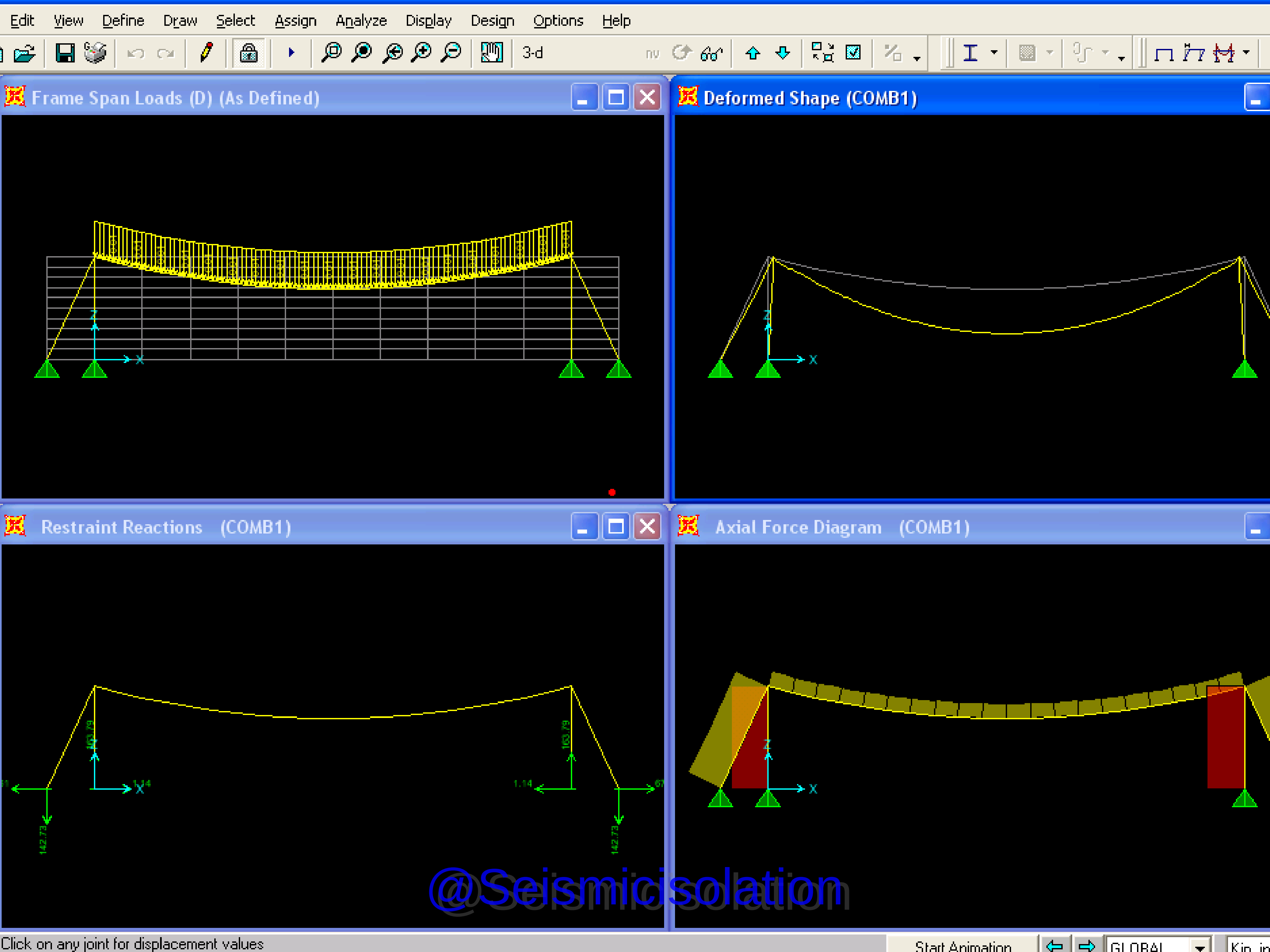
The deformation of a cable under its loads takes the shape of a funicular curve that is produced by only axial forces since a cable has negligible bending strength: polygonal and curved shapes (e.g. catenary shapes, parabolic shapes, circular shapes)


File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3D View

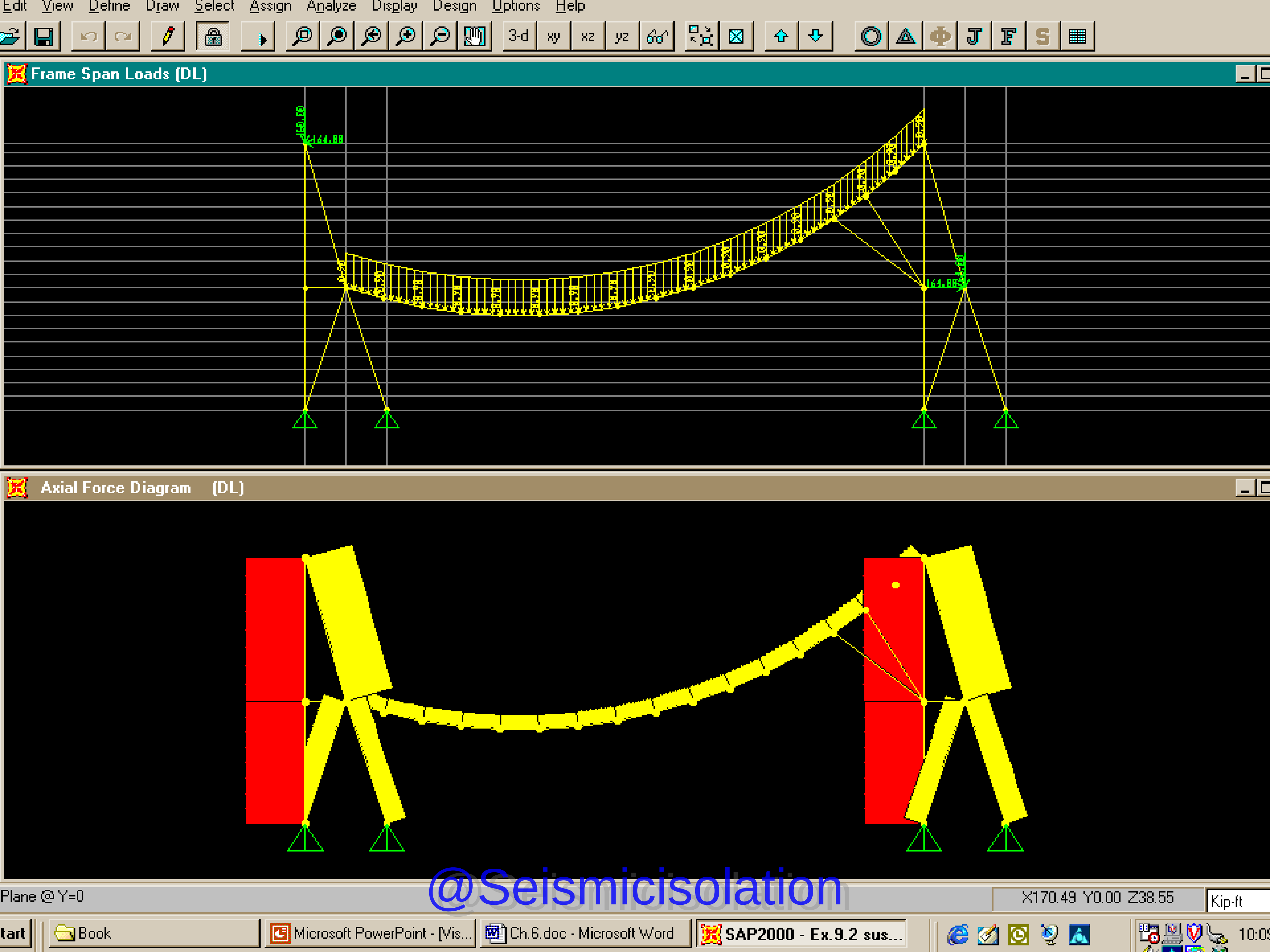


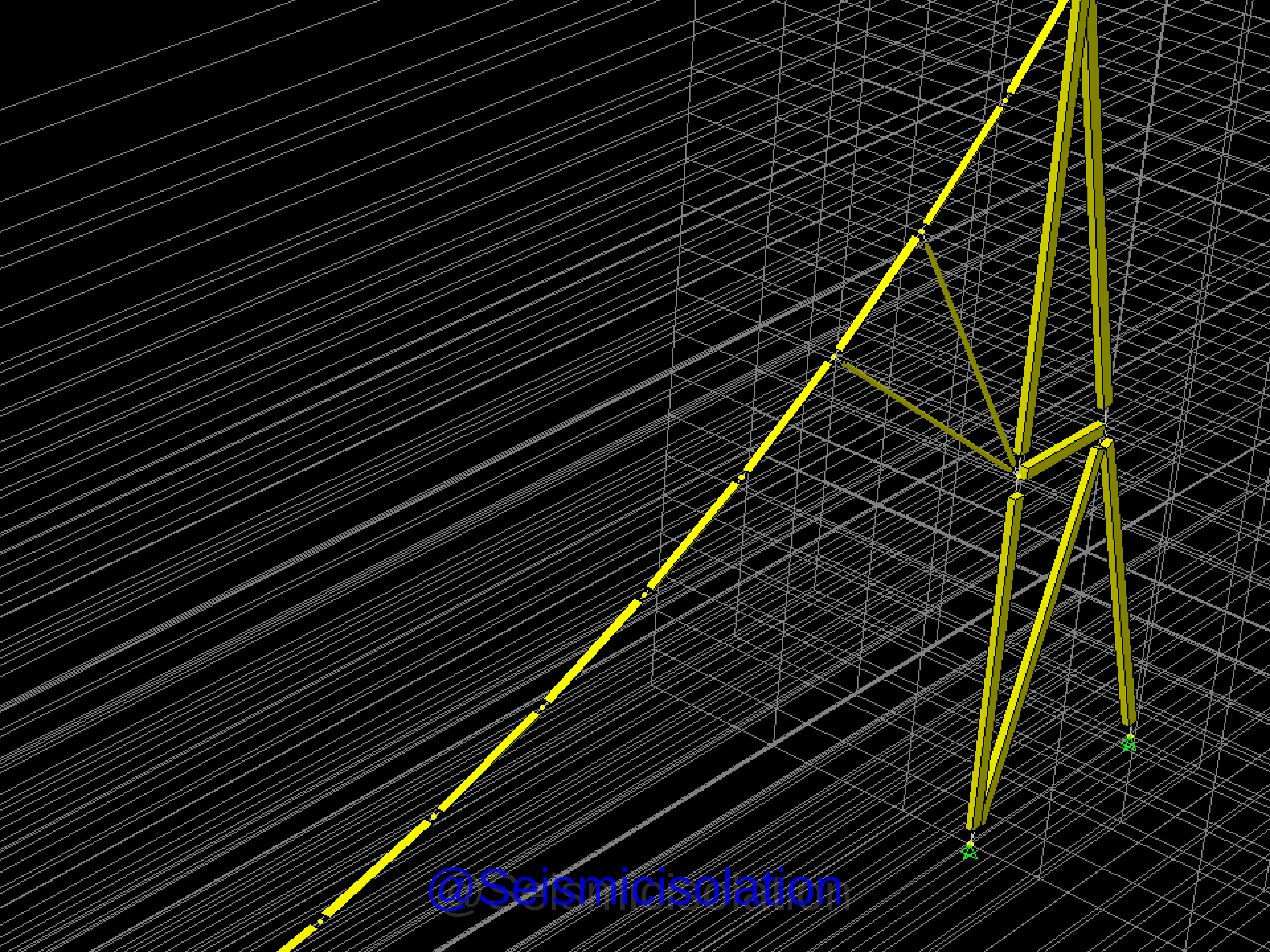


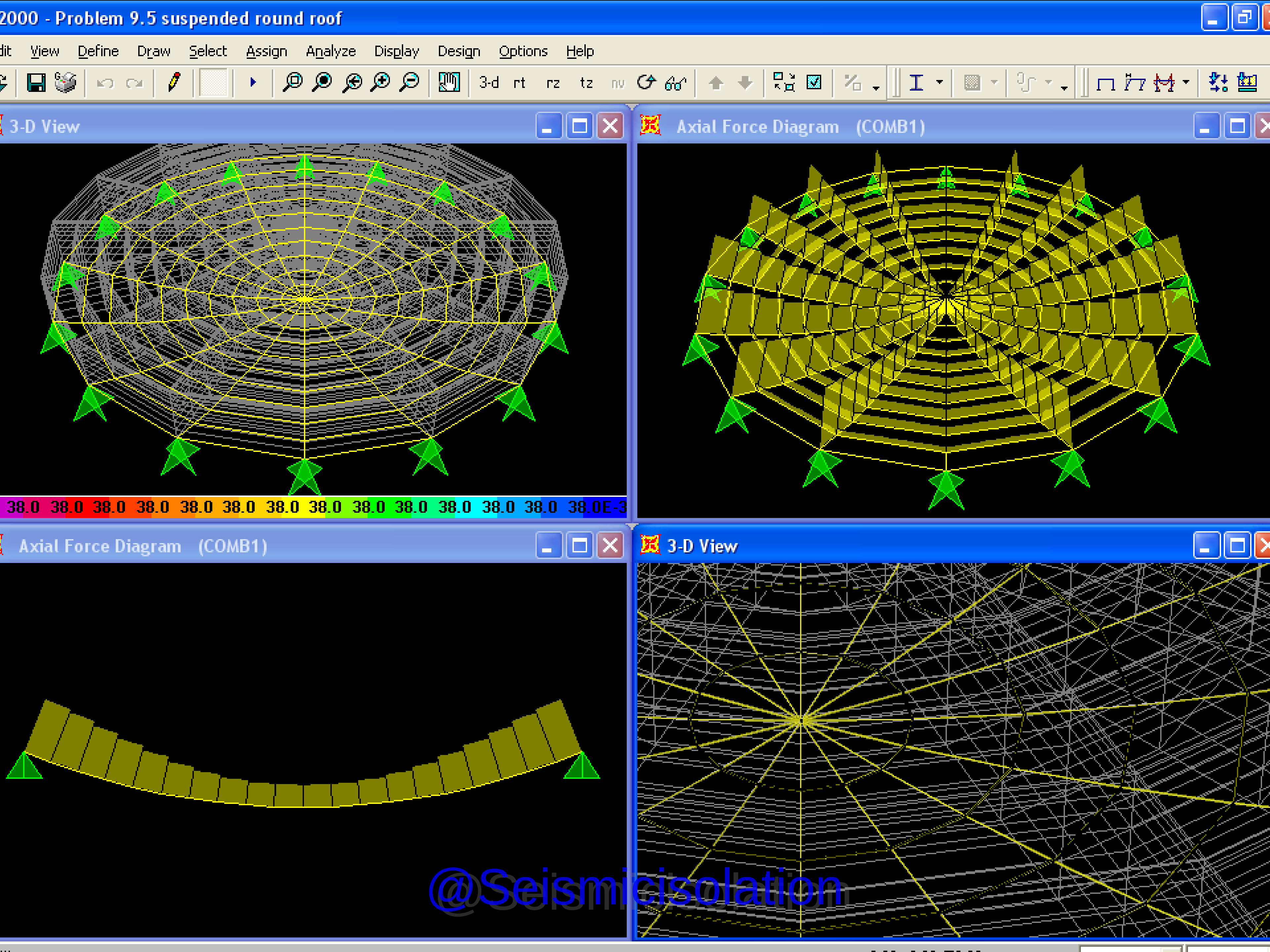

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3-D View

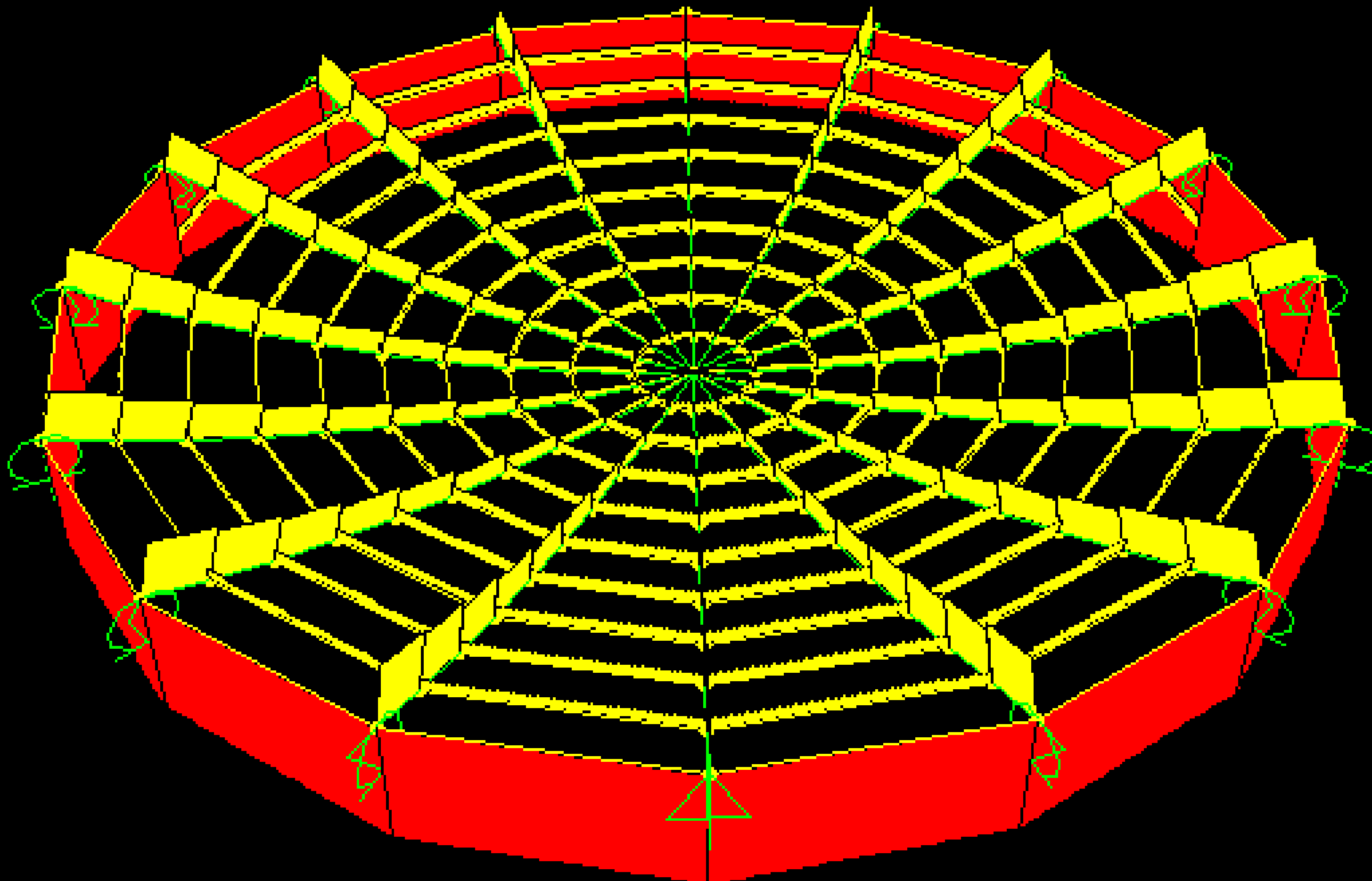


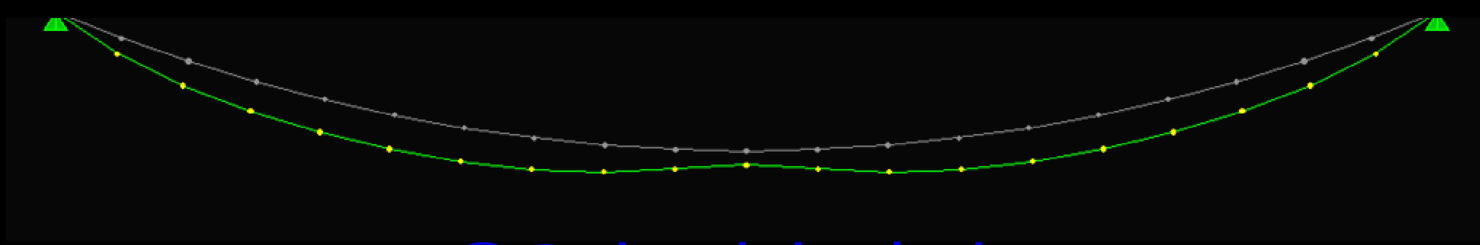
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
Frame Span Loads (D) (As Defined)
Deformed Shape (COMB1)
Restraint Reactions (COMB1)
Axial Force Diagram (COMB1)
@Seismicisolation
Click on any joint for displacement values
Start Animation
GLOBAL

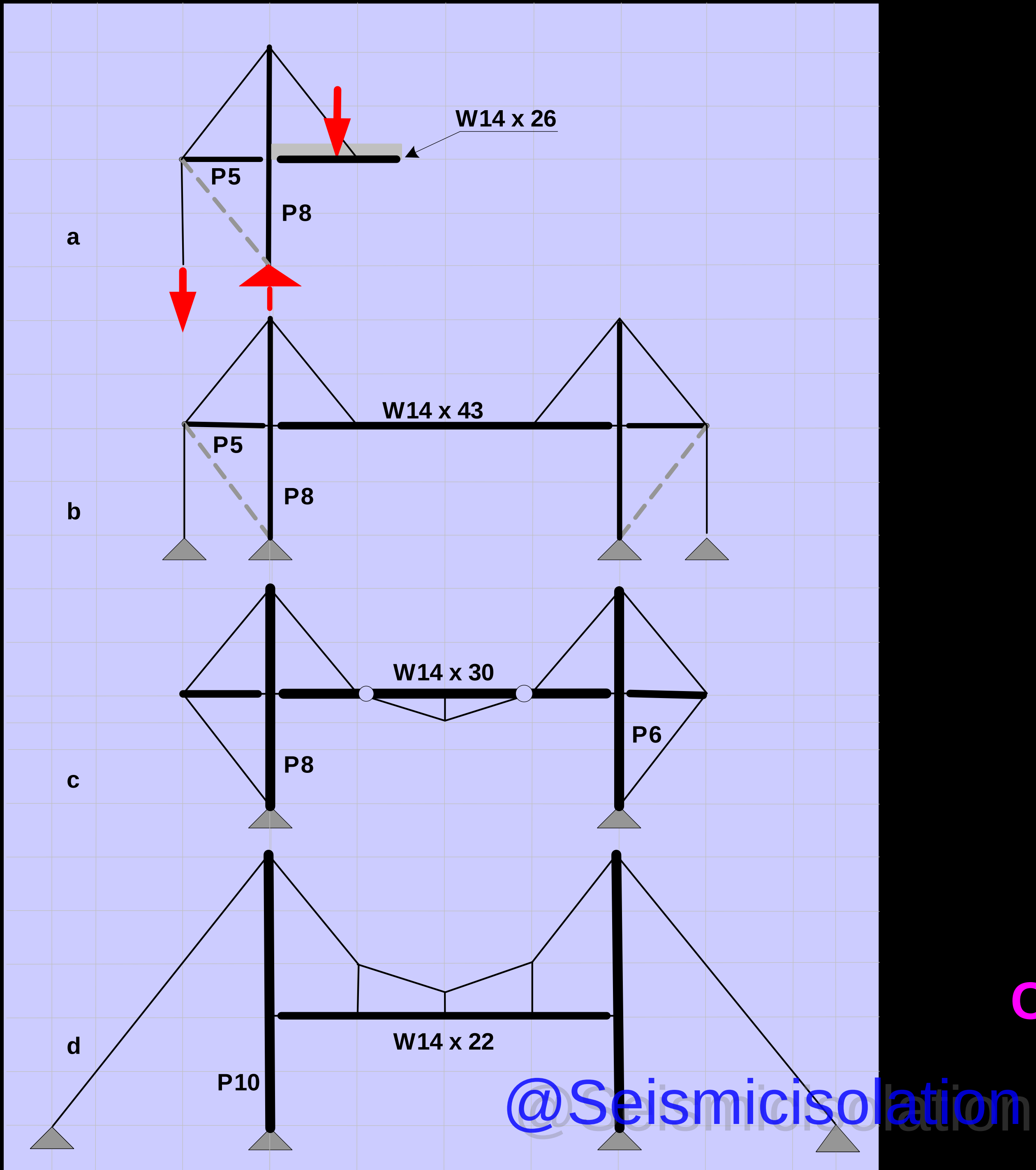

Edit View Define Draw Select Assign Analyze Display Design Options Help
Frame Span Loads (DL)
Axial Force Diagram (DL)
Plane @ Y=0
X170.49 Y0.00 Z38.55
Kip-ft
Book
Microsoft PowerPoint - [Vis...
Ch.6.doc - Microsoft Word
SAP2000 - Ex.9.2 sus...



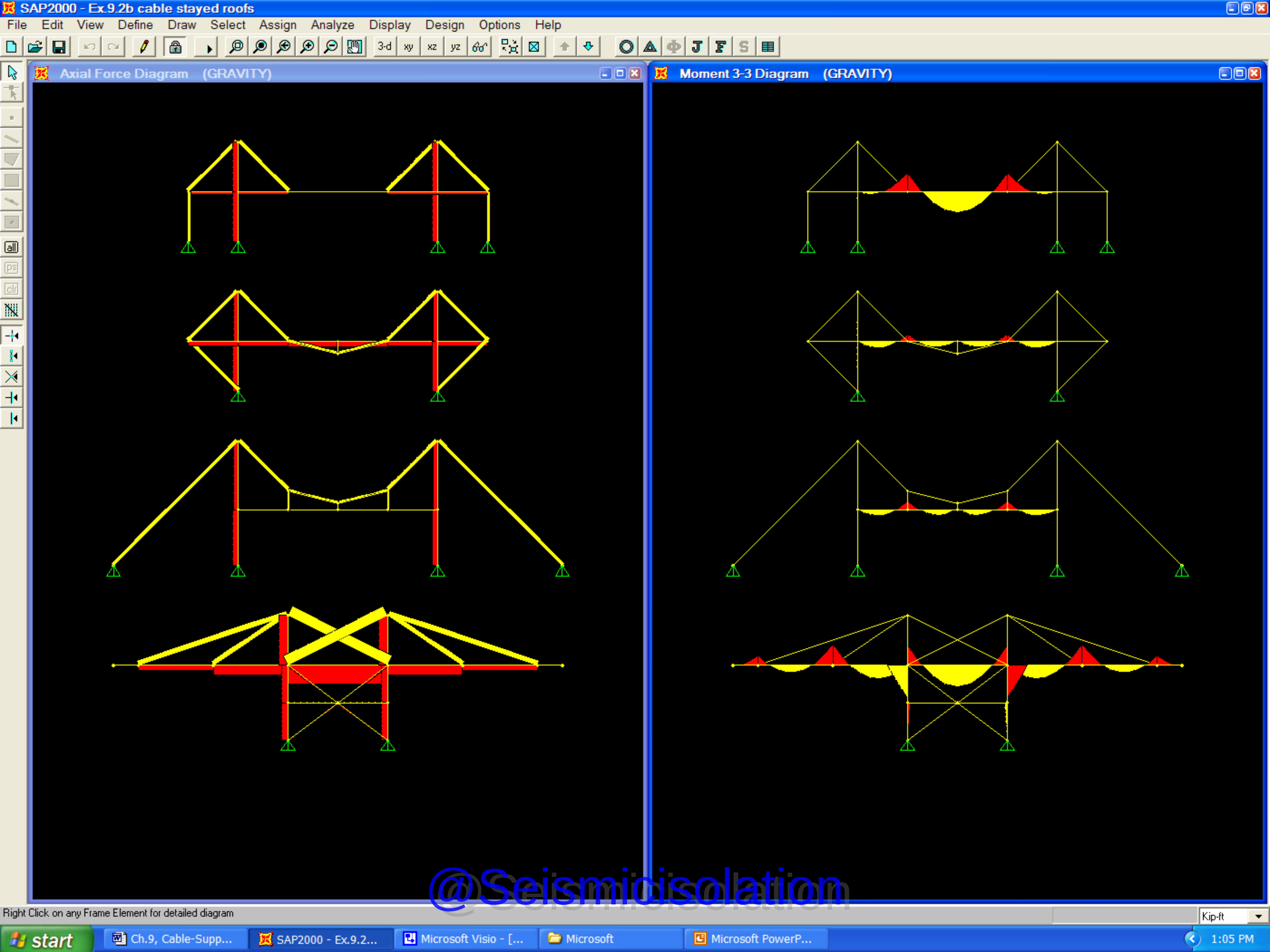

2000 - Problem 9.5 suspended round roof
dit View Define Draw Select Assign Analyze Display Design Options Help
3-d rt rz tz
3-D View
Axial Force Diagram (COMB1)
38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0 38.0E-3
Axial Force Diagram (COMB1)
3-D View
@Seismicisolation

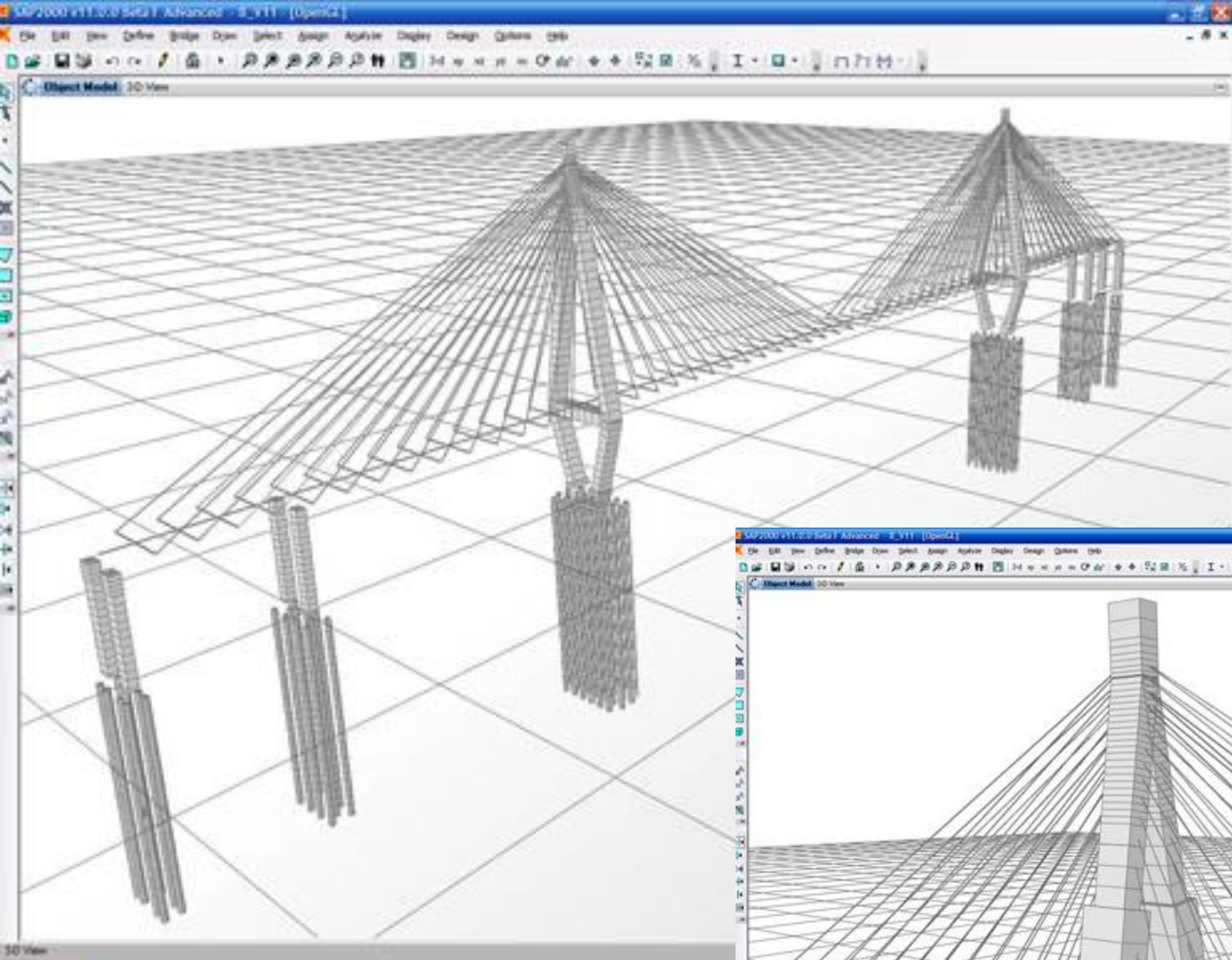


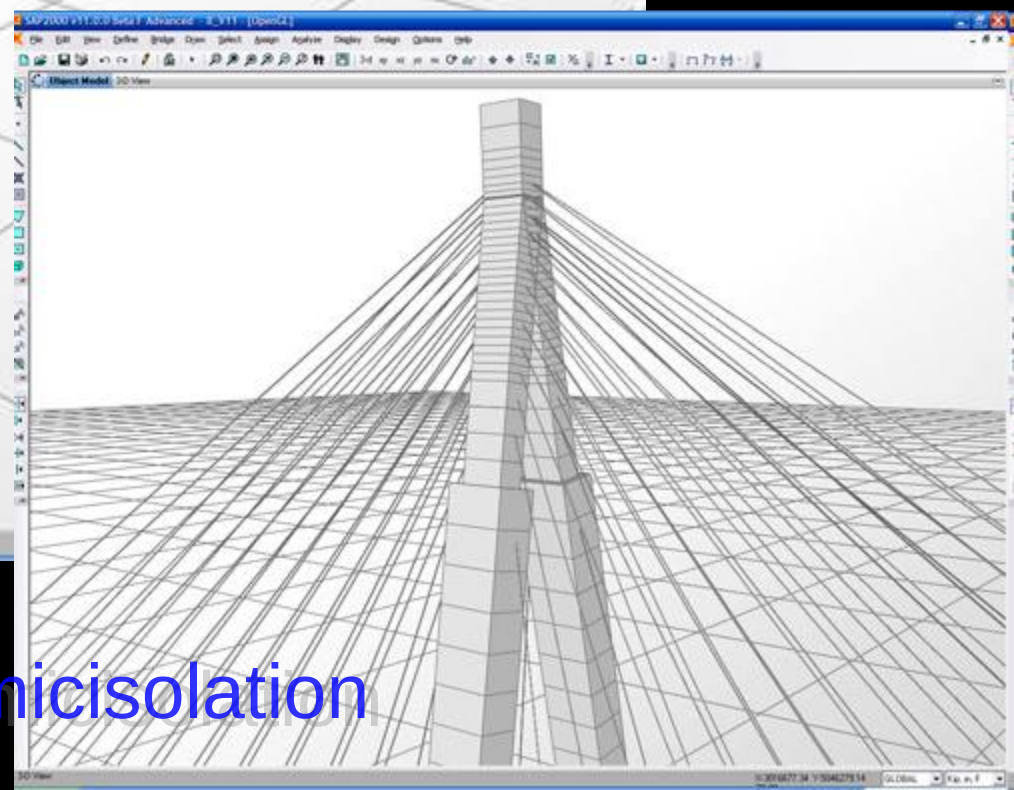


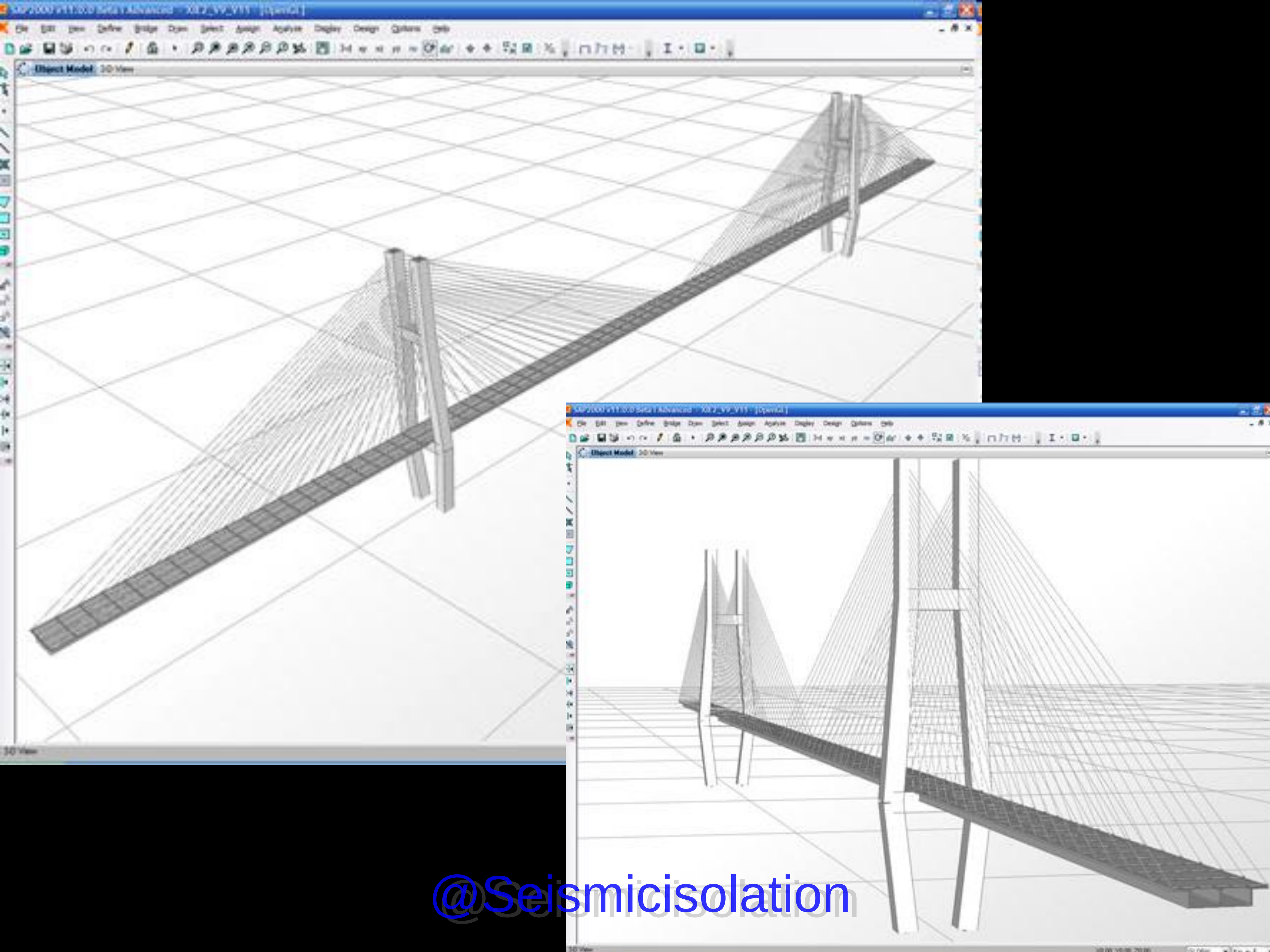


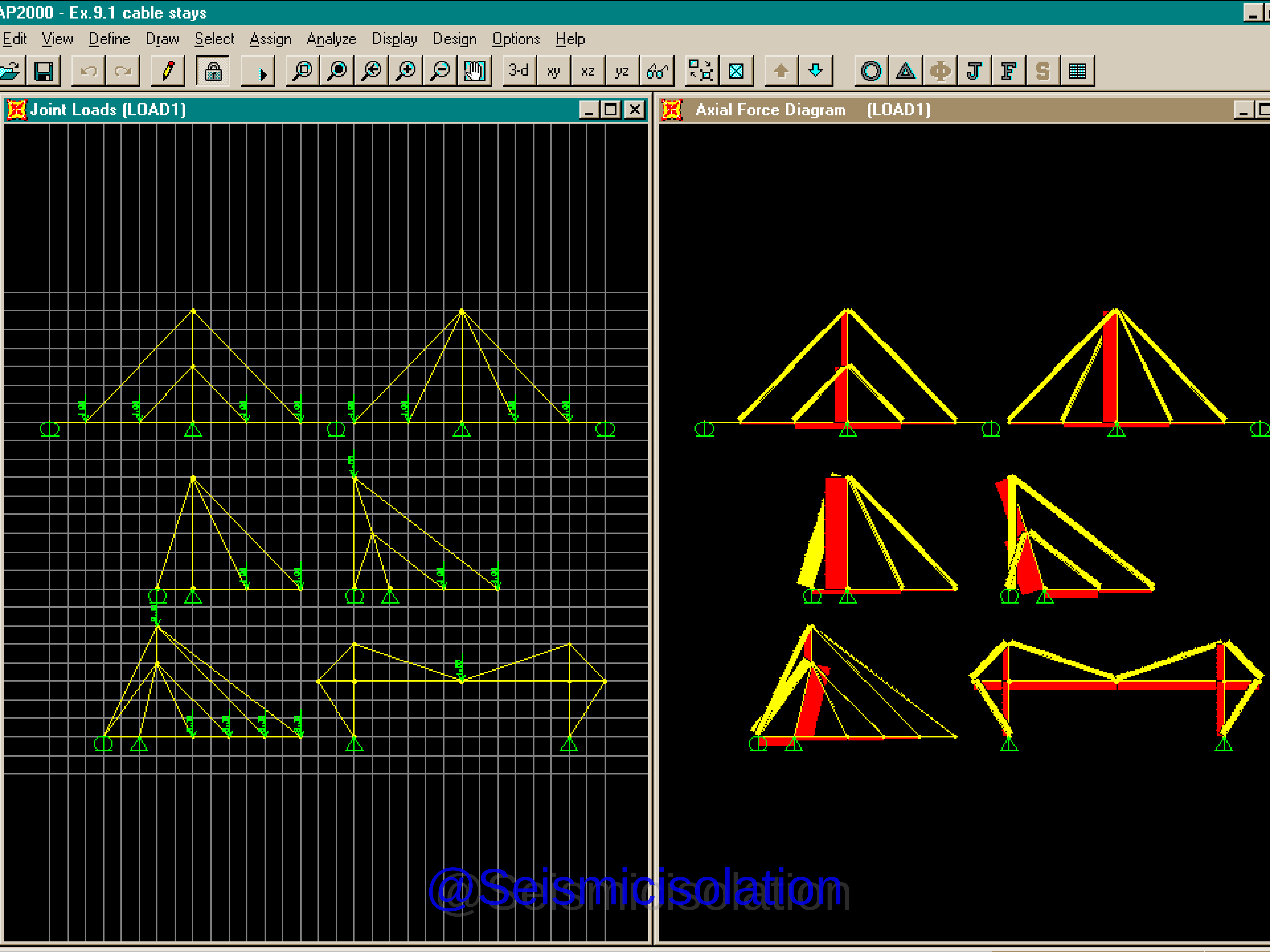
Cable-Stayed Roof Structures

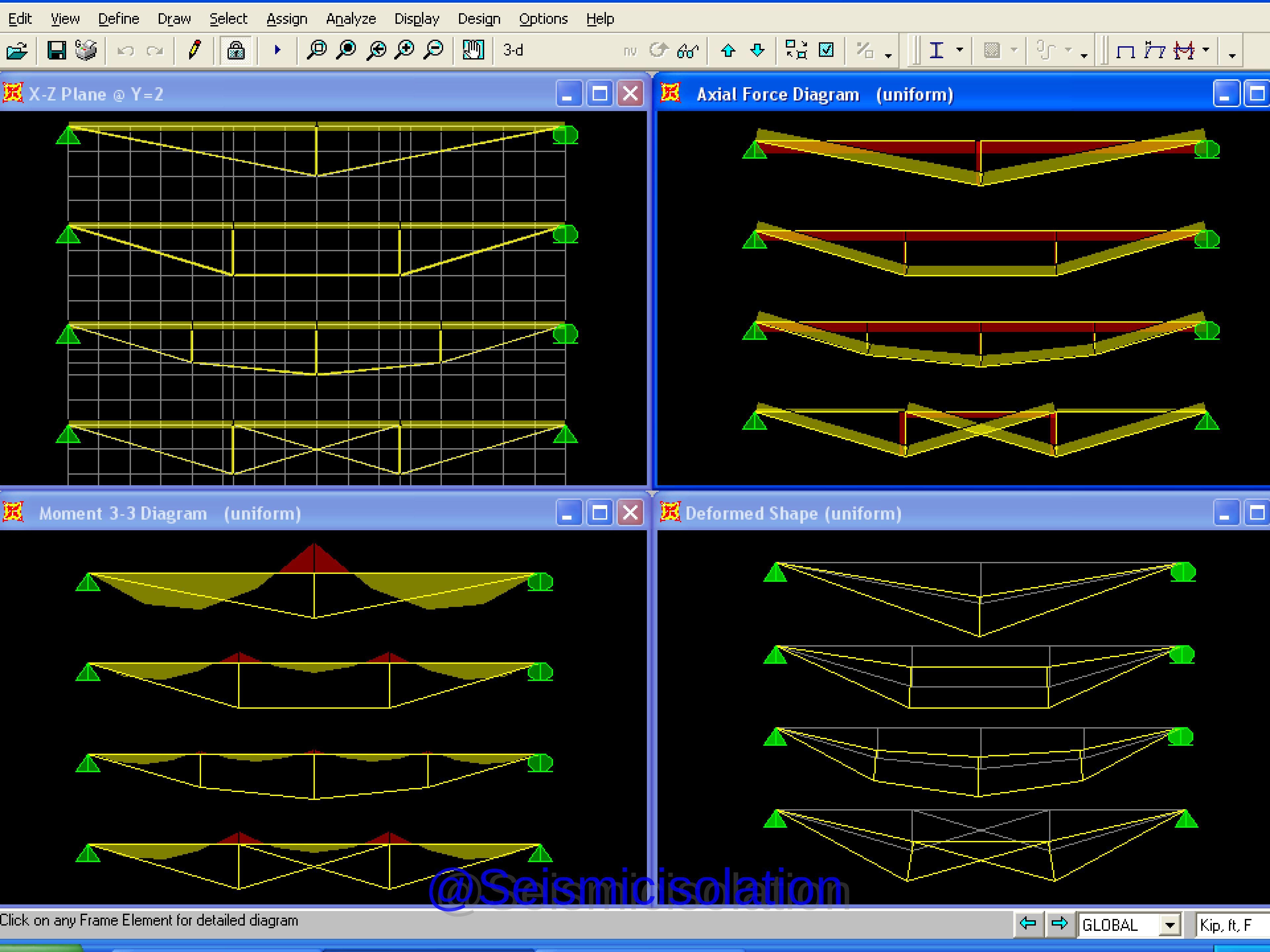

SAP2000 - Ex.9.2b cable stayed roofs
File Edit View Define Draw Select Assign Analyze Display Design Options Help
Axial Force Diagram (GRAVITY)
Moment 3-3 Diagram (GRAVITY)
Right Click on any Frame Element for detailed diagram
Kip-ft
start
Ch.9, Cable-Supp...
SAP2000 - Ex.9.2...
Microsoft Visio - [...
Microsoft
Microsoft PowerP...
1:05 PM

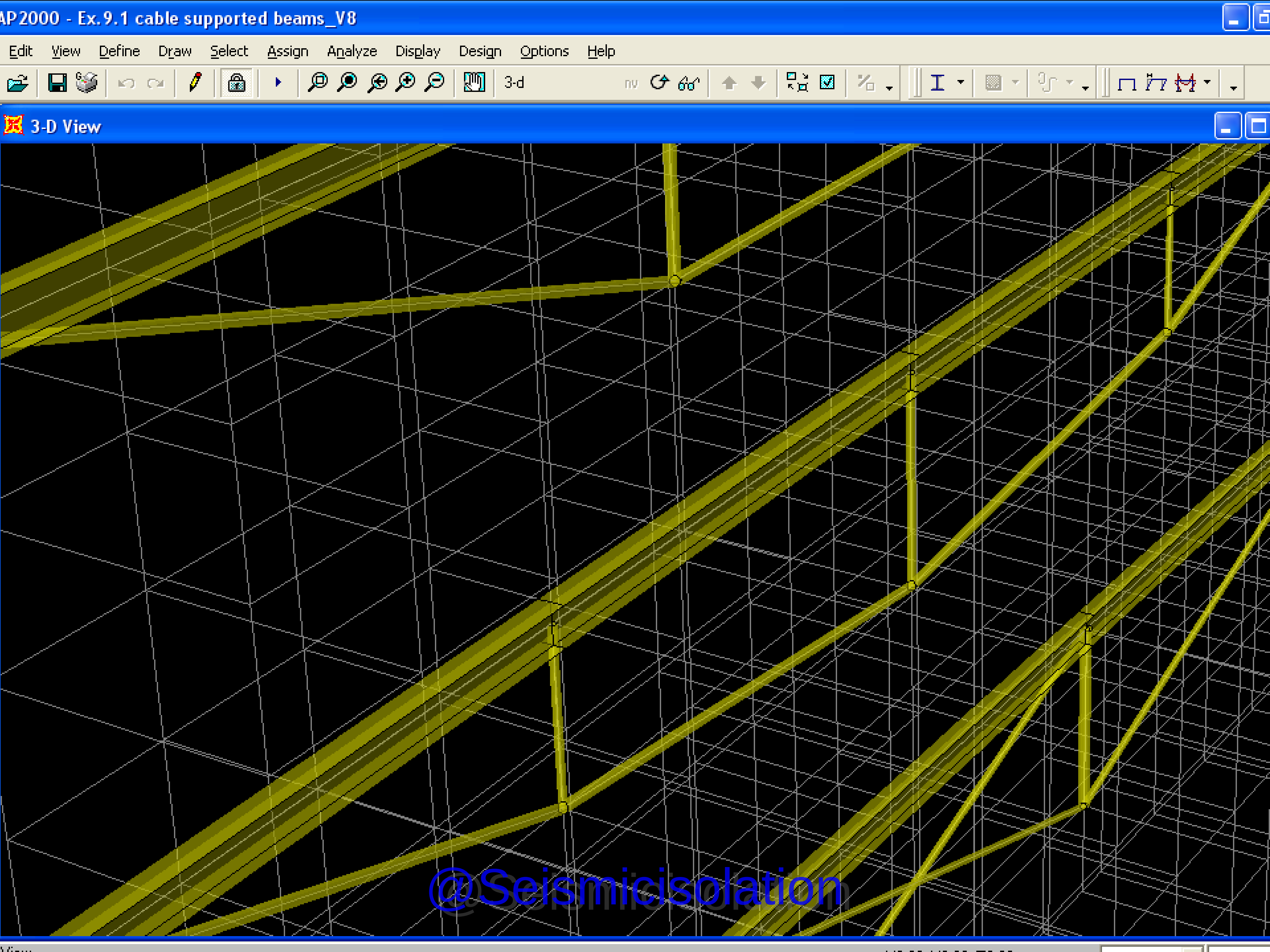

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model

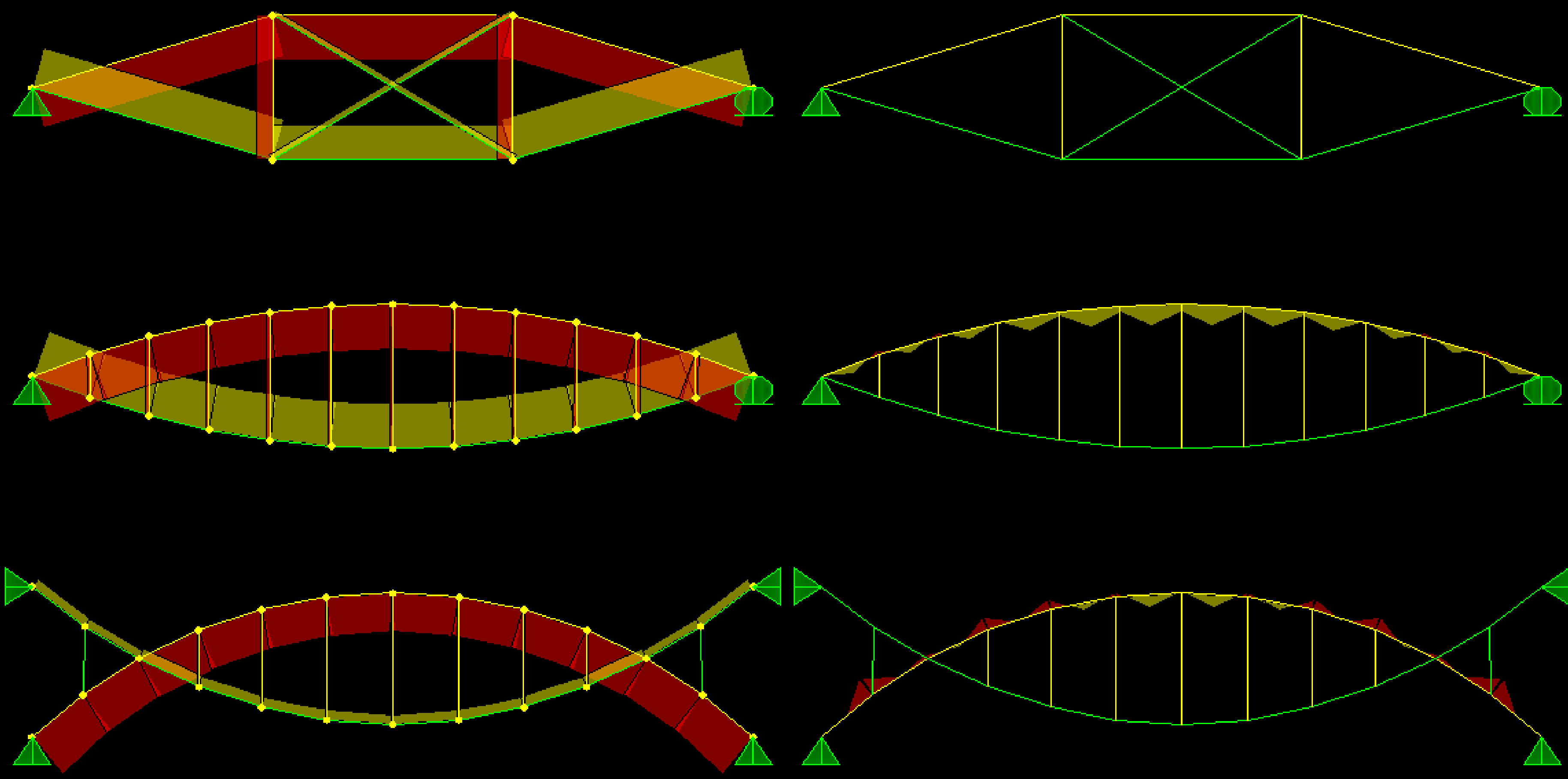

Object Model

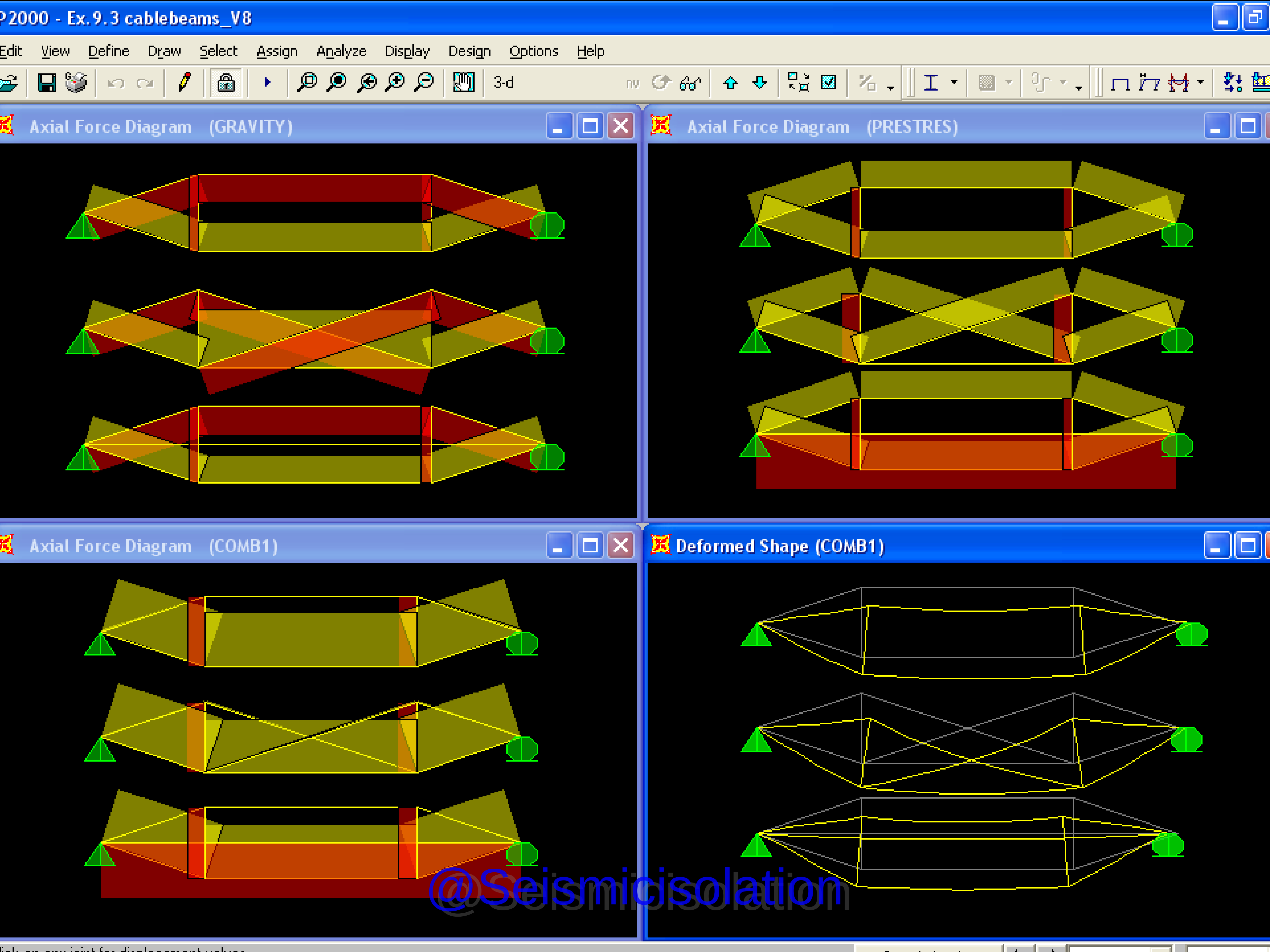

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model
3-D View

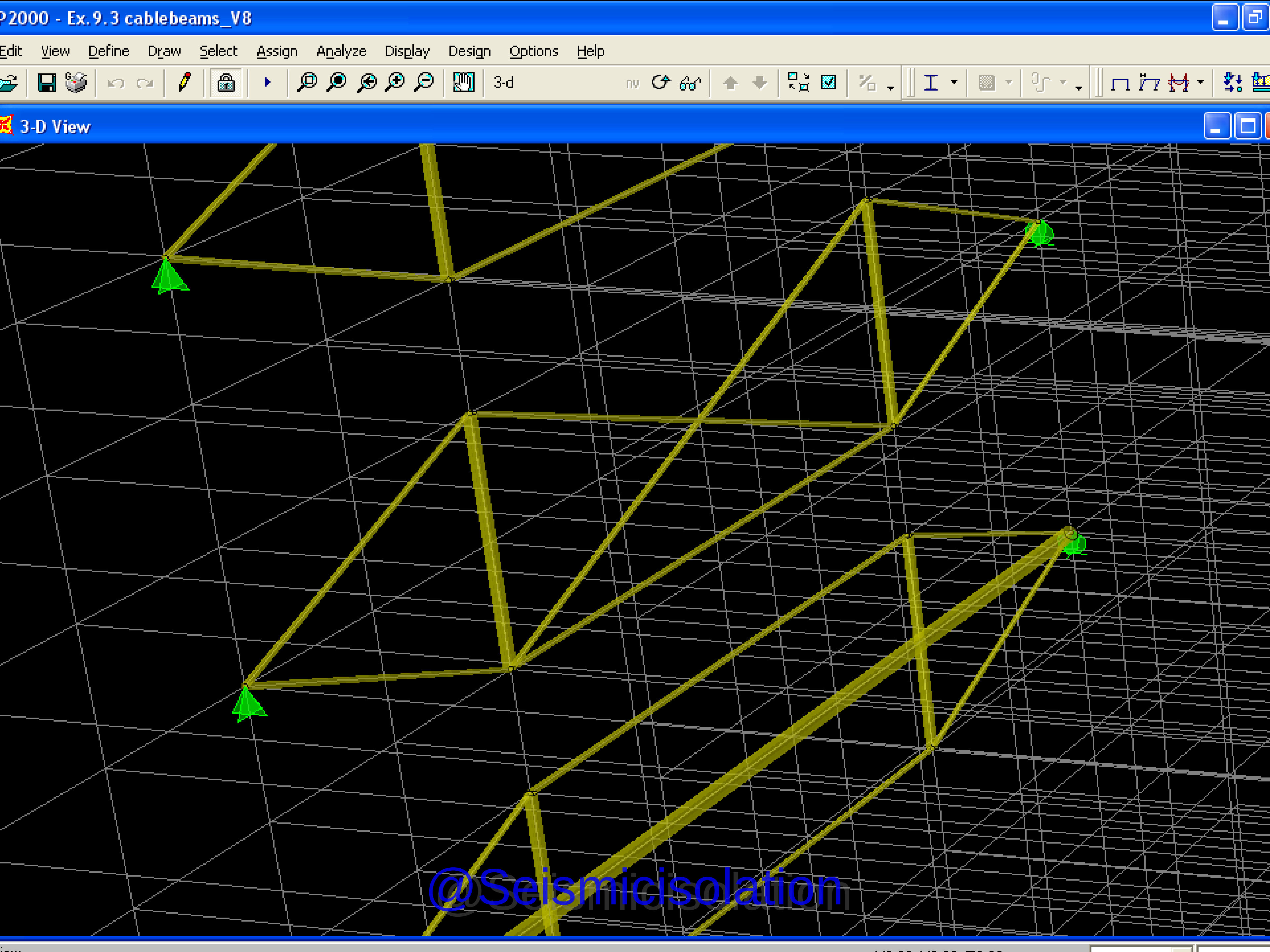

AP2000 - Ex.9.1 cable stays
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Joint Loads (LOAD1)
Axial Force Diagram (LOAD1)

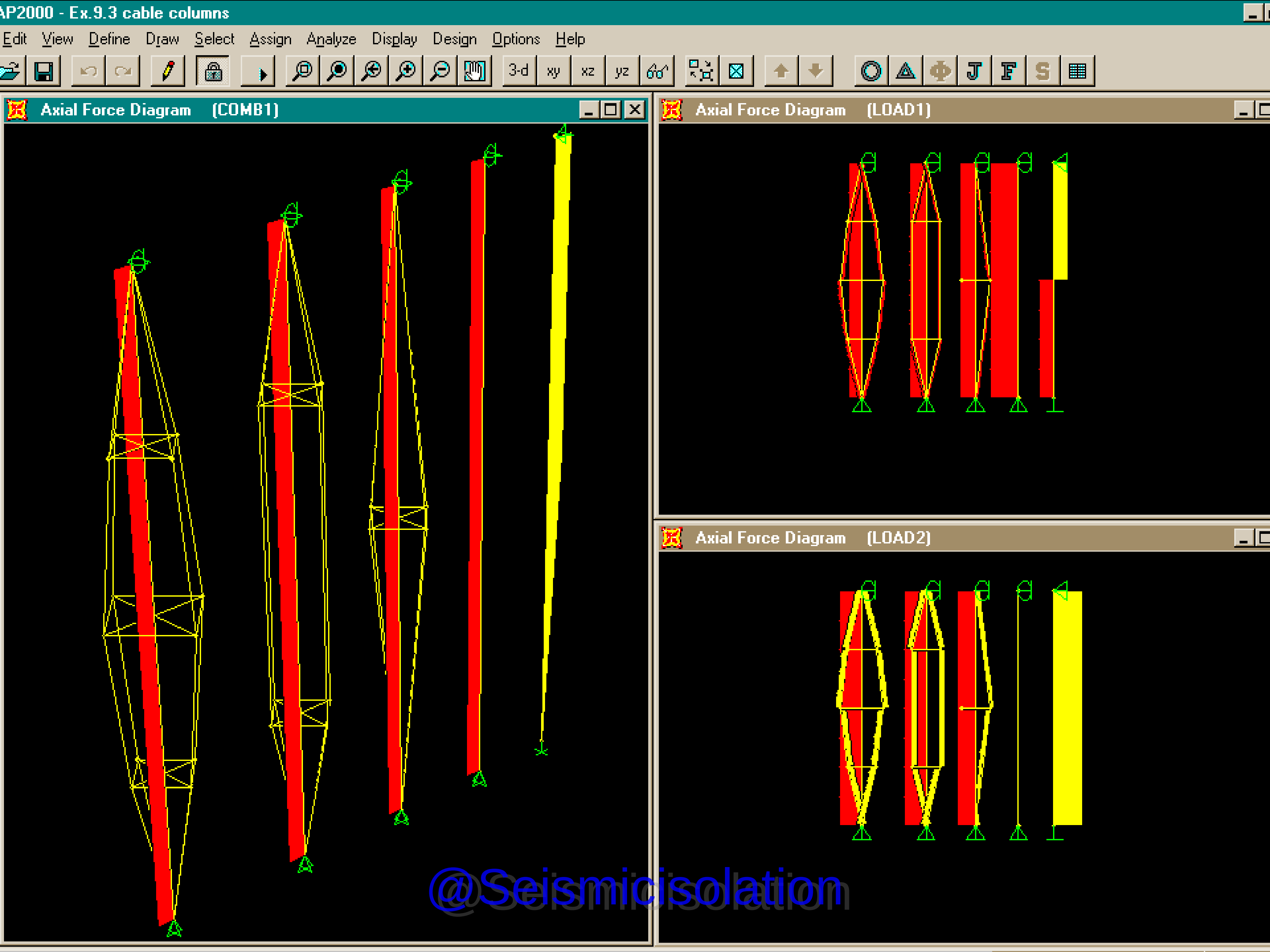

Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
X-Z Plane @ Y=2
Axial Force Diagram (uniform)
Moment 3-3 Diagram (uniform)
Deformed Shape (uniform)
@Seismicisolation
Click on any Frame Element for detailed diagram
GLOBAL
Kip, ft, F

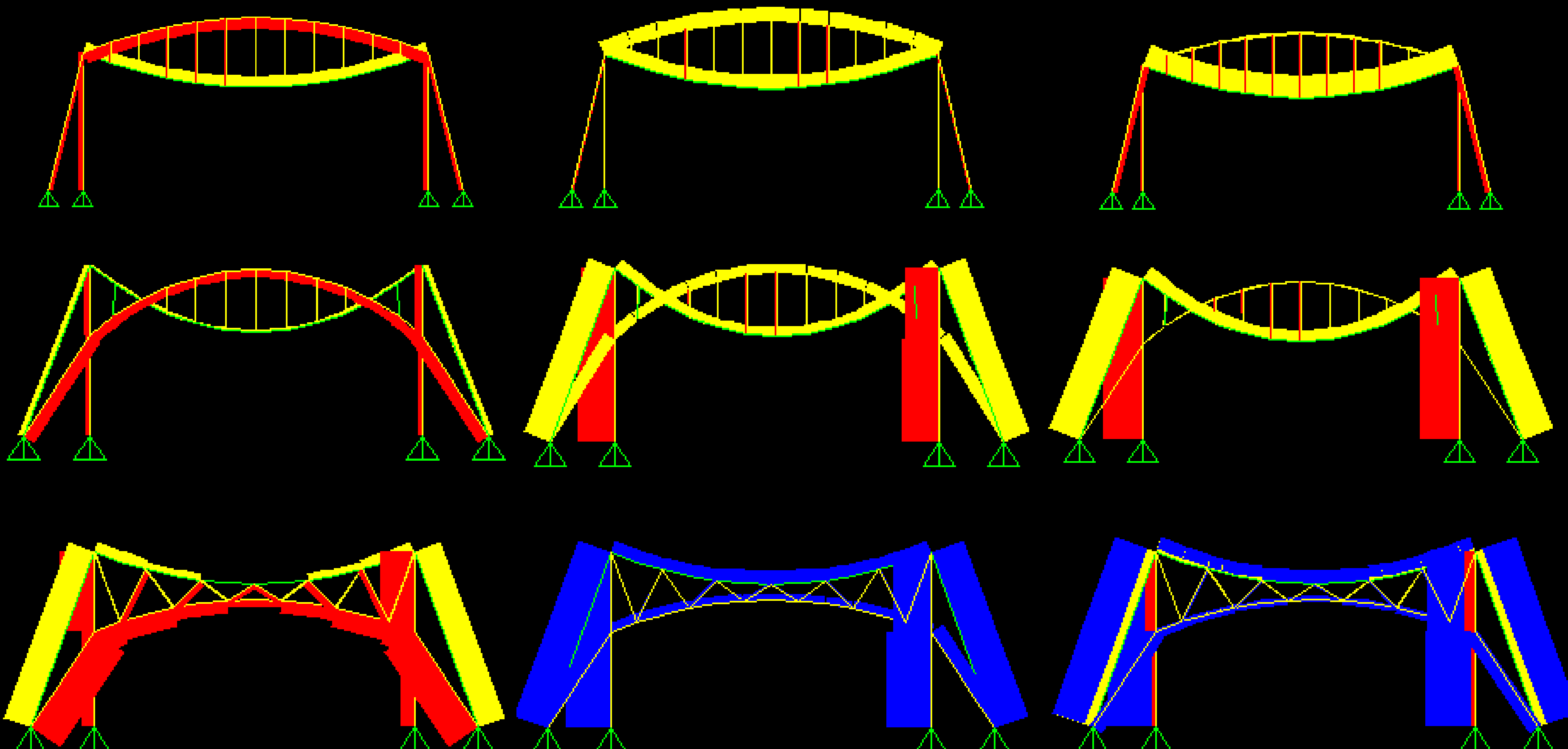

AP2000 - Ex.9.1 cable supported beams_V8
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
3-D View
@Seismicisolation




P2000 - Ex.9.3 cablebeams_V8
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
Axial Force Diagram (GRAVITY)
Axial Force Diagram (PRESTRES)
Axial Force Diagram (COMB1)
Deformed Shape (COMB1)
@Seismicisolation


P2000 - Ex.9.3 cablebeams_V8
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
3-D View


AP2000 - Ex.9.3 cable columns
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (COMB1)
Axial Force Diagram (LOAD1)
Axial Force Diagram (LOAD2)



@Seismicisolation

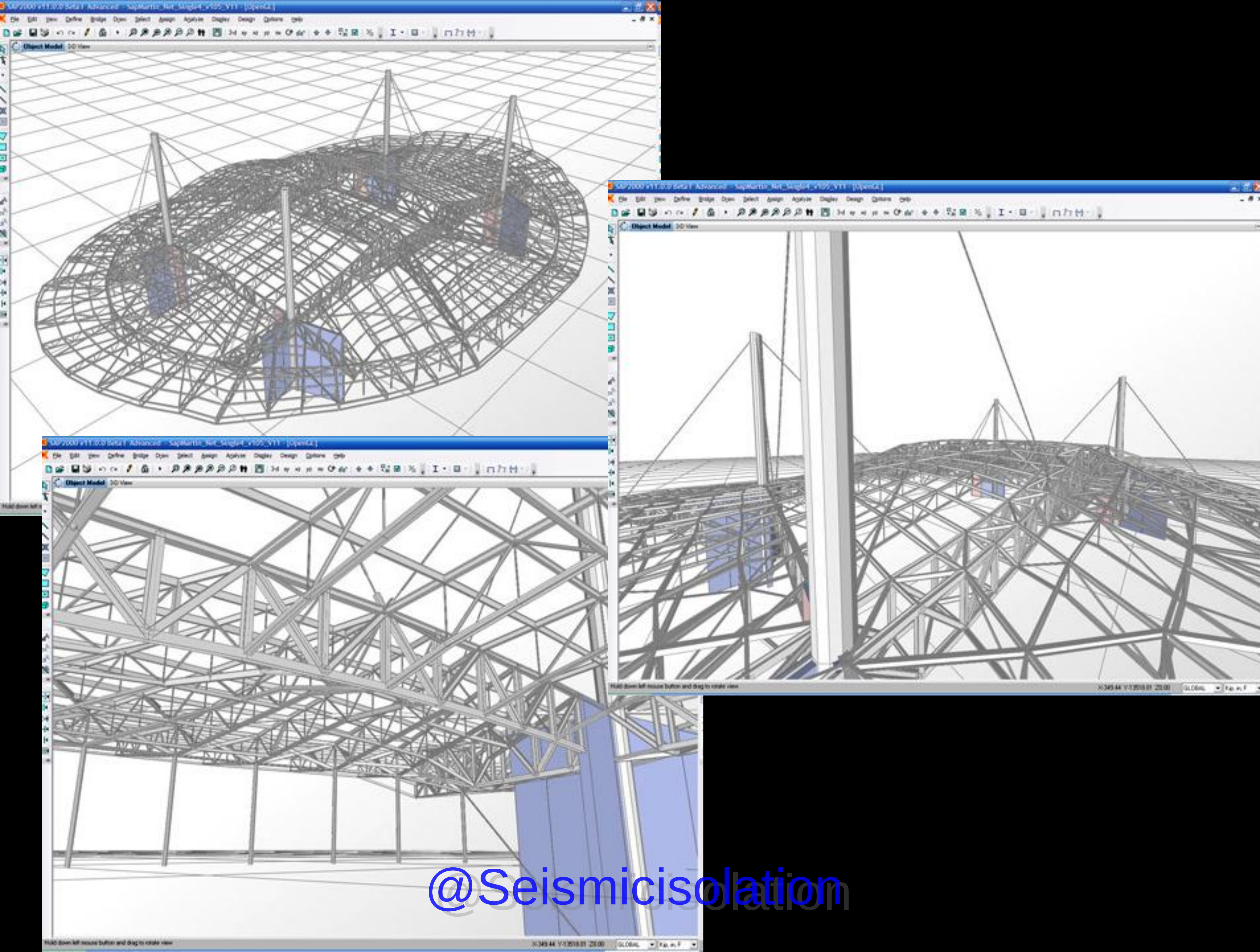
Shanghai-Pudong International Airport, 2001, Paul Andreu principal architect, Coyne et Bellier structural engineers

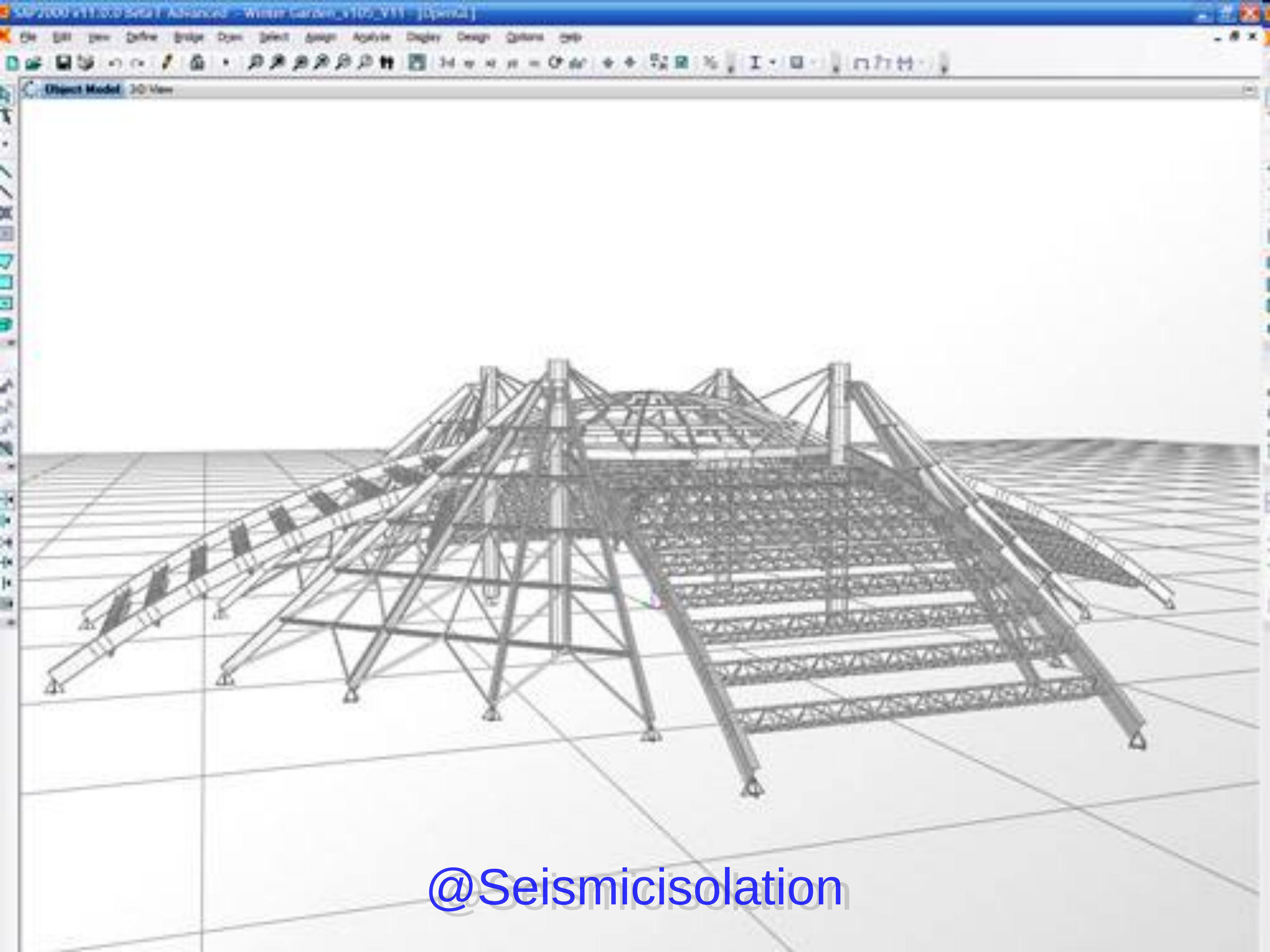

@Seismicisolation
14 6 2001

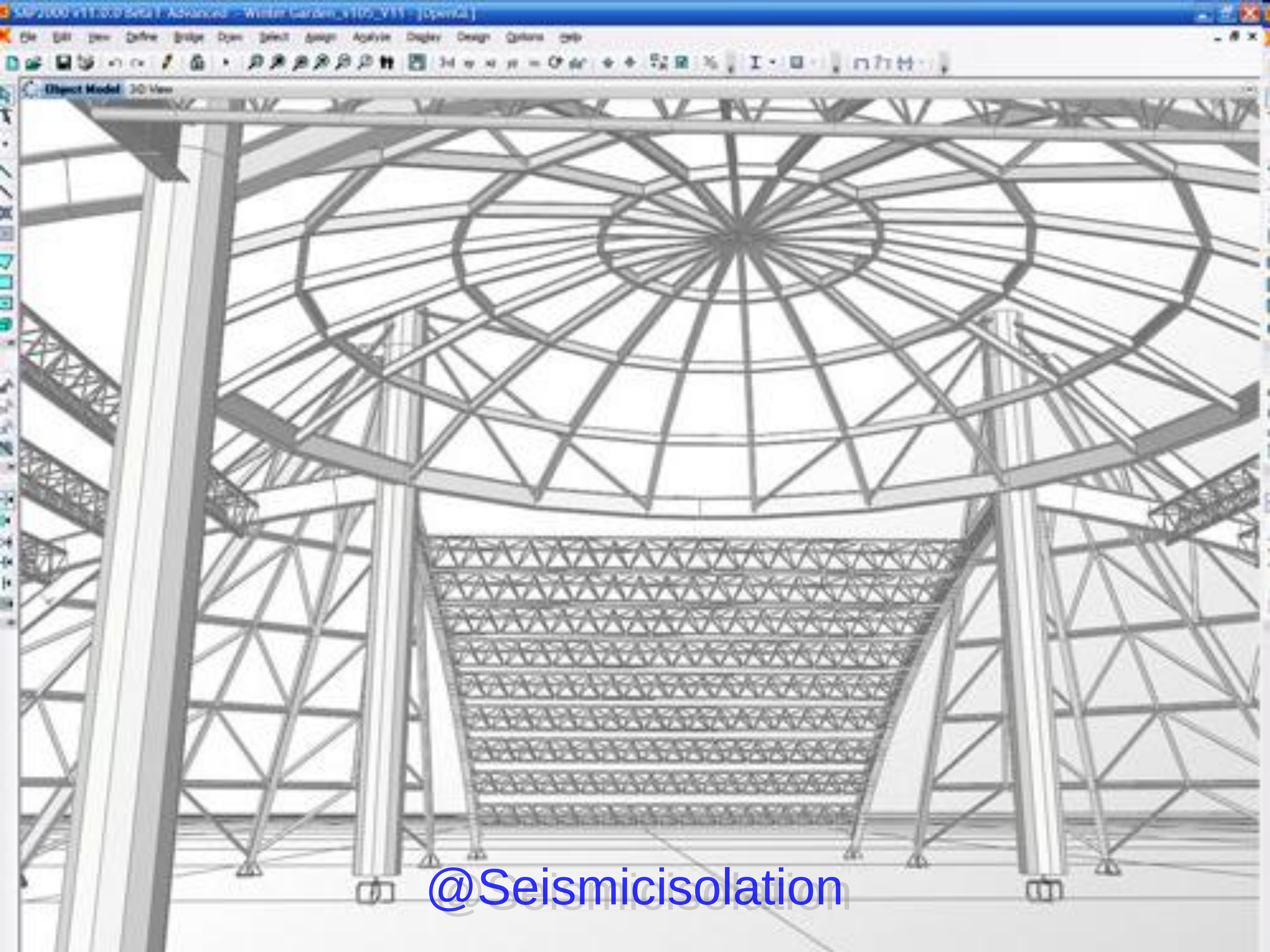

5 国内出发办票岛
Domestic Departures Check-in Island
14 6 2001

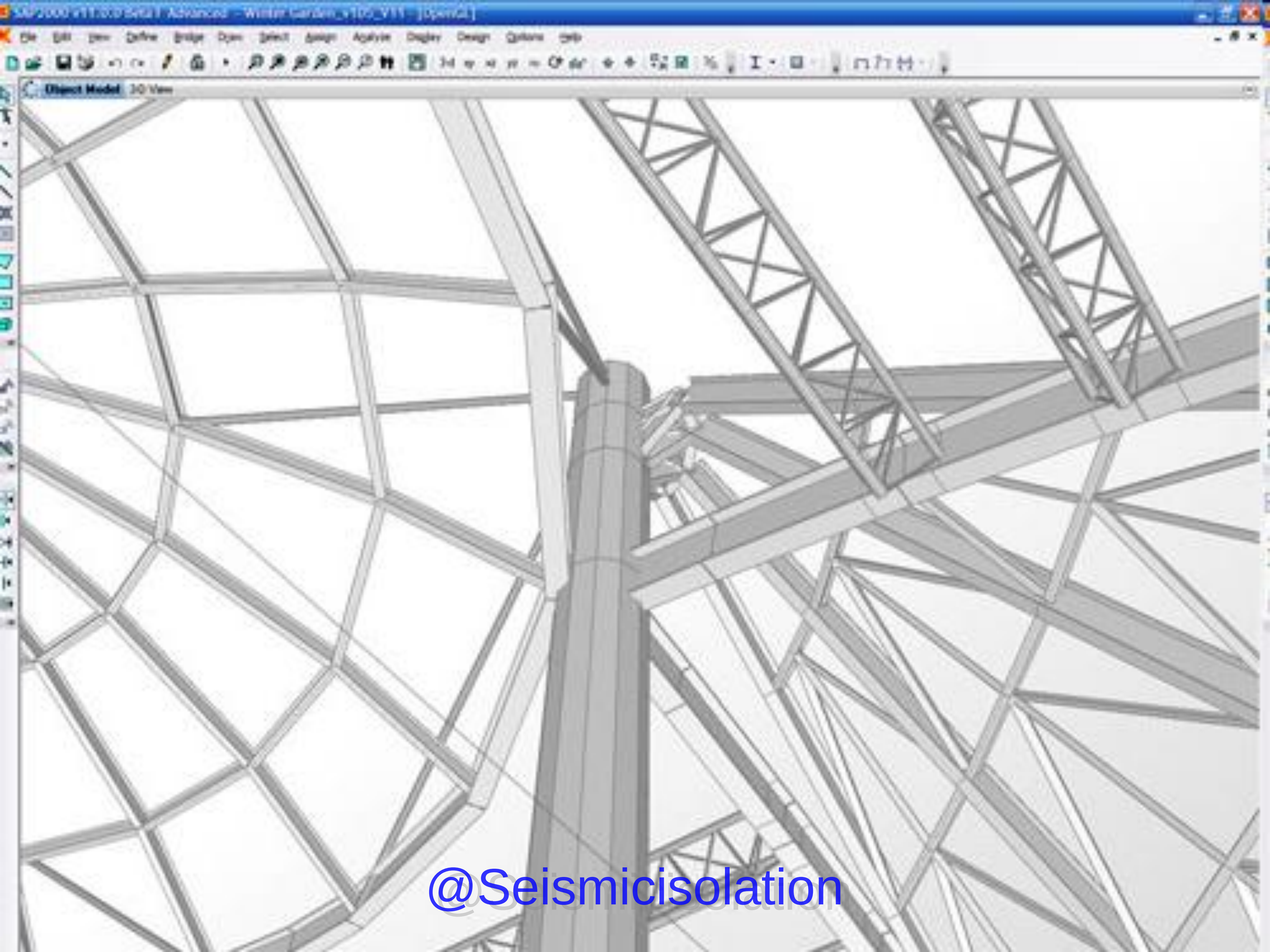
Debis Theater, Marlene Dietrich Platz, Berlin, 1998, Renzo Piano

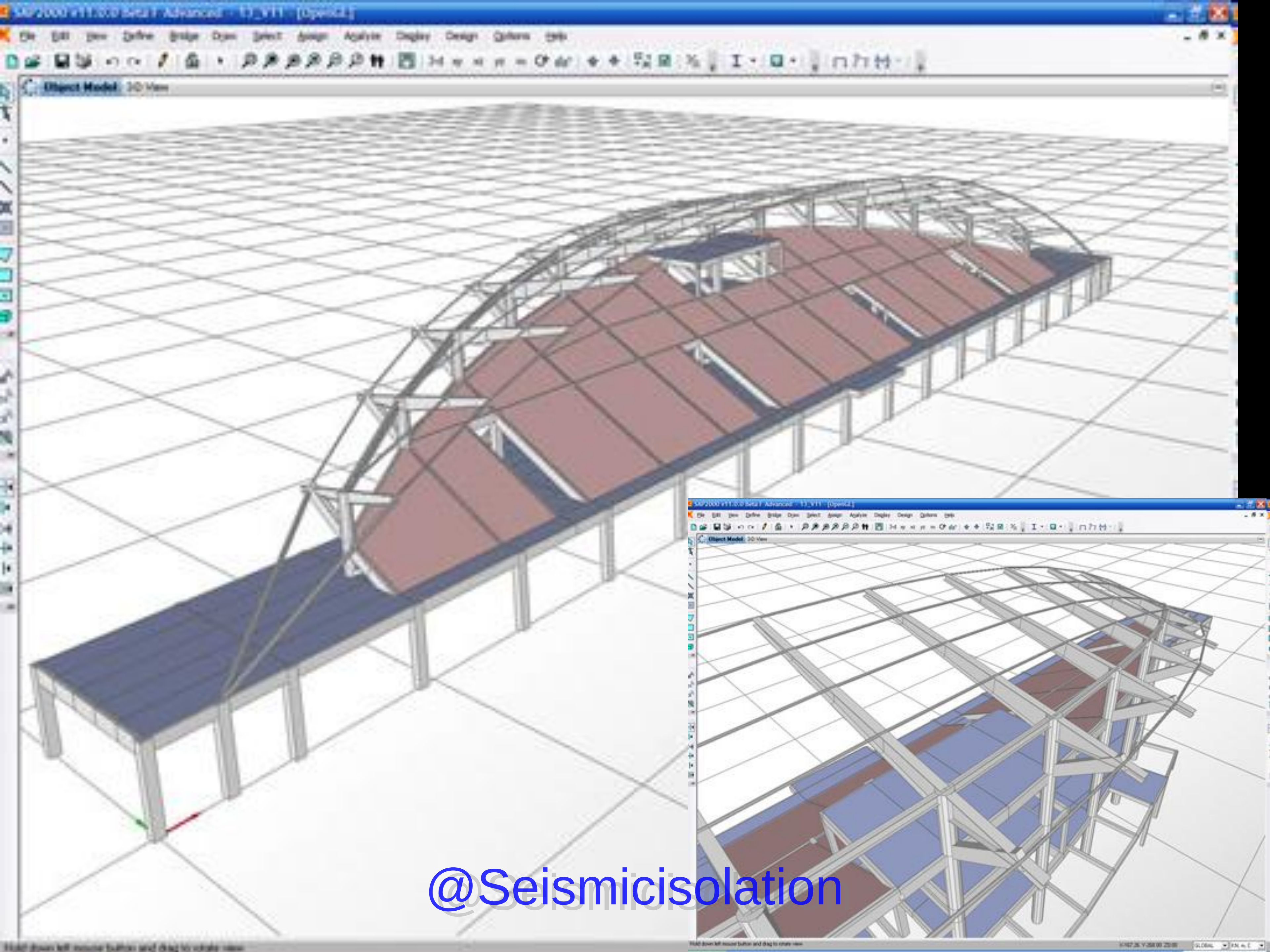

24 9 2001




File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model

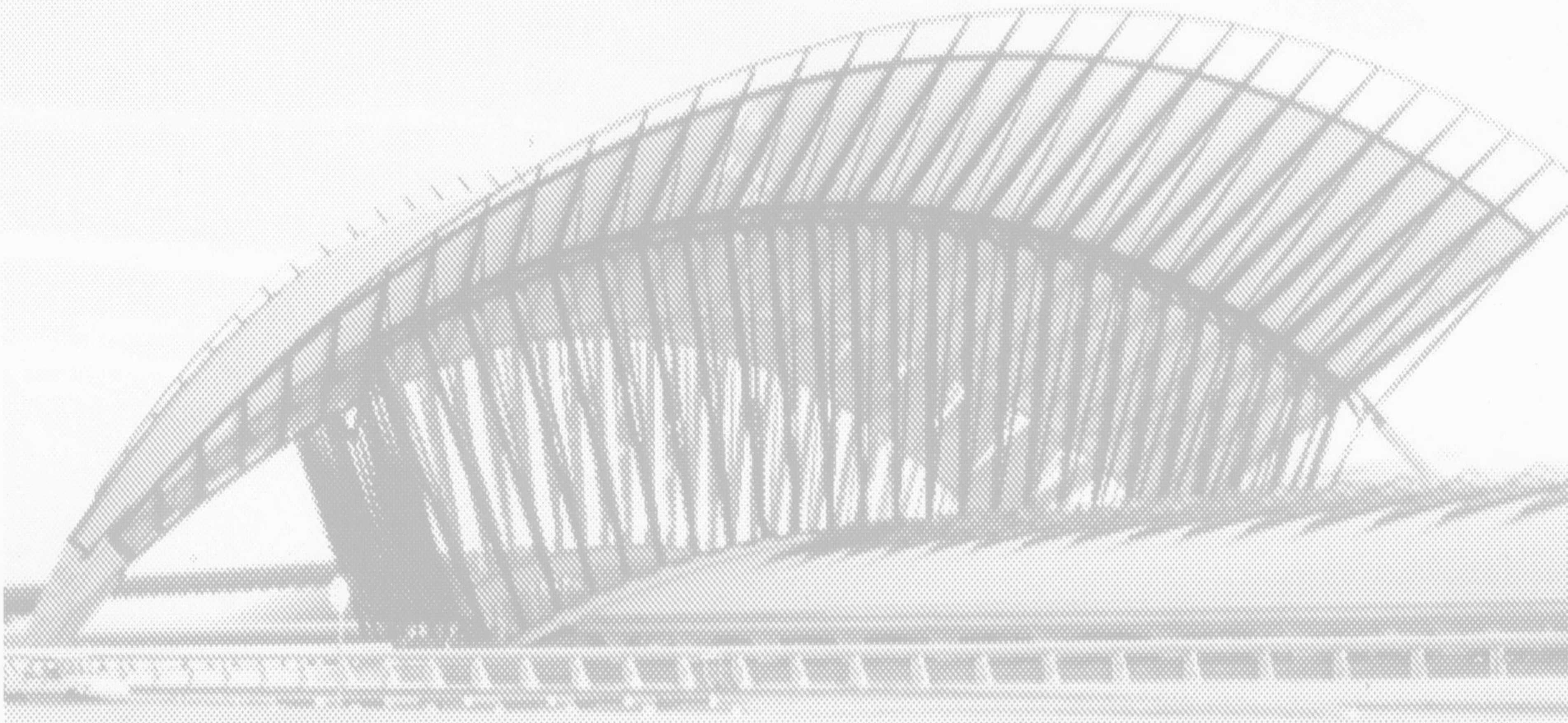

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3-D View


File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3-D View


File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3D View

Sony Center, Potzdamer Platz, Berlin, 2000, Helmut Jahn Arch., Ove Arup USA Struct. Eng


24 9 2001



FRAMES/ ARCHES

FRAMES are **flexural-axial systems** in contrast to **hinged trusses**, which are axial systems, and **beams**, which are flexural systems. Flexural-axial systems are identified by **beam-column behavior** that includes the effects of biaxial bending, torsion, axial deformation, and biaxial shear deformations.

Here, **two-dimensional skeleton structures composed of linear elements** are briefly investigated. The most common group of planar structure systems includes

post-beam structures,
bent and folded beams,
rectangular portal frames,
cantilever frames,
braced frames,
pitched frames,
arches,
and so on.

@Seismicisolation

These structures may form

short-span or **long-span,**
single-story or **multi-story,**
single-bay or **multi-bay systems.**

They range from low-rise to

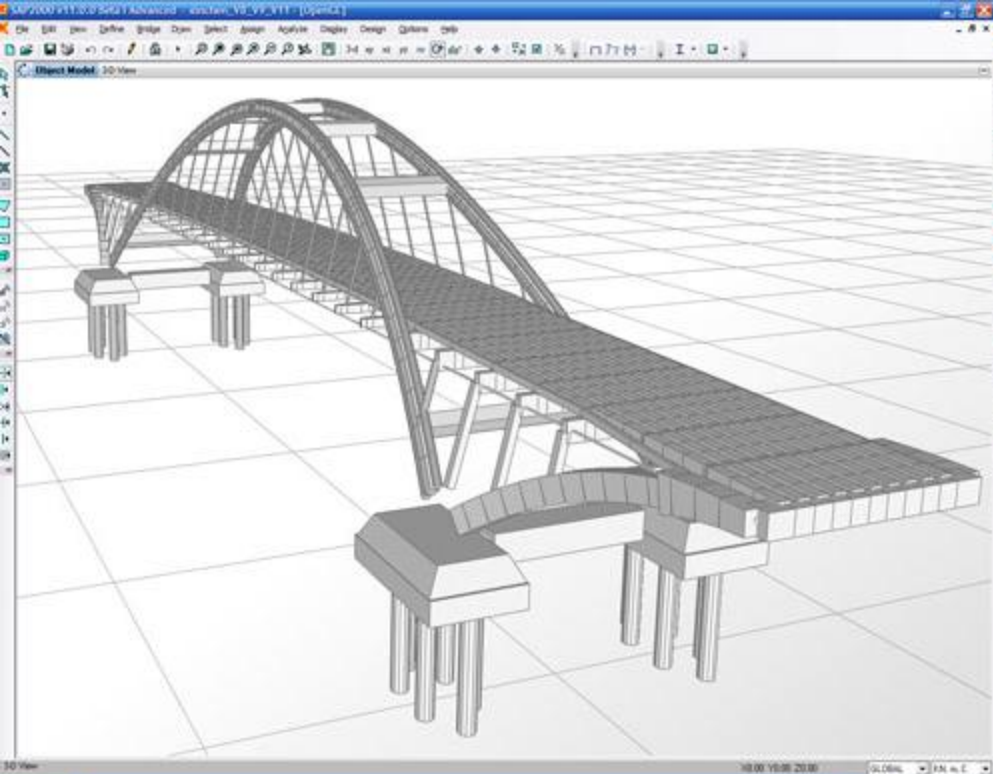
single, open, large volume buildings
cellular massive buildings
skyscrapers.

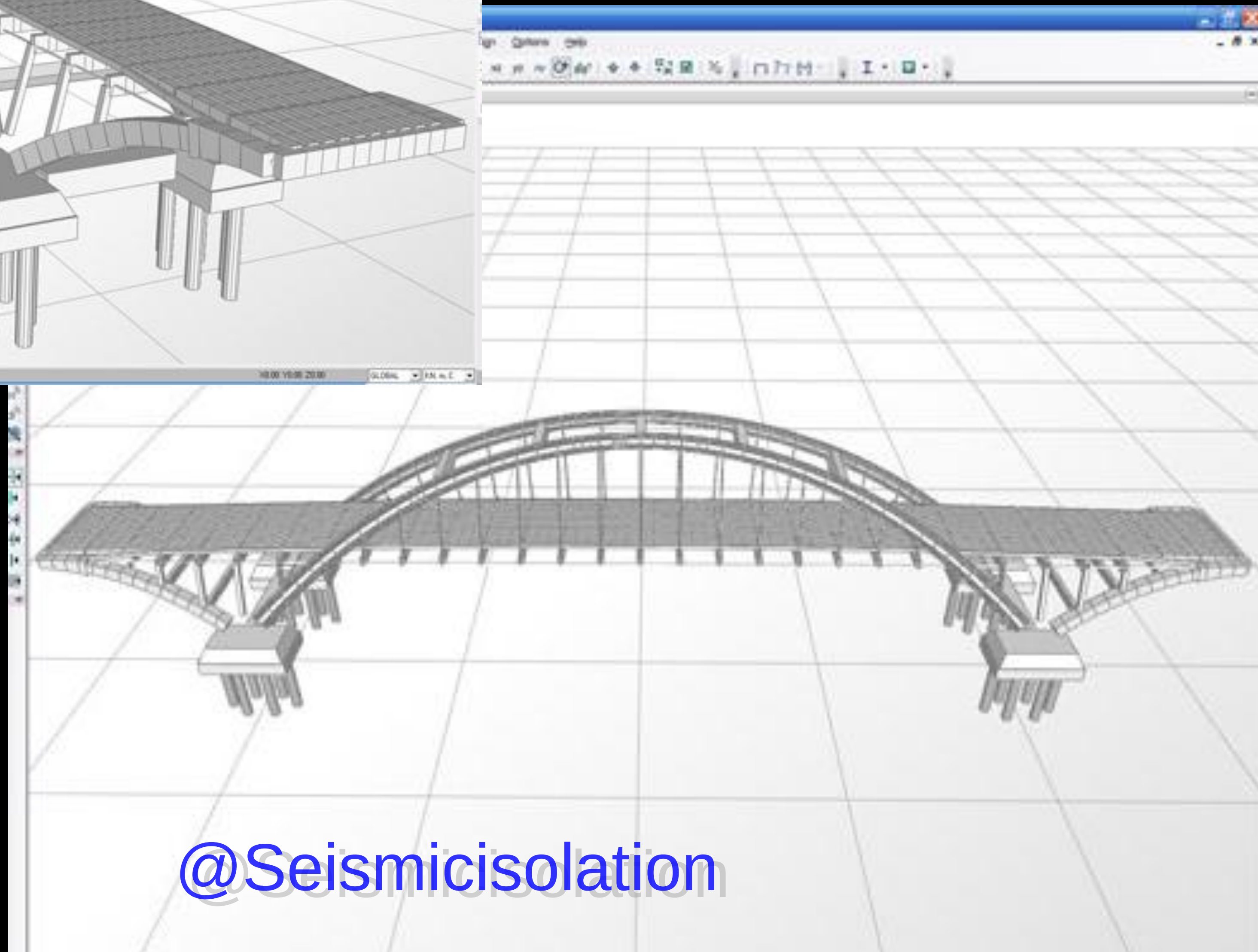
Primary emphasis here is on the investigation of simple, but common single-story enclosure systems to develop a feeling for the behavior of structures under gravity and lateral loads. Investigated are the

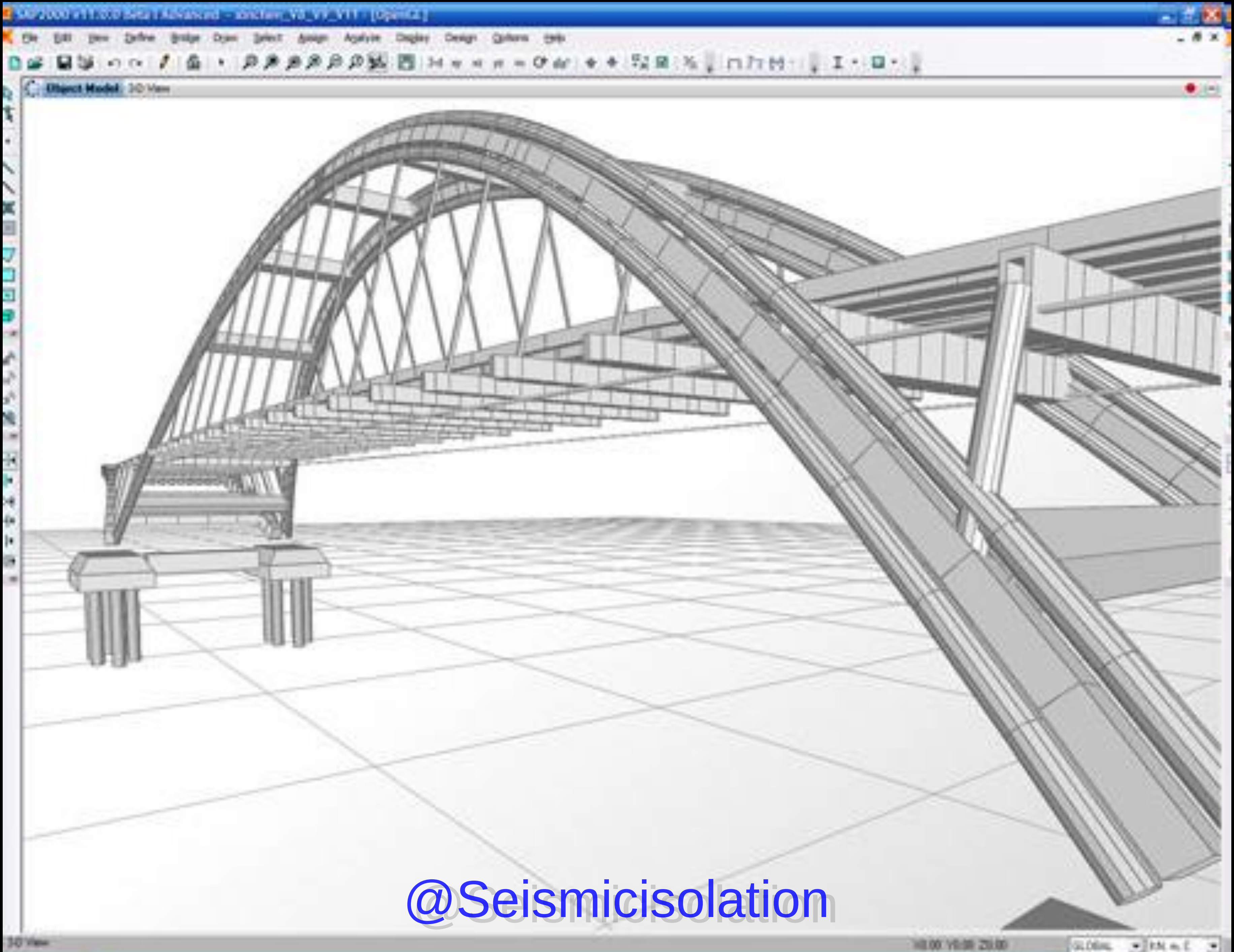
- response and effect of the frame profile on **uniform gravity action** and on **lateral loading**.
- the magnitude of the internal member forces is determined so that the computer results can be checked.

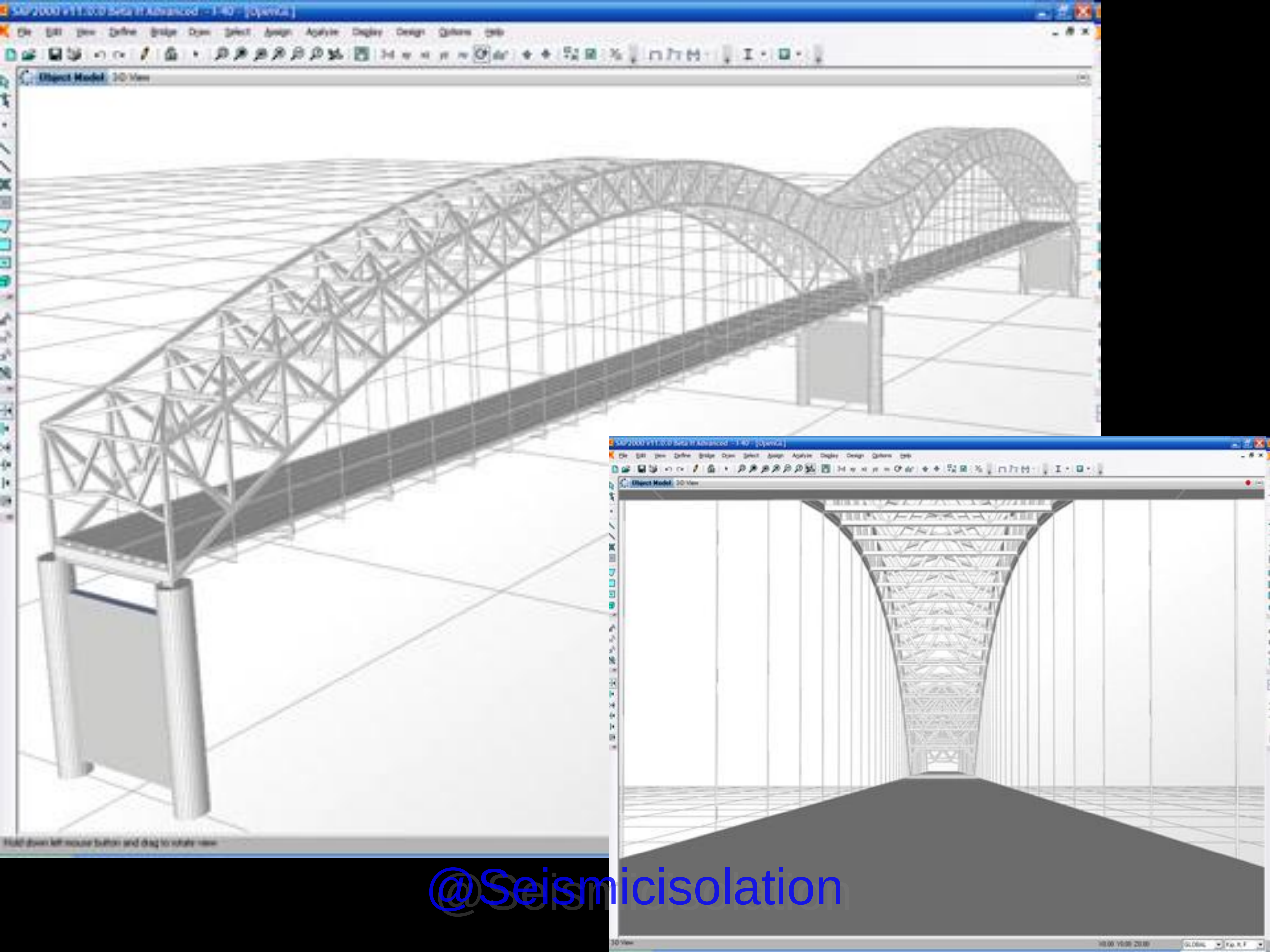
Glass House, New Caanan, 1949, Philip Johnson

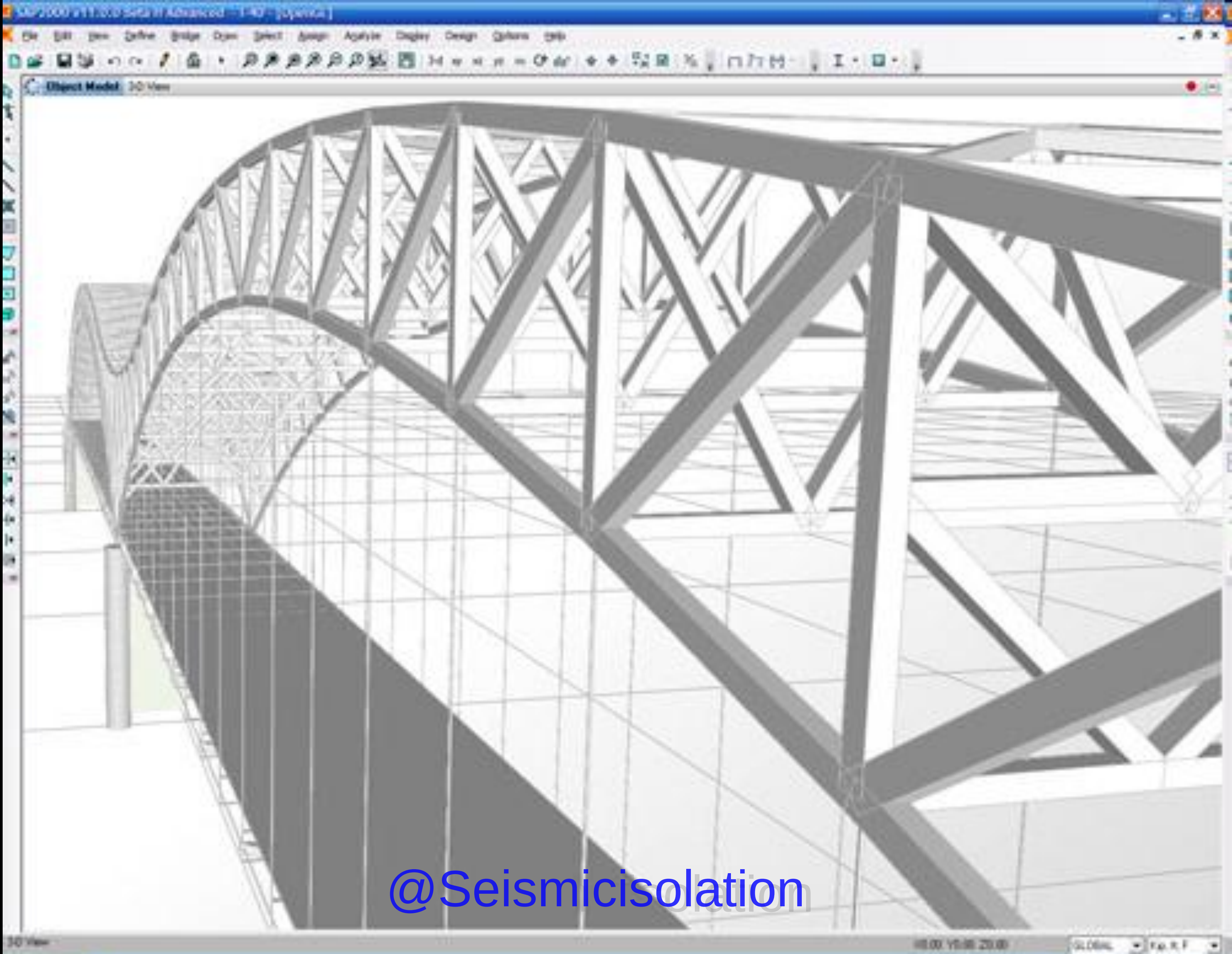
Crown Hall, IIT, Chicago, 1956, Mies van der Rohe

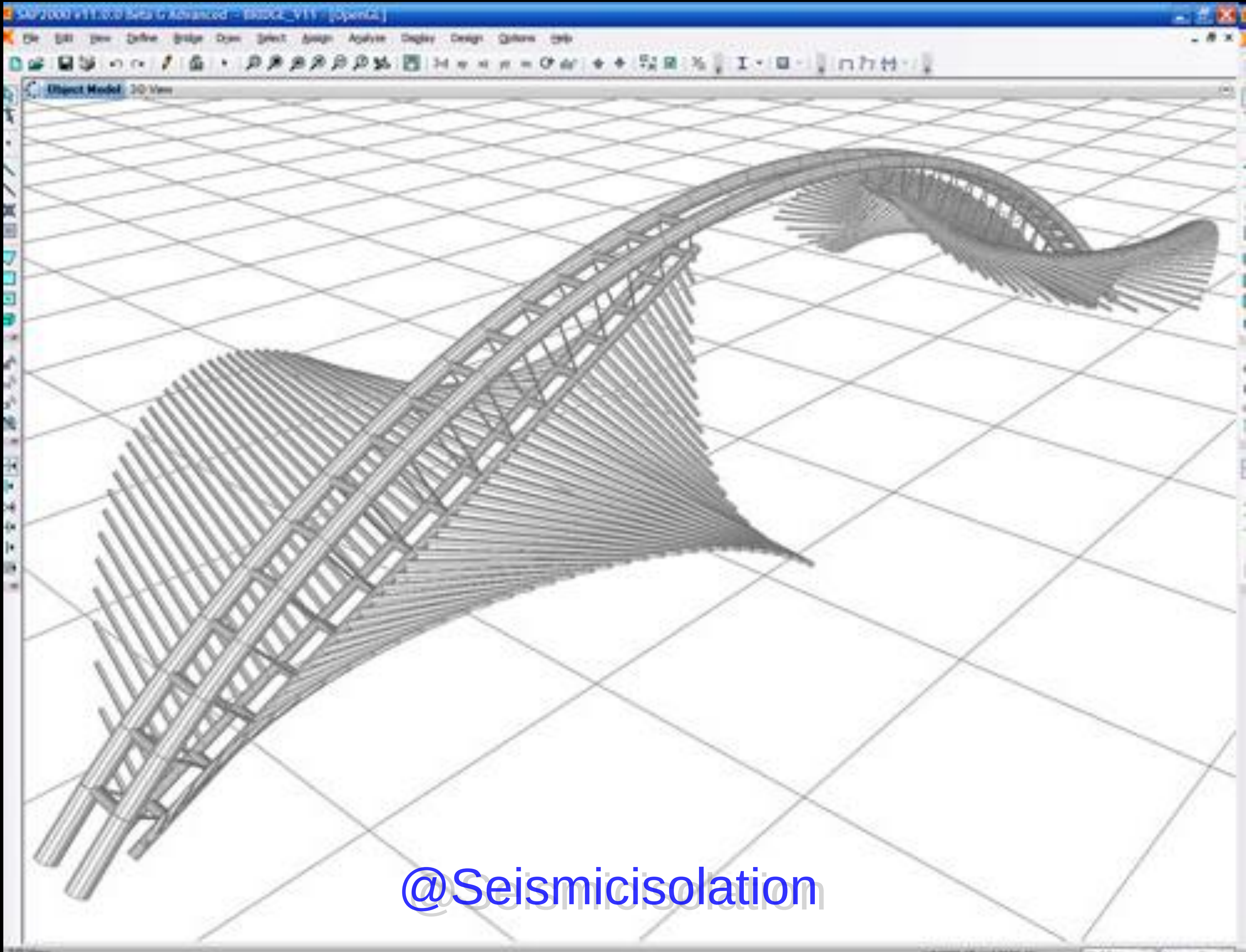


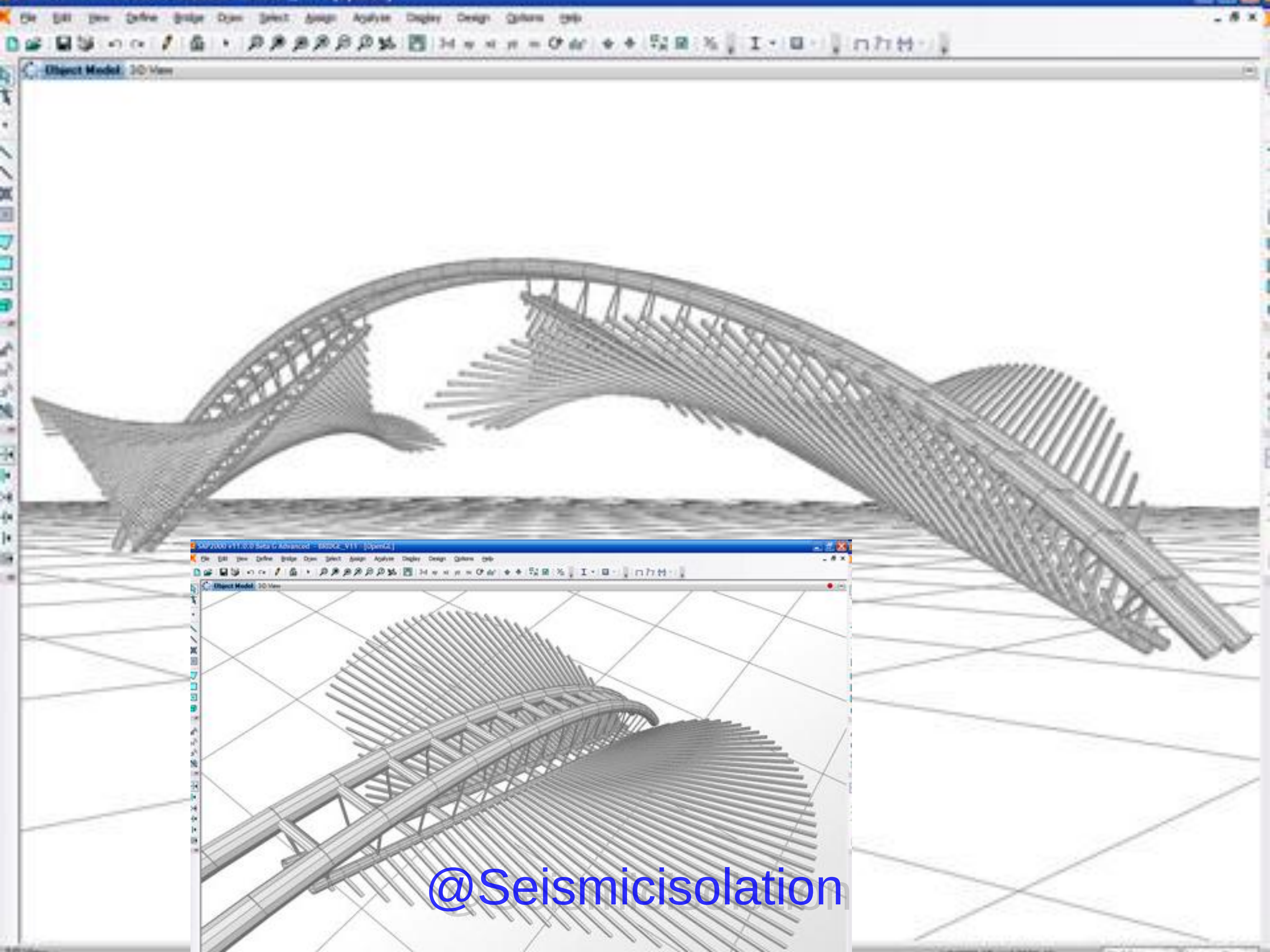


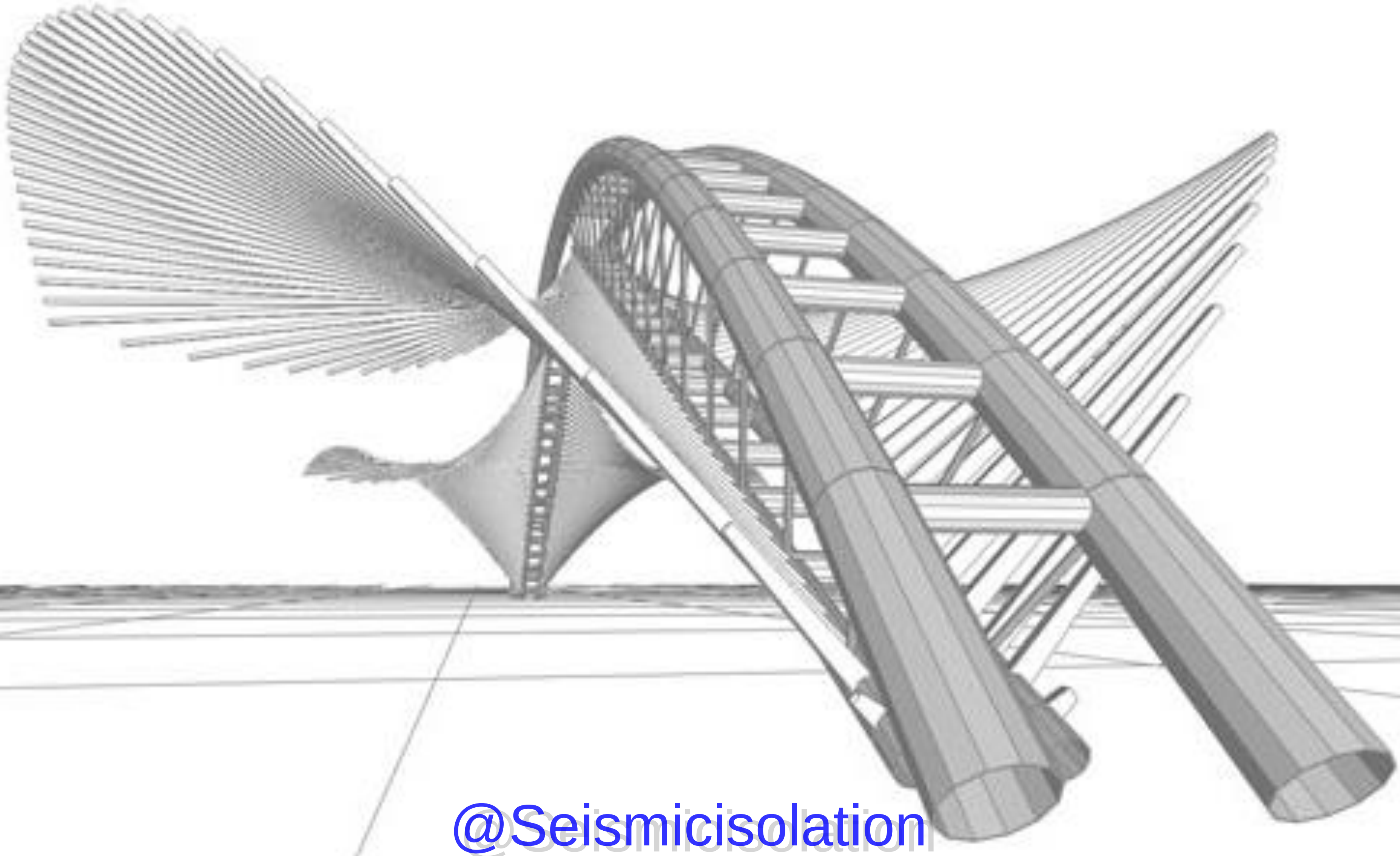







File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model





Berlin Stock Exchange, Berlin, Germany, 1999, Nick Grimshaw

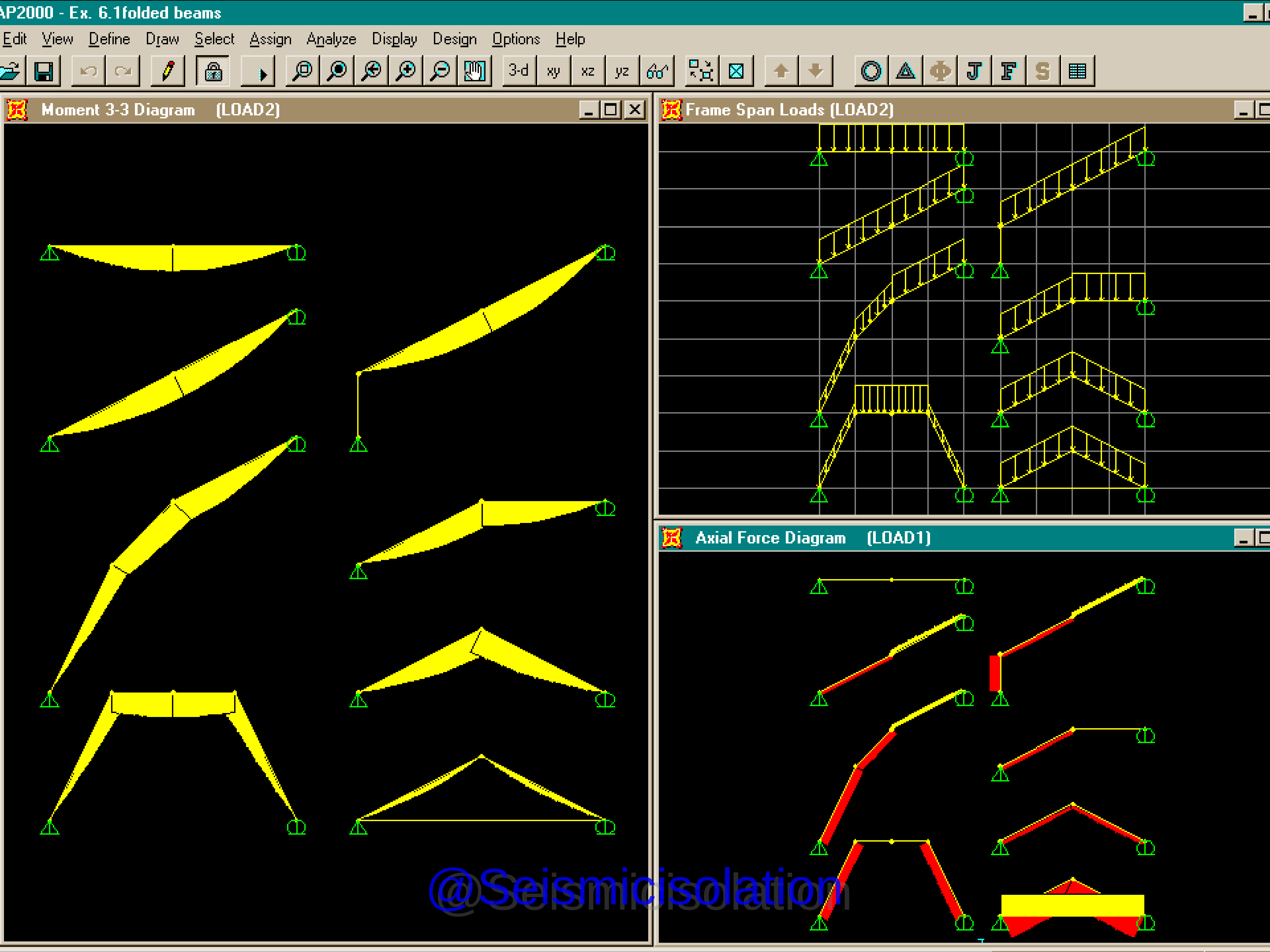

25 9 2001

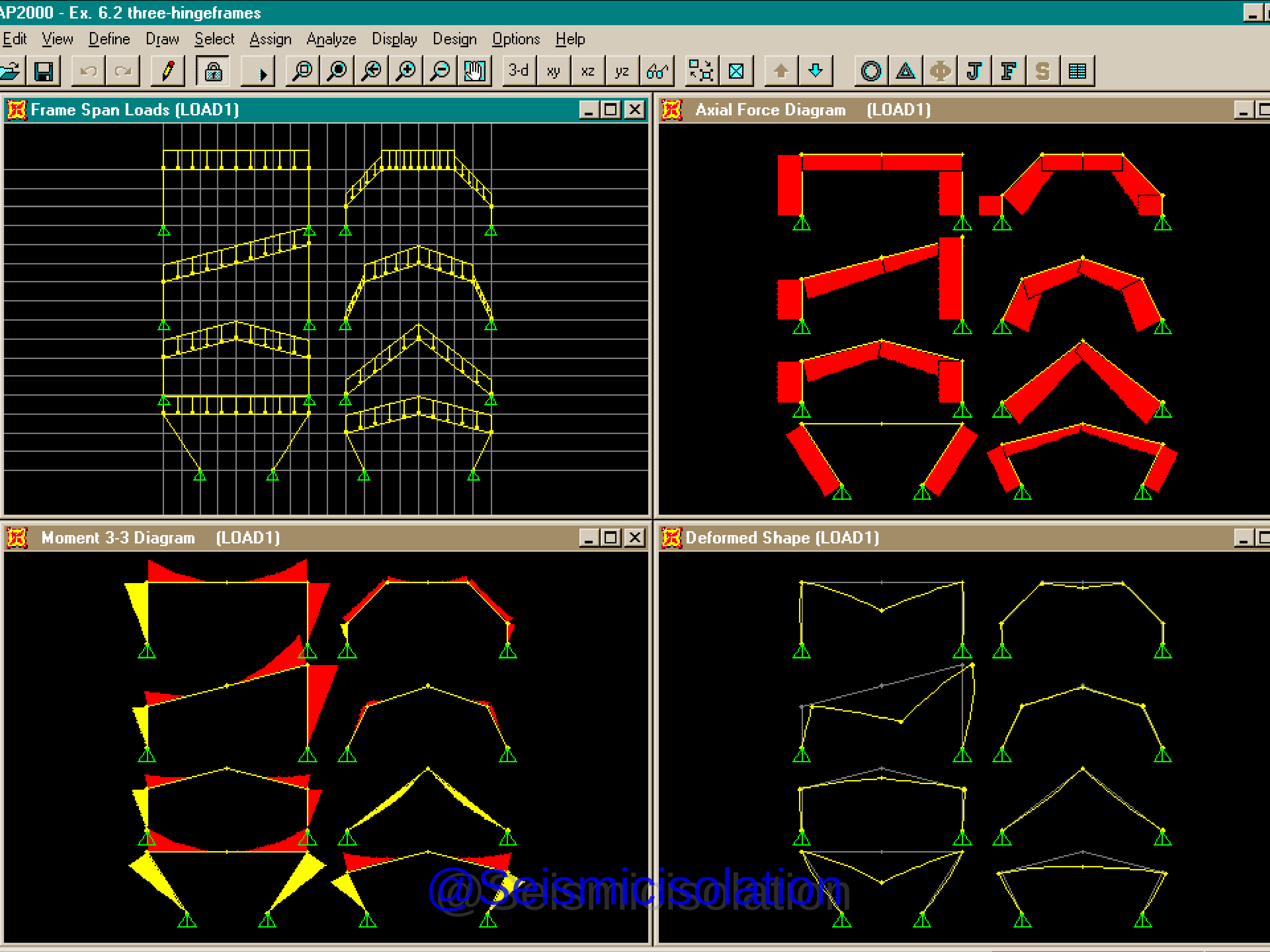

25 9 2001

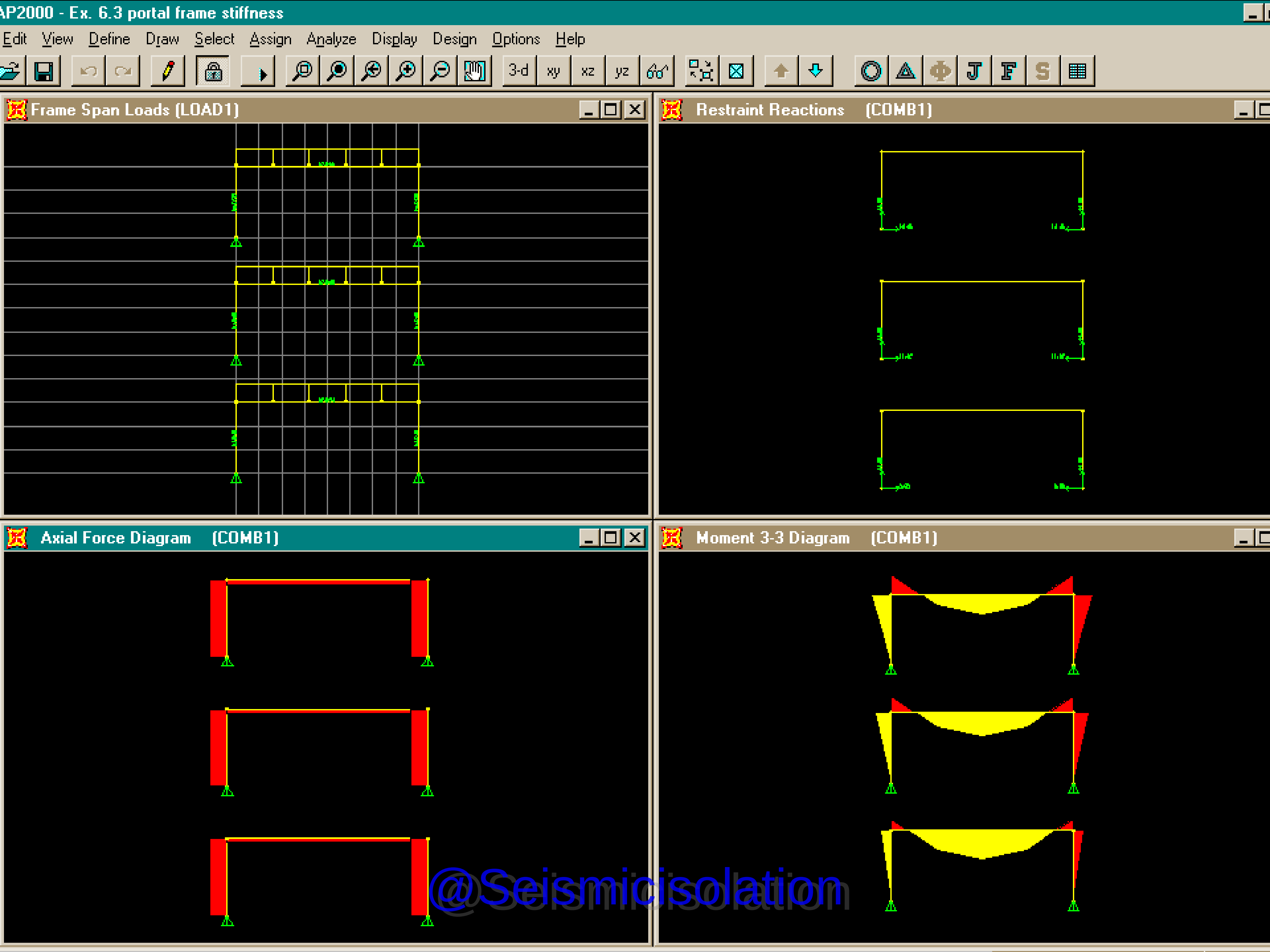

@Seismicisolation

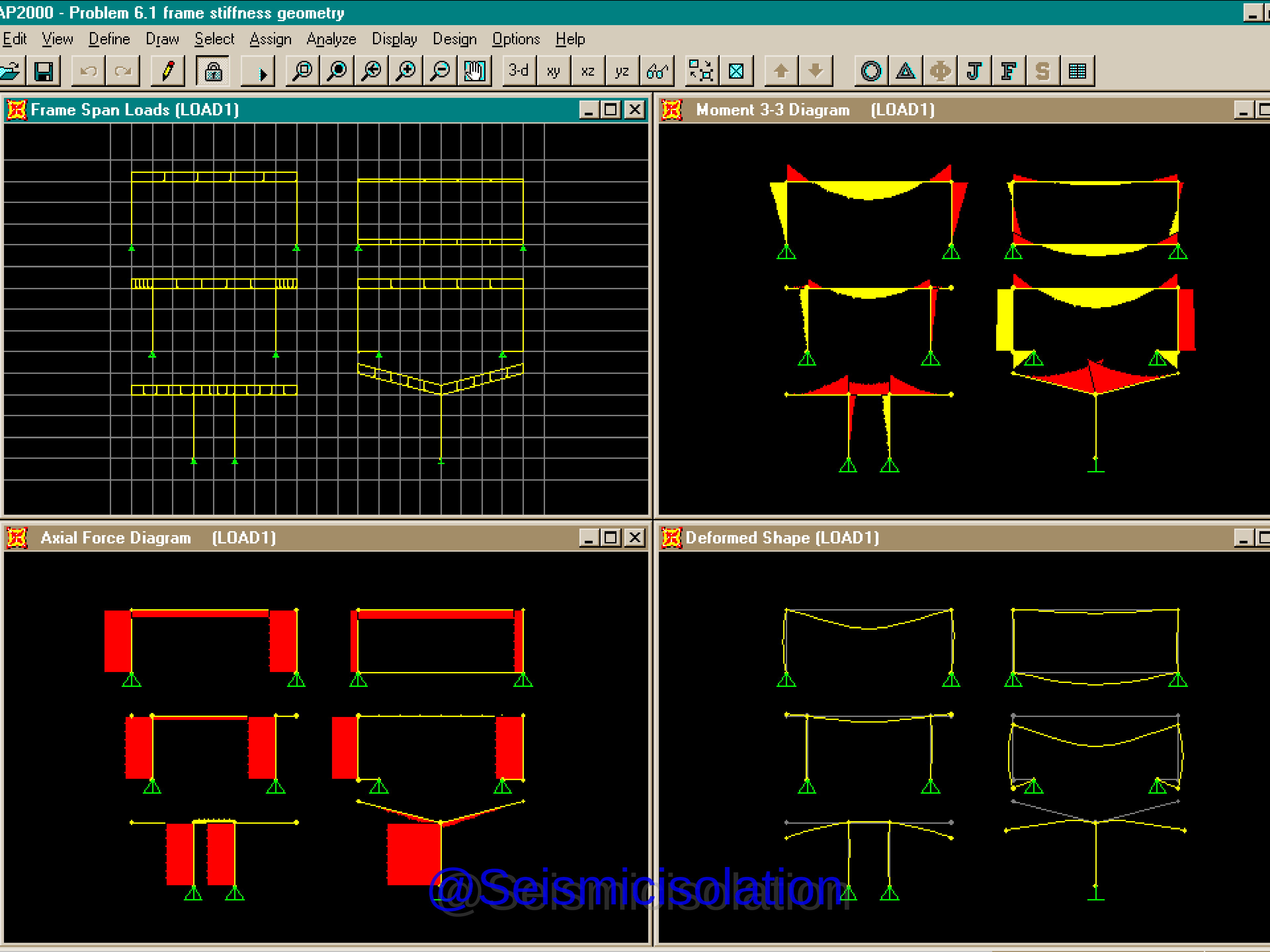

25 9 2001

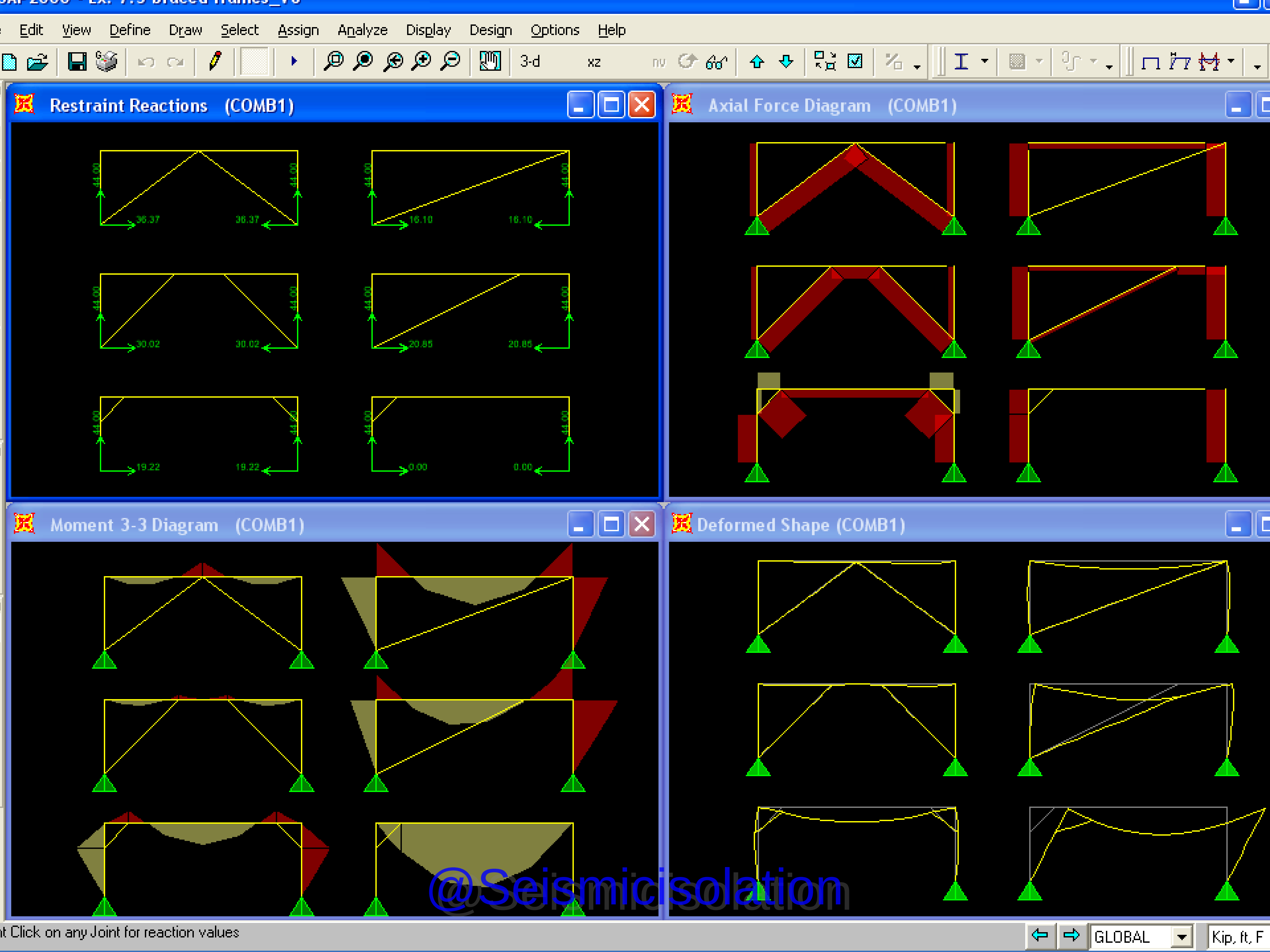
San Diego Library, 1970, William L. Pereira

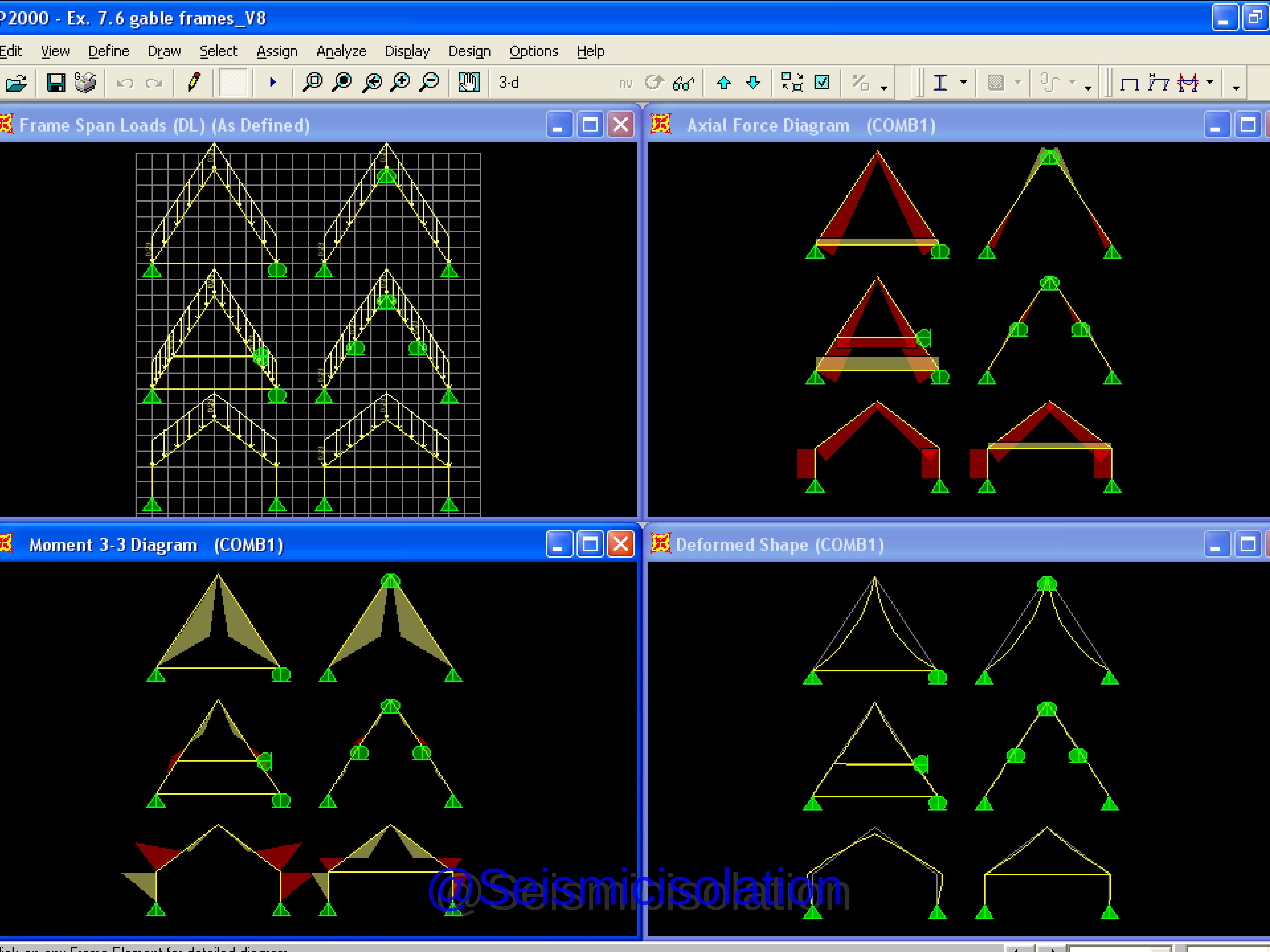

AP2000 - Ex. 6.1folded beams
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Moment 3-3 Diagram (LOAD2)
Frame Span Loads (LOAD2)
Axial Force Diagram (LOAD1)
@Seismicisolation

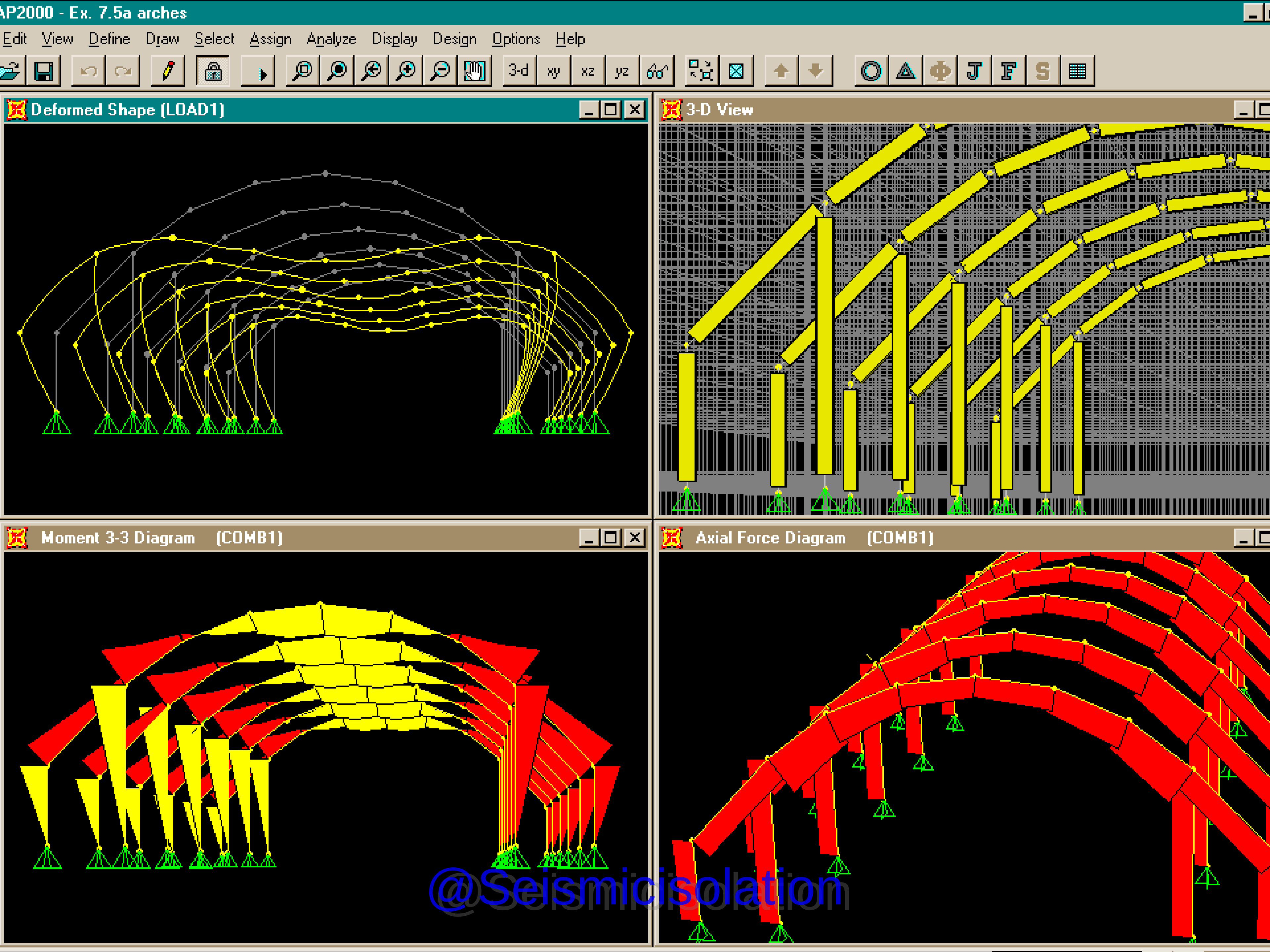

AP2000 - Ex. 6.2 three-hingeframes
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
Axial Force Diagram (LOAD1)
Moment 3-3 Diagram (LOAD1)
Deformed Shape (LOAD1)


AP2000 - Ex. 6.3 portal frame stiffness
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
Restraint Reactions (COMB1)
Axial Force Diagram (COMB1)
Moment 3-3 Diagram (COMB1)
@Seismicisolation

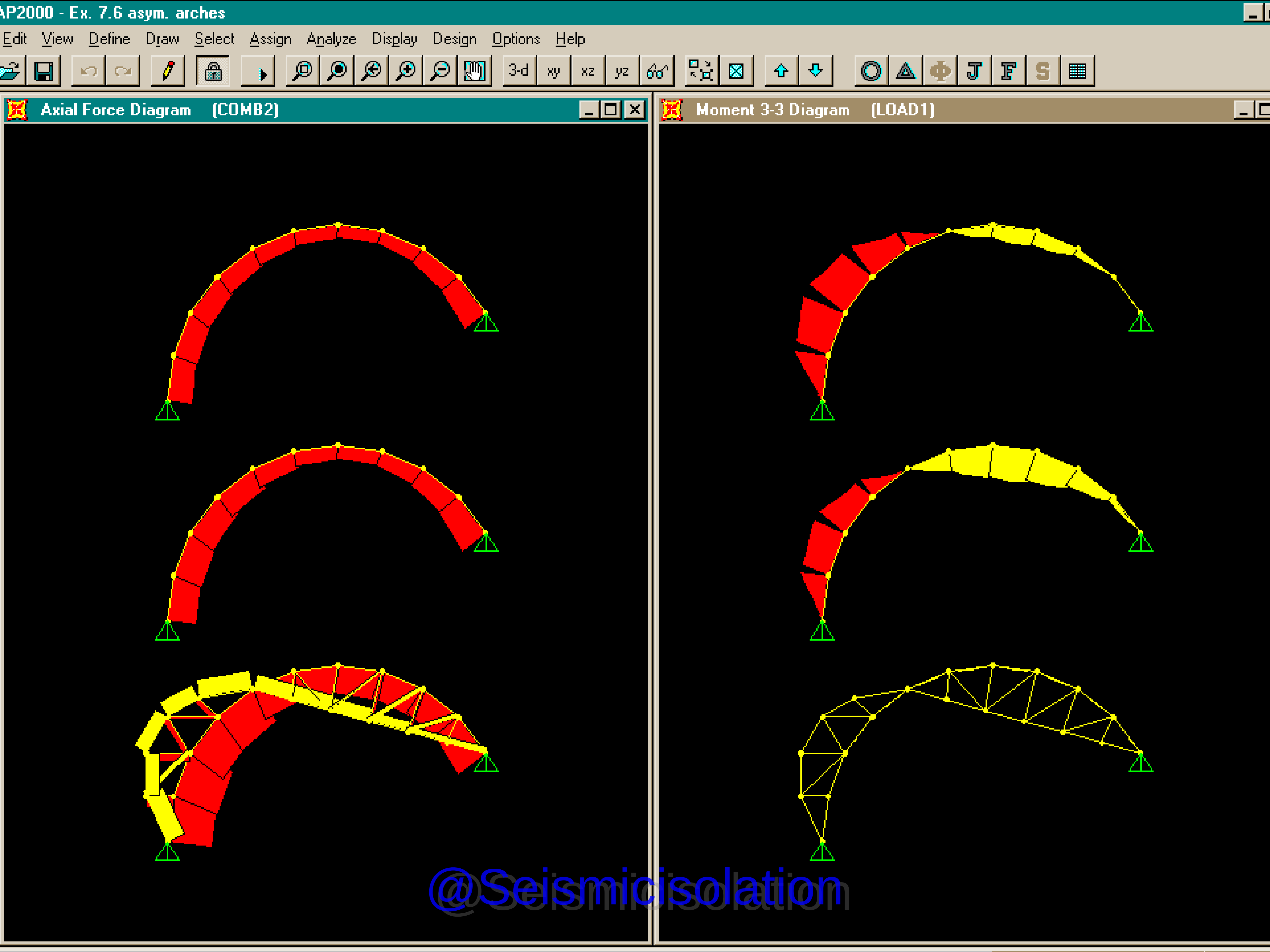

AP2000 - Problem 6.1 frame stiffness geometry
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
Moment 3-3 Diagram (LOAD1)
Axial Force Diagram (LOAD1)
Deformed Shape (LOAD1)

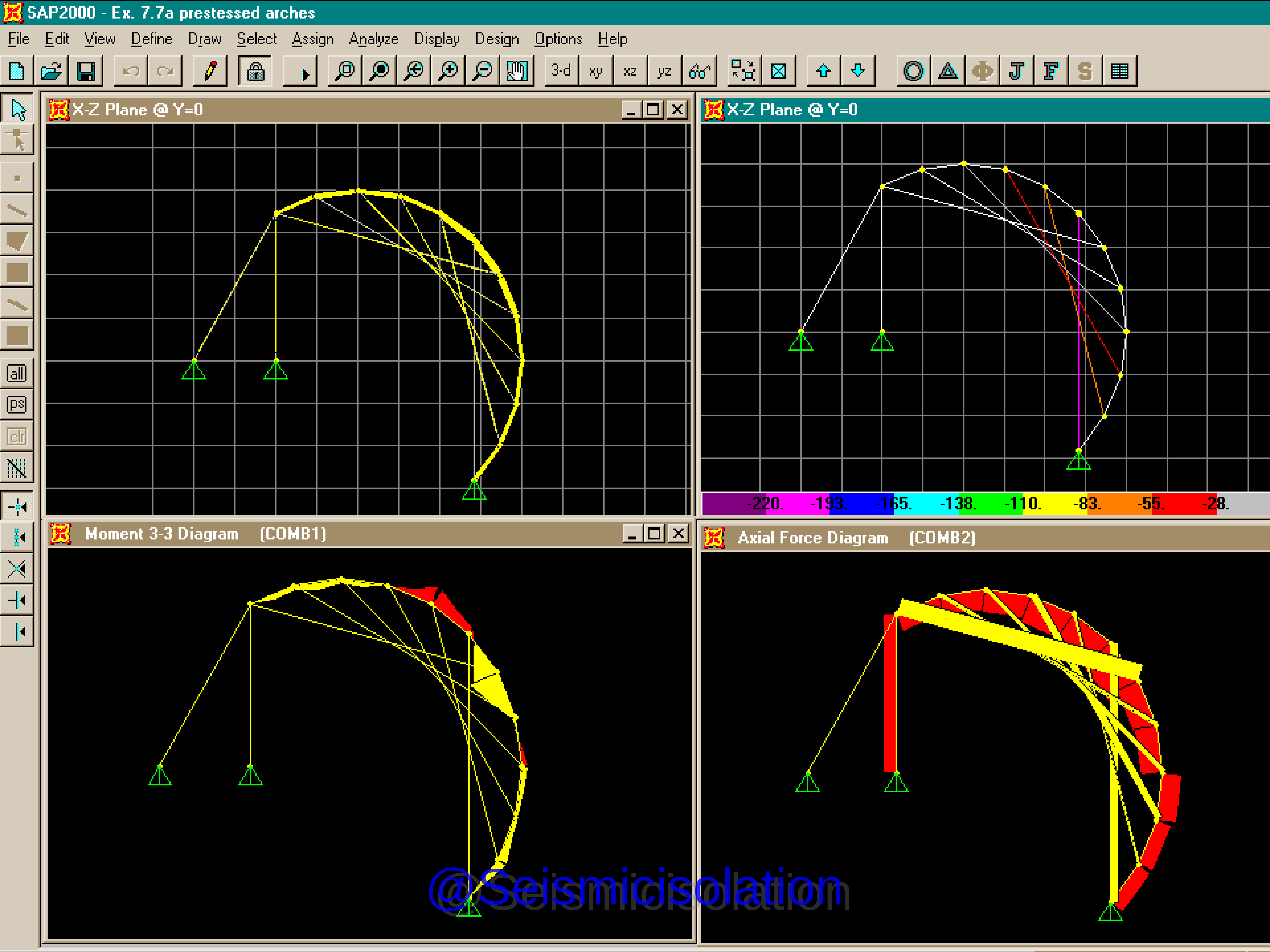

Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
xz
Restraint Reactions (COMB1)
44.00
36.37
36.37
16.10
16.10
30.02
30.02
20.85
20.85
19.22
19.22
0.00
0.00
Axial Force Diagram (COMB1)
Moment 3-3 Diagram (COMB1)
Deformed Shape (COMB1)
Click on any Joint for reaction values
GLOBAL
Kip, ft, F

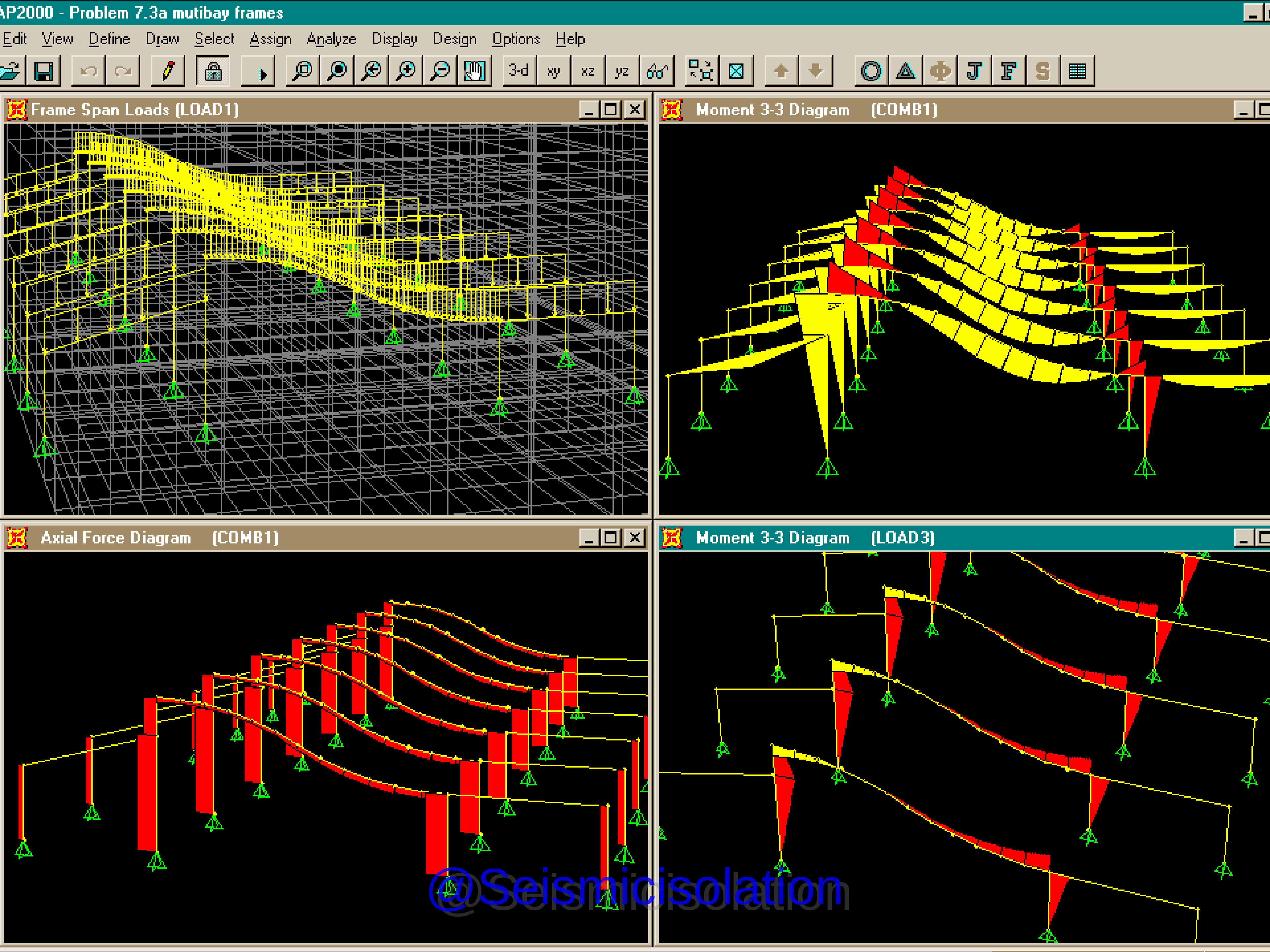

P2000 - Ex. 7.6 gable frames_V8
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d
Frame Span Loads (DL) (As Defined)
Axial Force Diagram (COMB1)
Moment 3-3 Diagram (COMB1)
Deformed Shape (COMB1)
@Seismicisolation

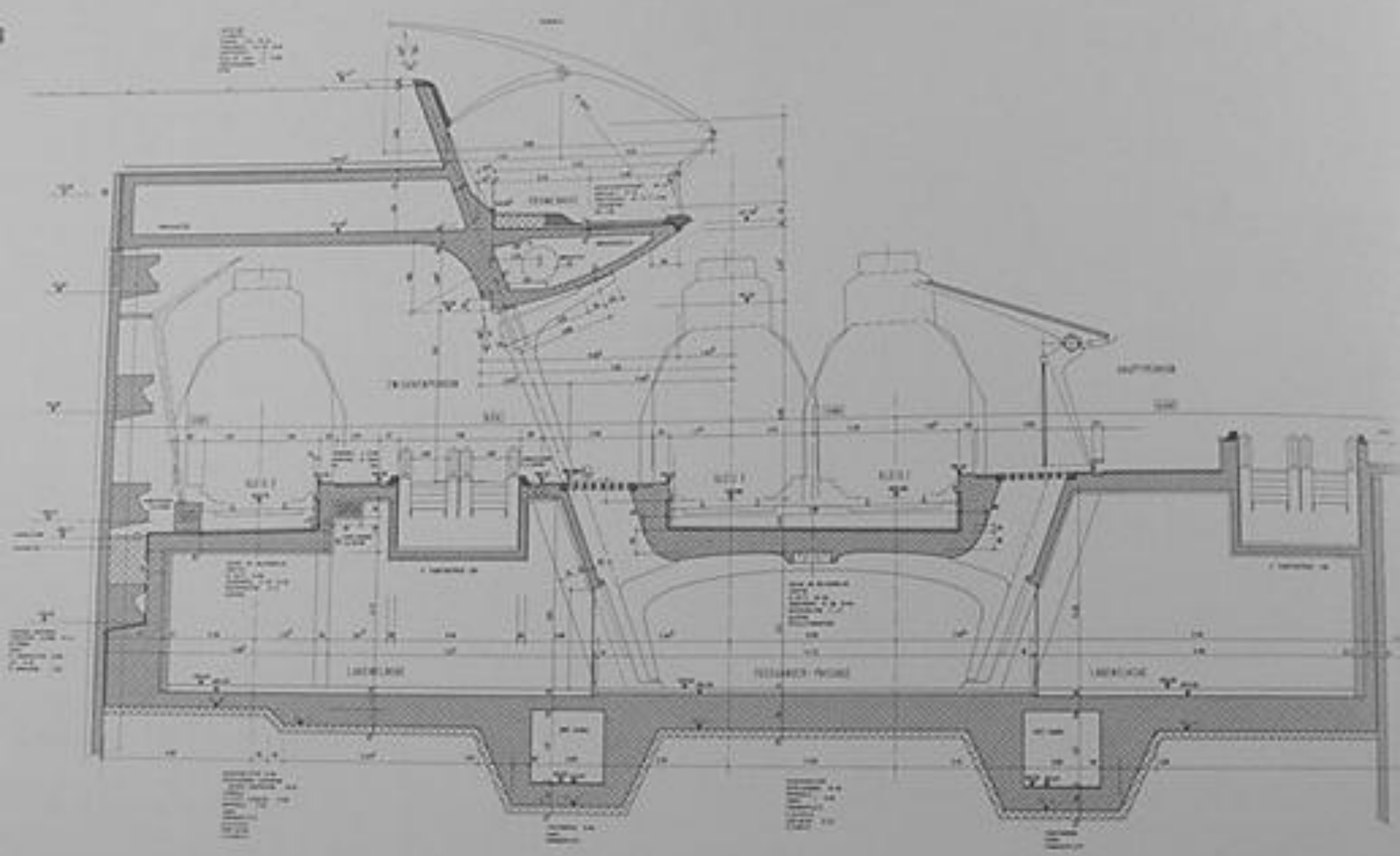

AP2000 - Ex. 7.5a arches
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Deformed Shape (LOAD1)
3-D View
Moment 3-3 Diagram (COMB1)
Axial Force Diagram (COMB1)
@Seismicisolation

Waterloo Terminal, London, 1993, Nicholas Grimshaw + Anthony Hunt

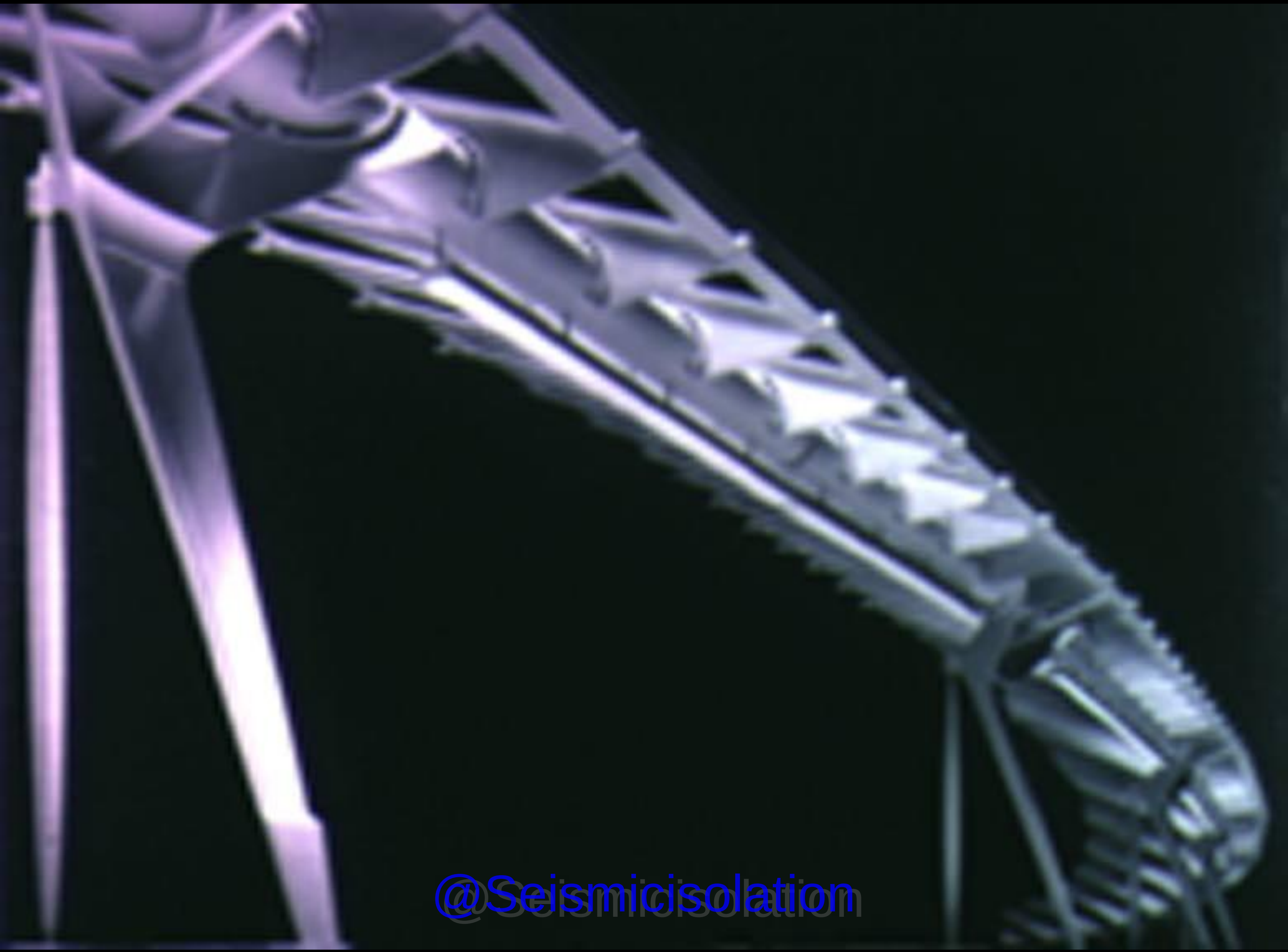

AP2000 - Ex. 7.6 asym. arches
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Axial Force Diagram (COMB2)
Moment 3-3 Diagram (LOAD1)

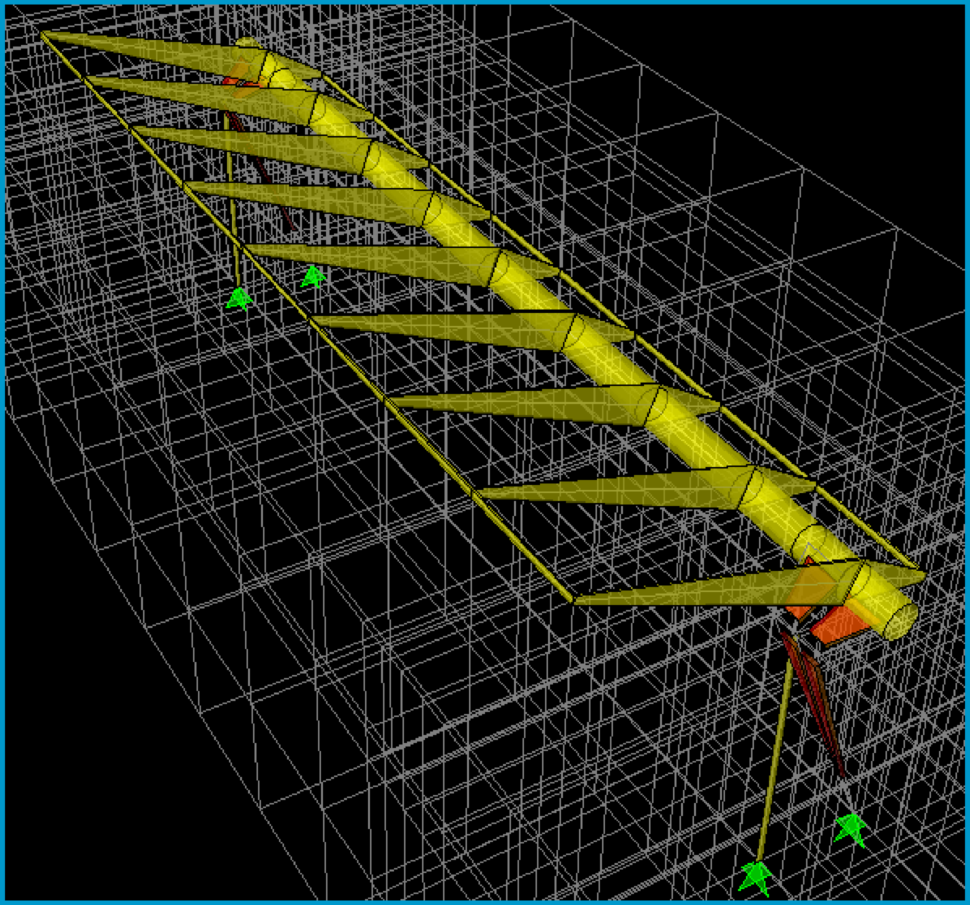

SAP2000 - Ex. 7.7a prestessed arches
File Edit View Define Draw Select Assign Analyze Display Design Options Help
X-Z Plane @ Y=0
X-Z Plane @ Y=0
-220. -193. -165. -138. -110. -83. -55. -28.
Moment 3-3 Diagram (COMB1)
Axial Force Diagram (COMB2)
@Seismicisolation

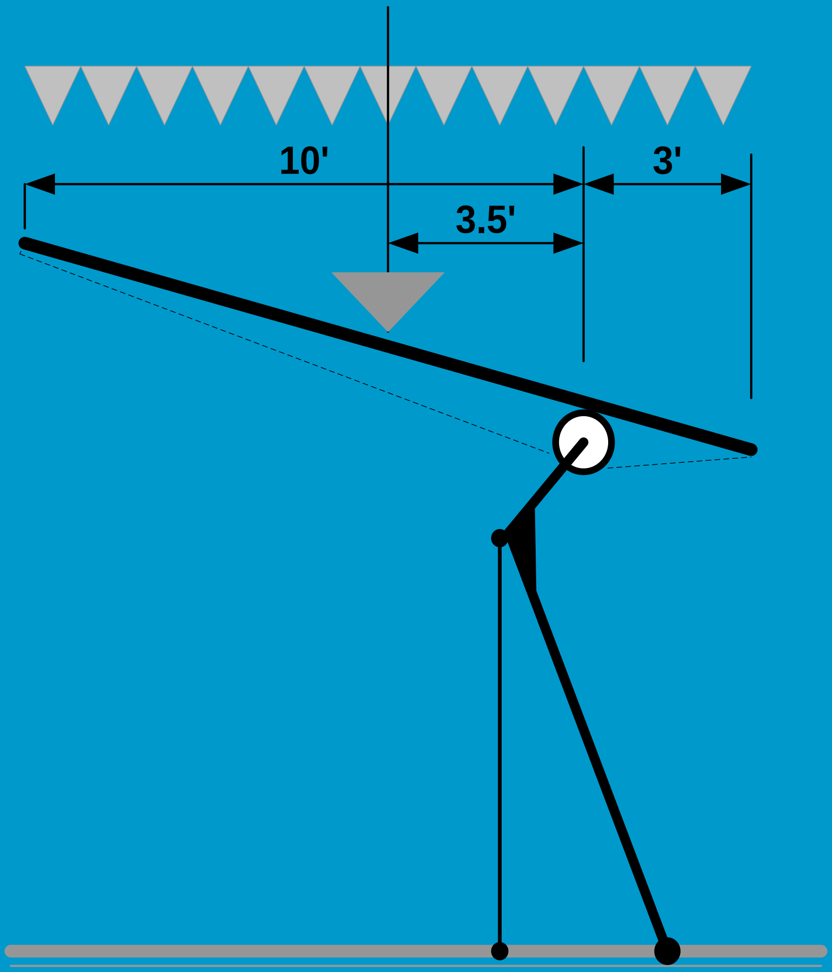

AP2000 - Problem 7.3a mutibay frames
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Frame Span Loads (LOAD1)
Moment 3-3 Diagram (COMB1)
Axial Force Diagram (COMB1)
Moment 3-3 Diagram (LOAD3)
@Seismicisolation



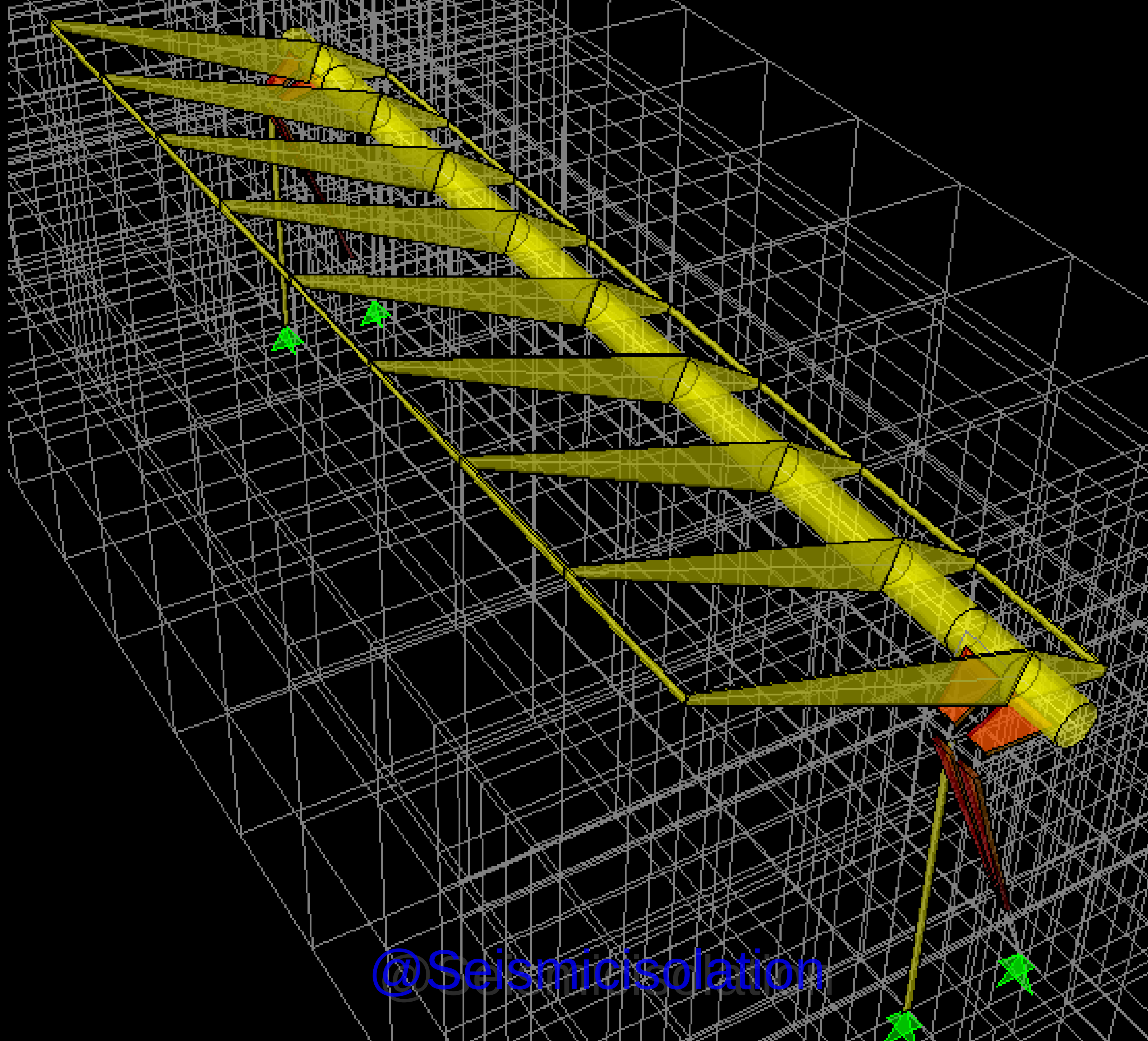
Stadelhofen Railway Station, Zurich, 1990, Santiago Calatrava, cross section

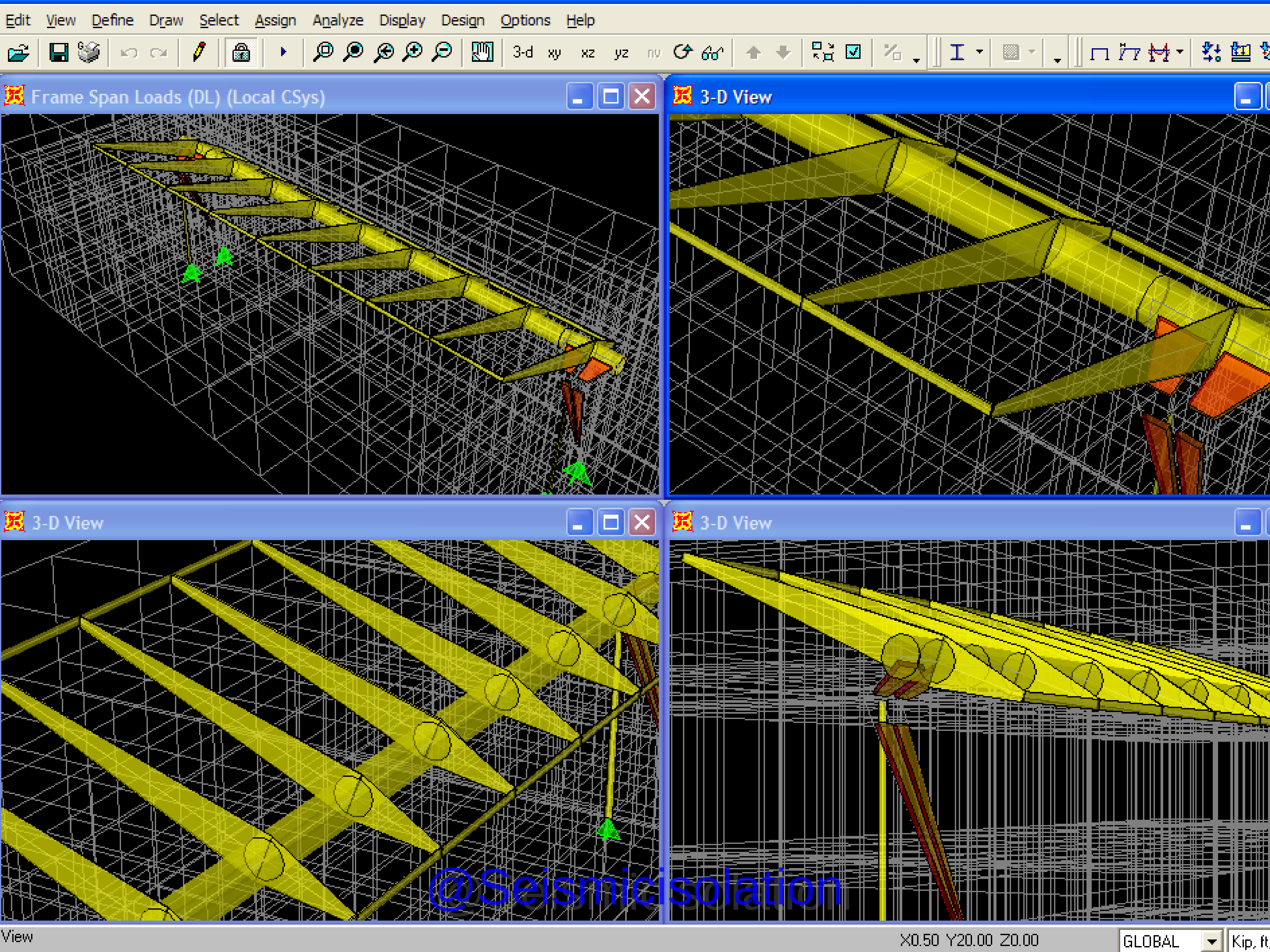


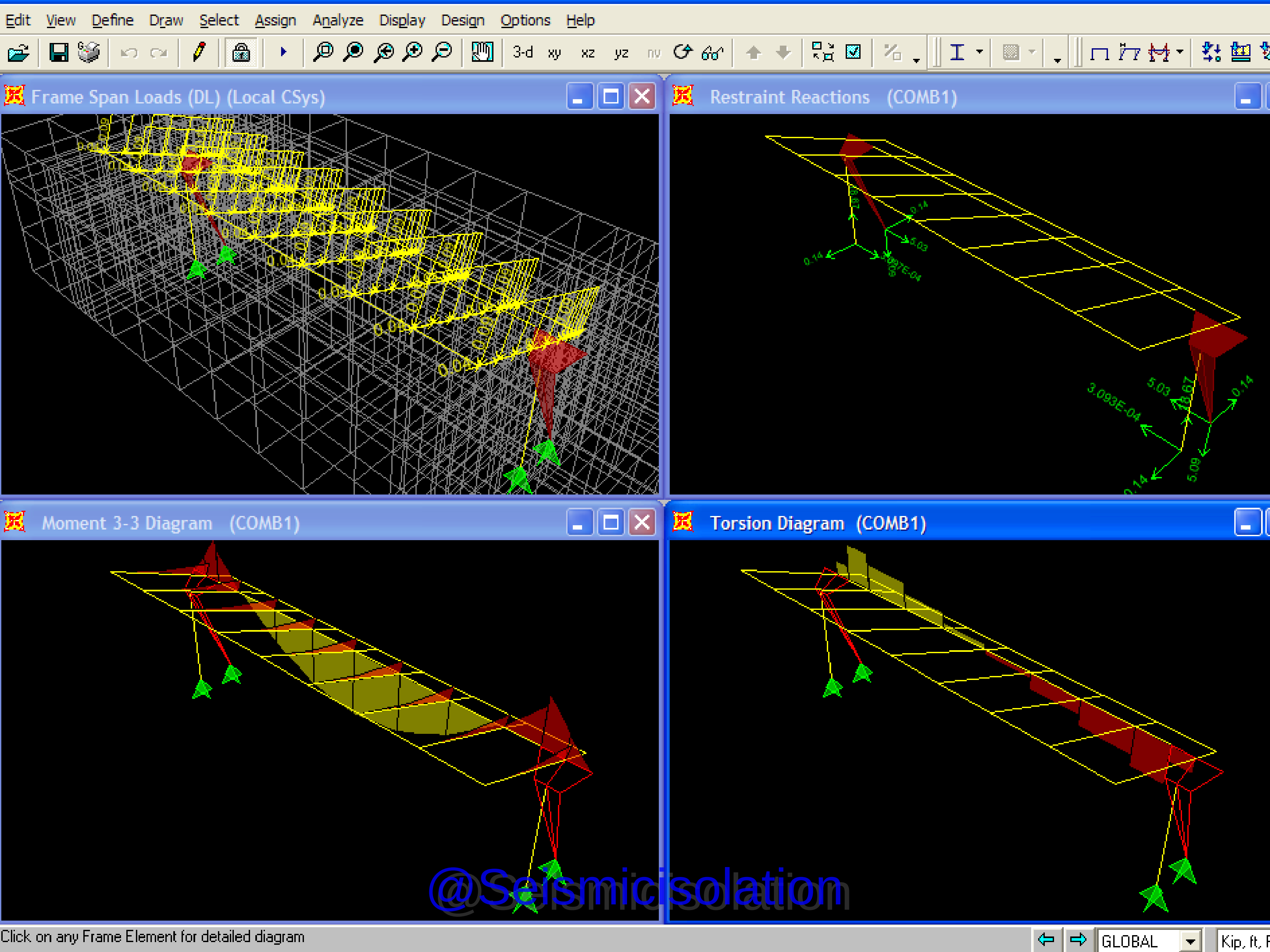


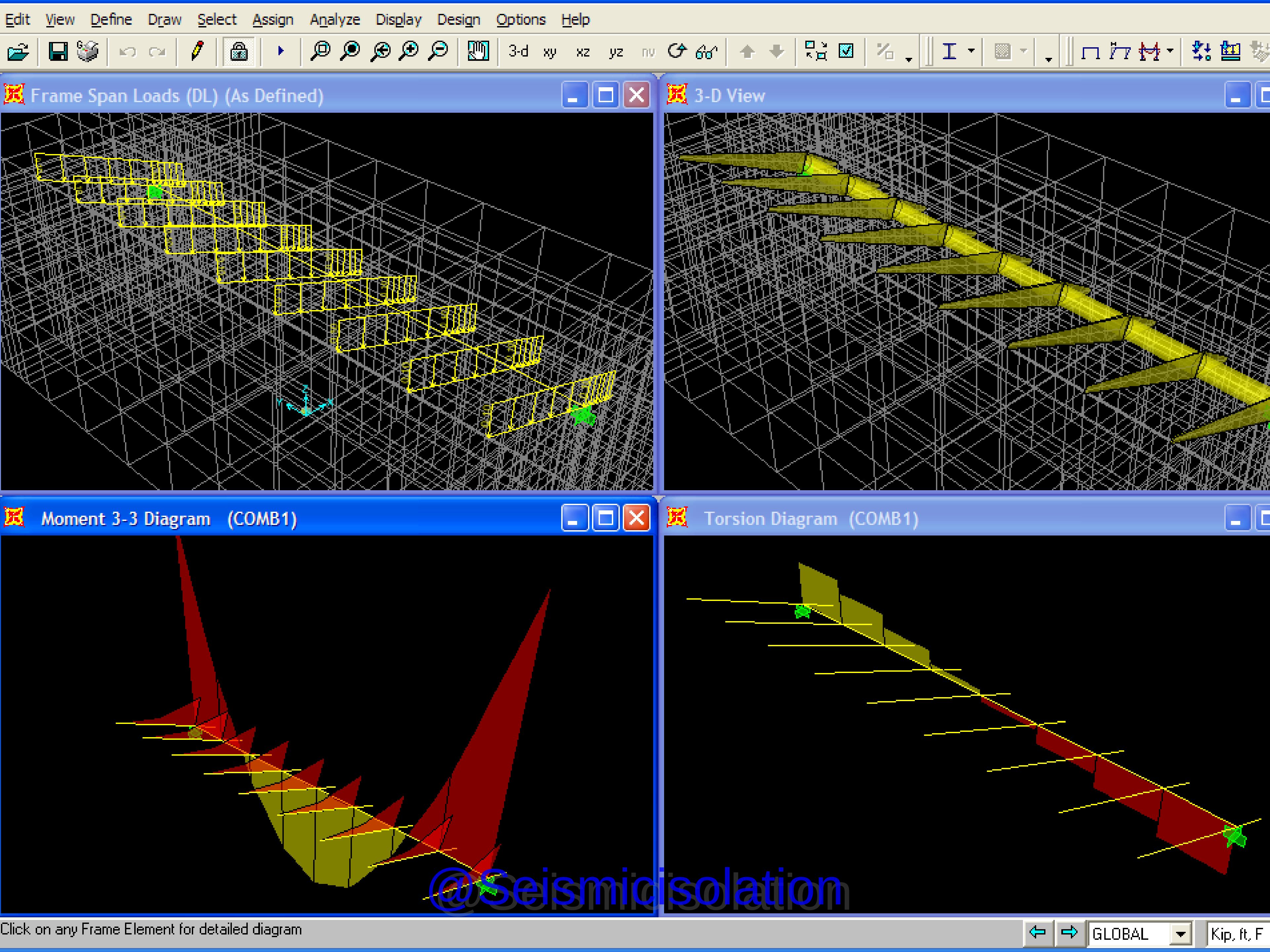


@Seismicisolation




Edit View Define Draw Select Assign Analyze Display Design Options Help
Frame Span Loads (DL) (Local CSys)
3-D View
3-D View
3-D View
@Seismicisolation
View
X0.50 Y20.00 Z0.00
GLOBAL
Kip, ft


Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
Frame Span Loads (DL) (Local CSys)
Restraint Reactions (COMB1)
Moment 3-3 Diagram (COMB1)
Torsion Diagram (COMB1)
3.093E-04
5.03
18.67
0.14
5.09
Click on any Frame Element for detailed diagram
GLOBAL
Kip, ft, F


Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
Frame Span Loads (DL) (As Defined)
3-D View
Moment 3-3 Diagram (COMB1)
Torsion Diagram (COMB1)
@Seismicisolation
Click on any Frame Element for detailed diagram
GLOBAL
Kip, ft, F

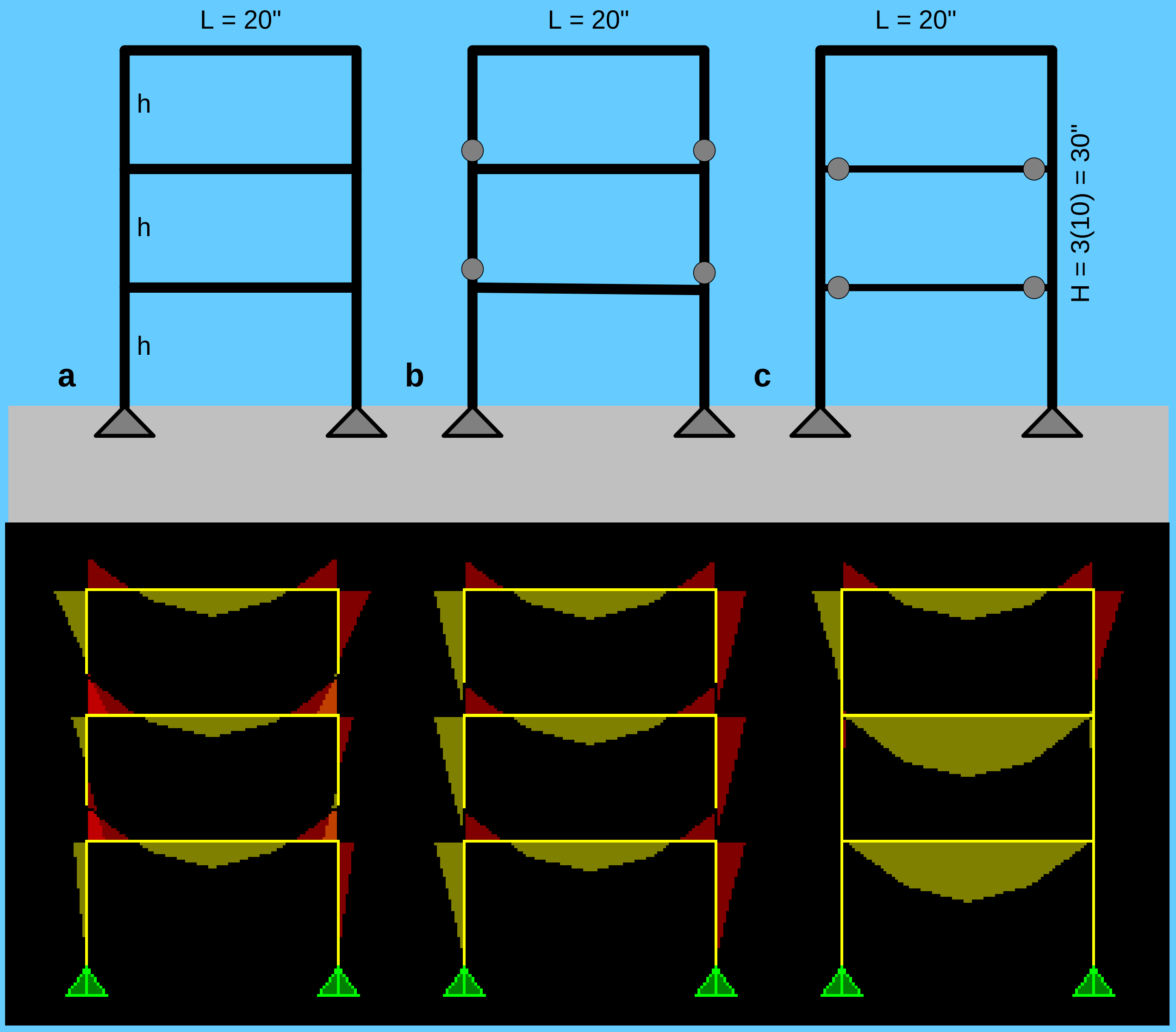
Dresdner Bank, Verwaltungszentrum, Leipzig, 1997, Engel und Zimmermann Arch

Theater Erfurt, 2003, Germany, Joerg Friedrich Arch, glass house

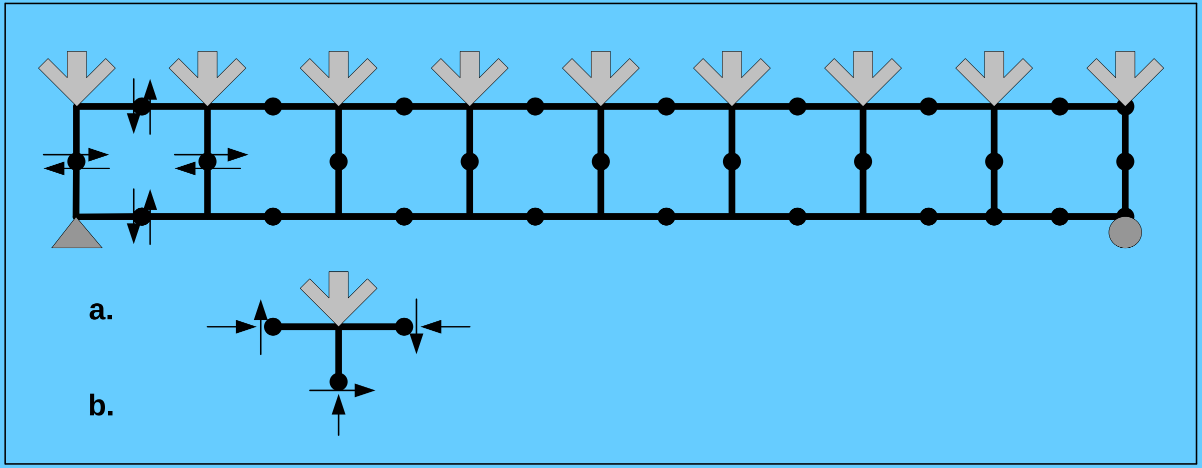
Ningbo downtown, 2002,
Qingyun Ma

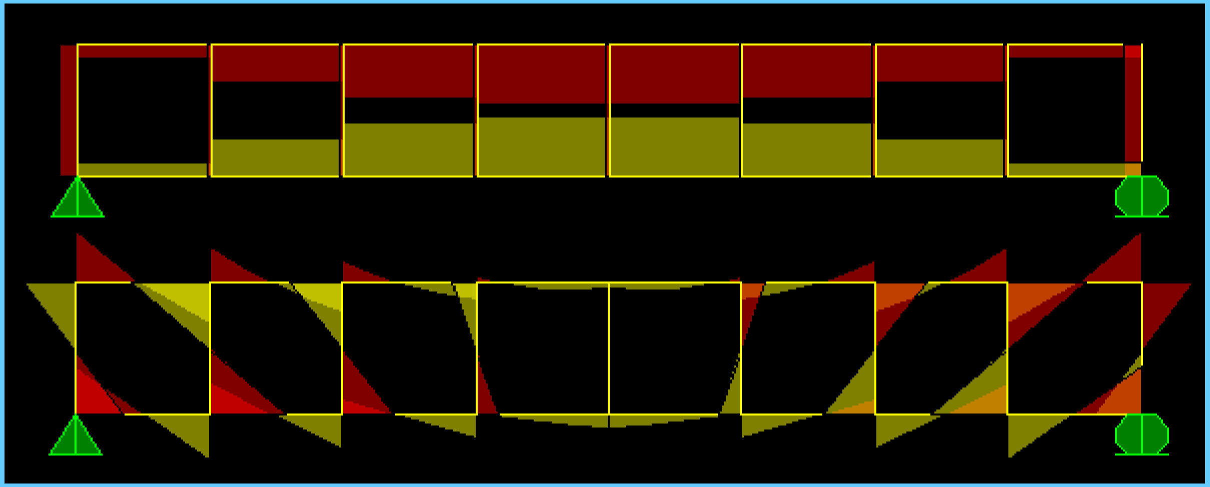
Dokumenta Urbana, Kassel, 1984, Otto Seidle

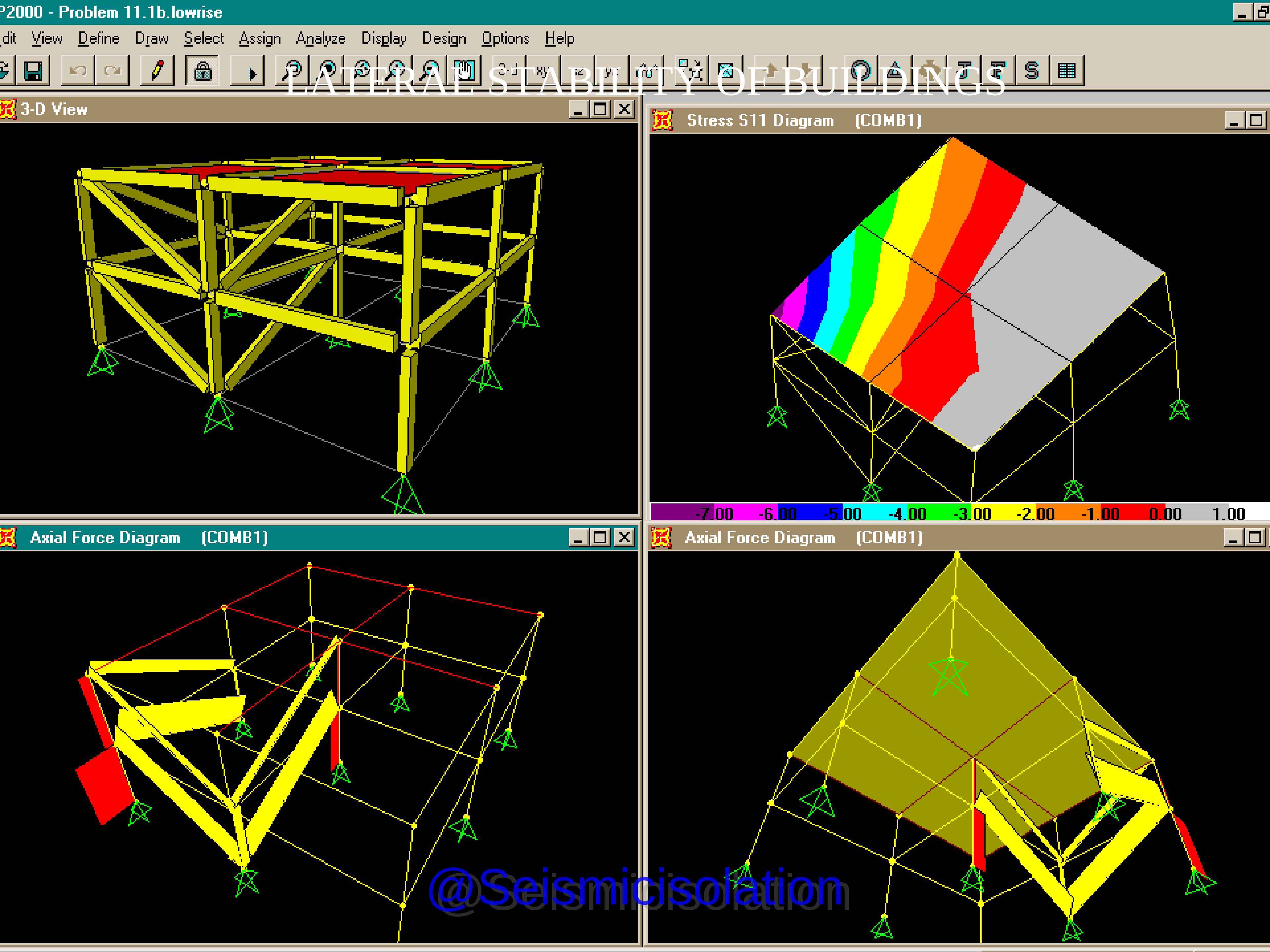
Residential complex, Berlin, Herman Herzberger, 1997

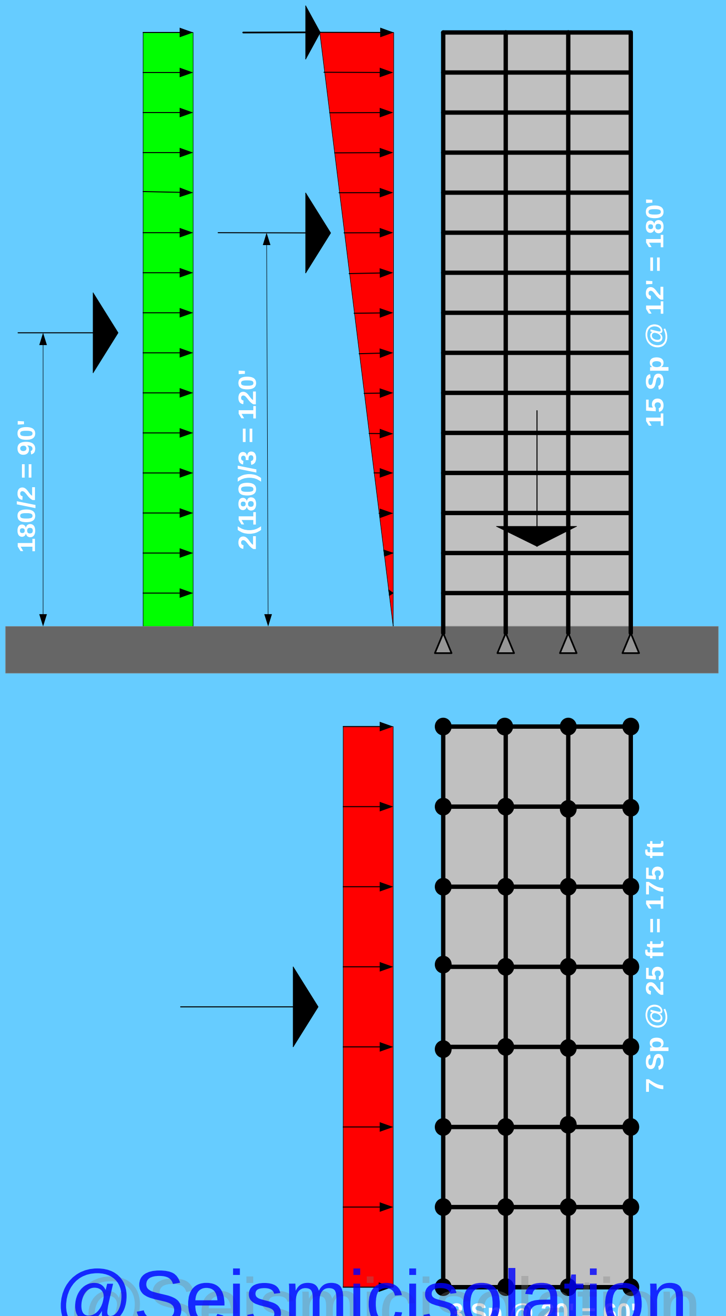


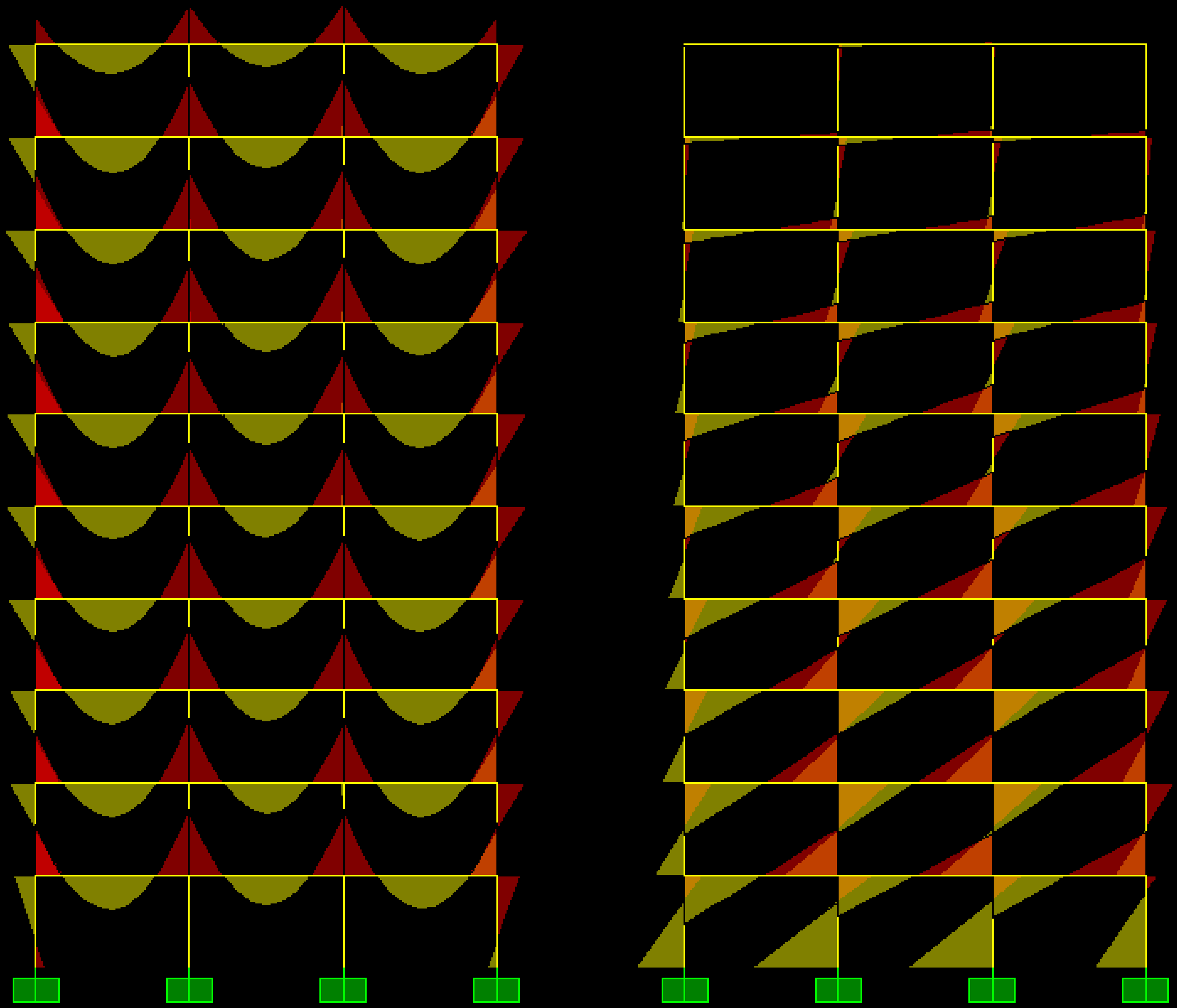
@Seismicisolation


a.
b.

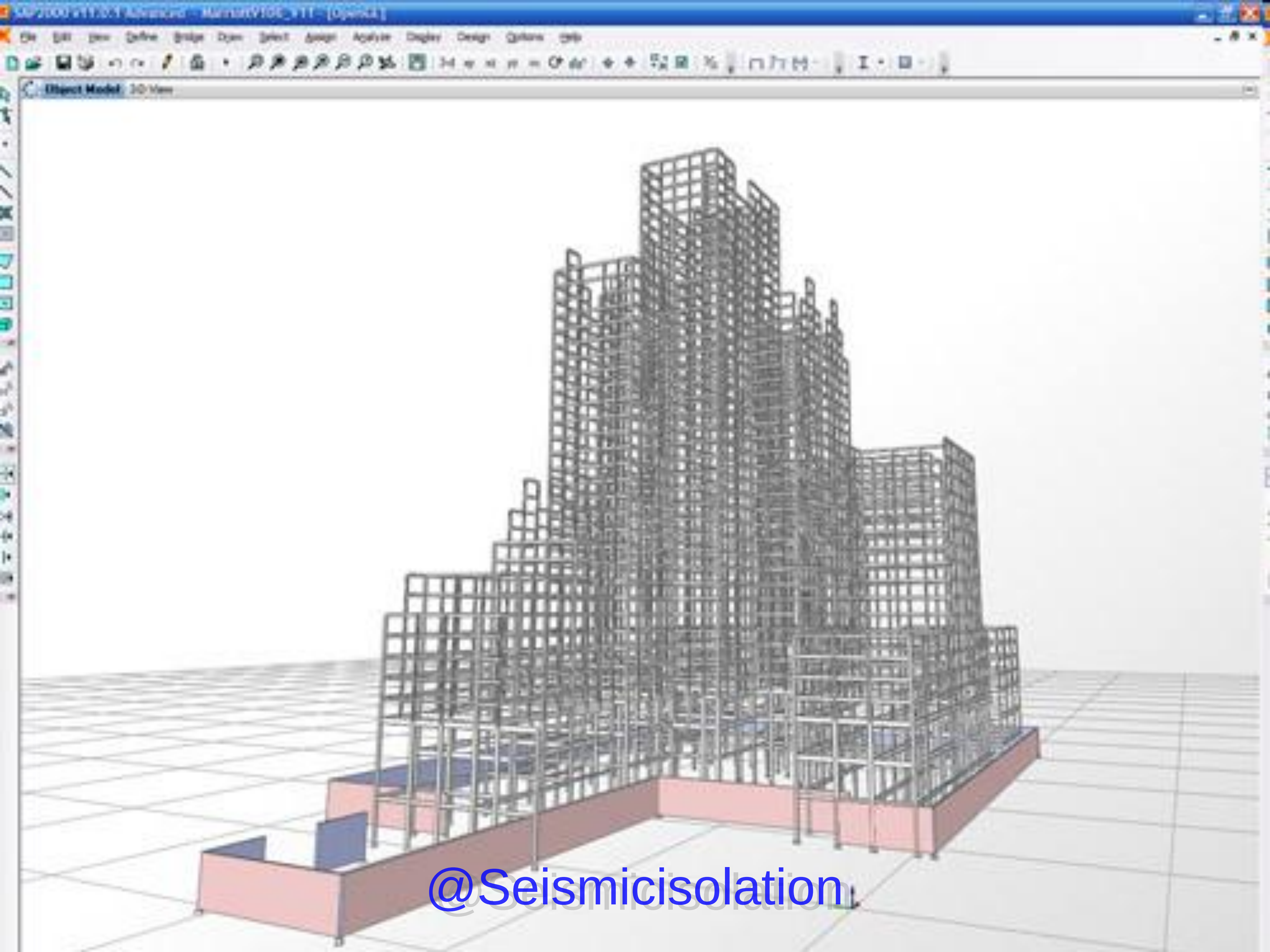


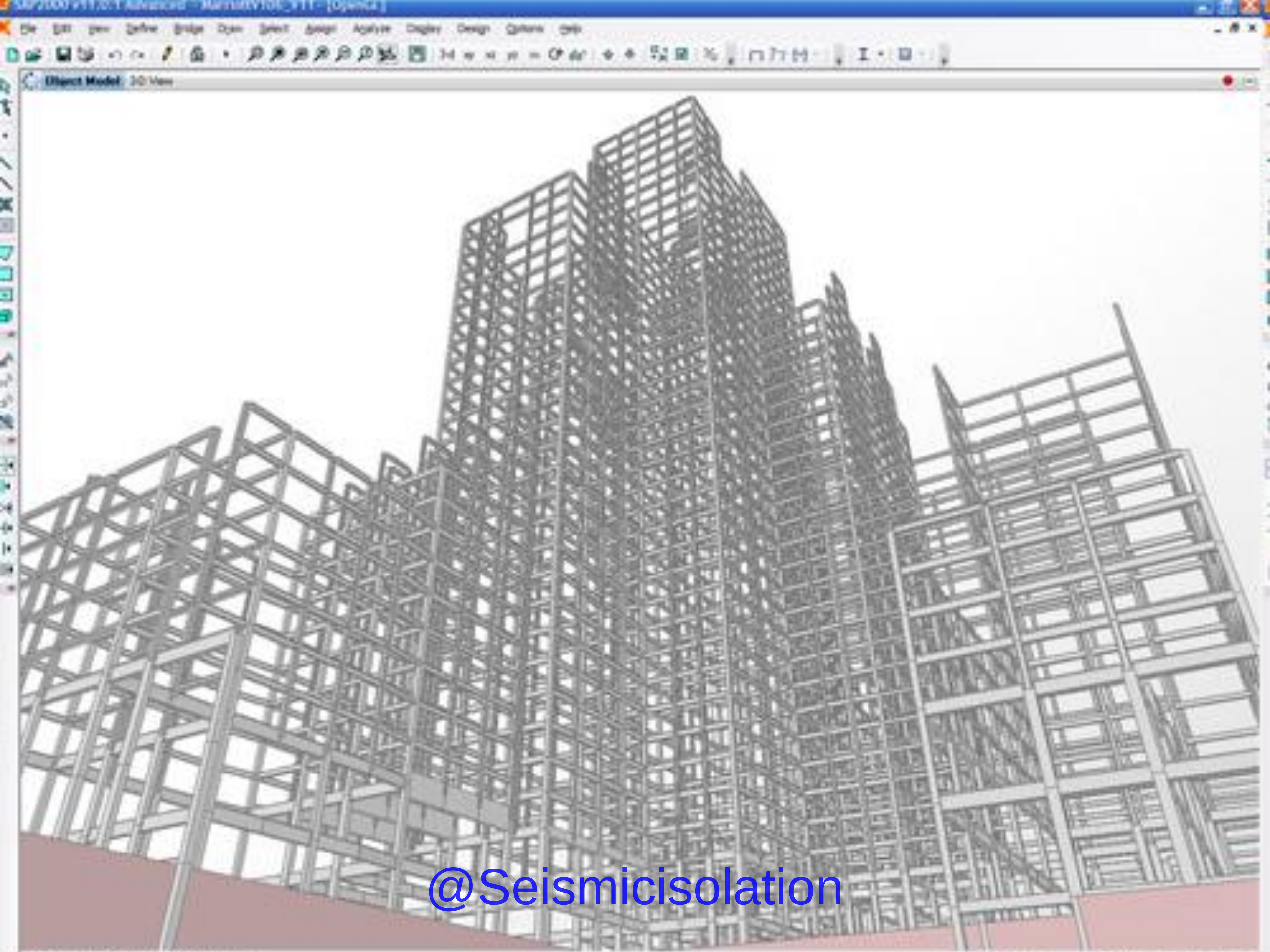

P2000 - Problem 11.1b.lowrise
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-D View
Stress S11 Diagram (COMB1)
-7.00 -6.00 -5.00 -4.00 -3.00 -2.00 -1.00 0.00 1.00
Axial Force Diagram (COMB1)
Axial Force Diagram (COMB1)

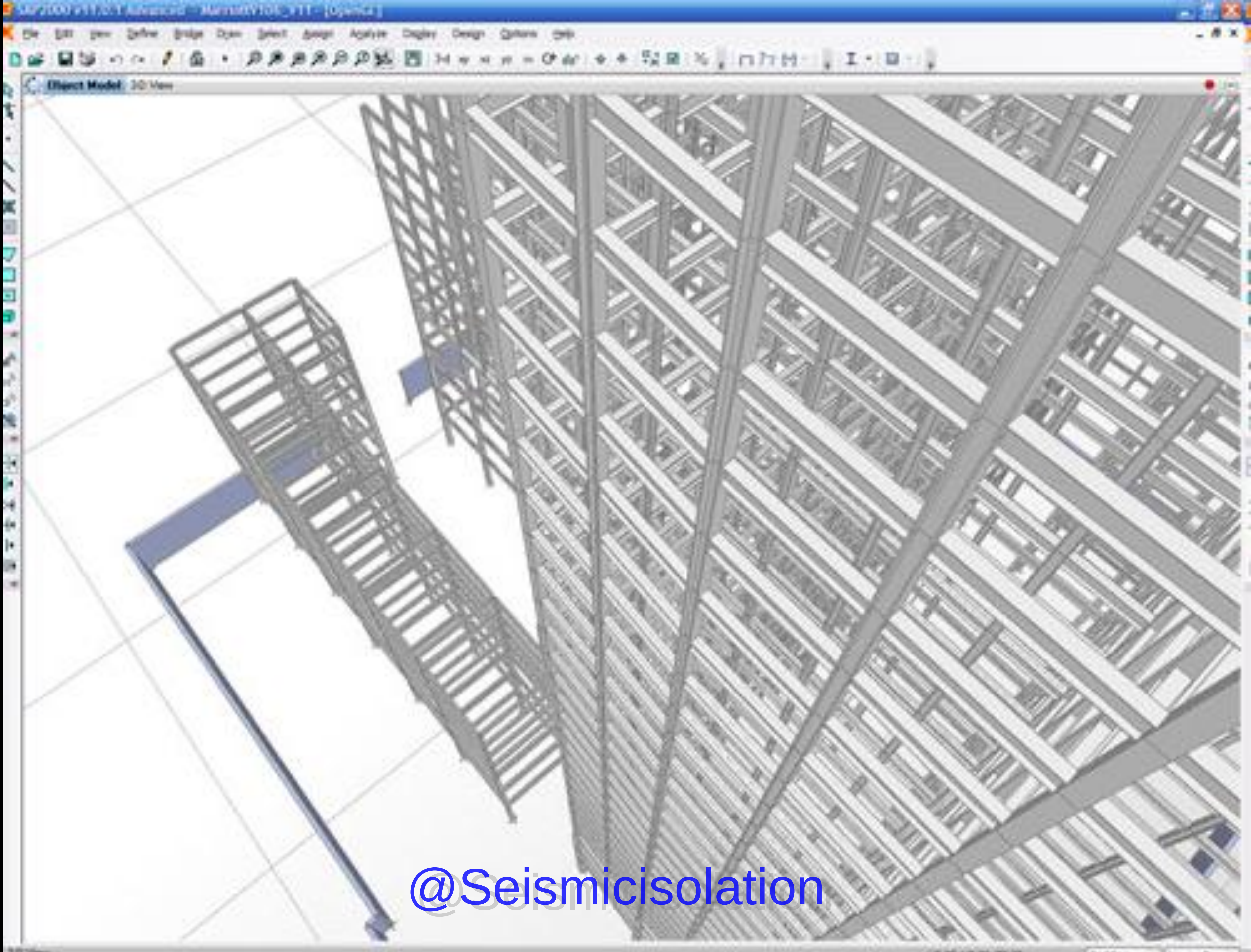

180/2 = 90'
2(180)/3 = 120'
15 Sp @ 12' = 180'
7 Sp @ 25 ft = 175 ft
3 Sp @ 20' = 60'

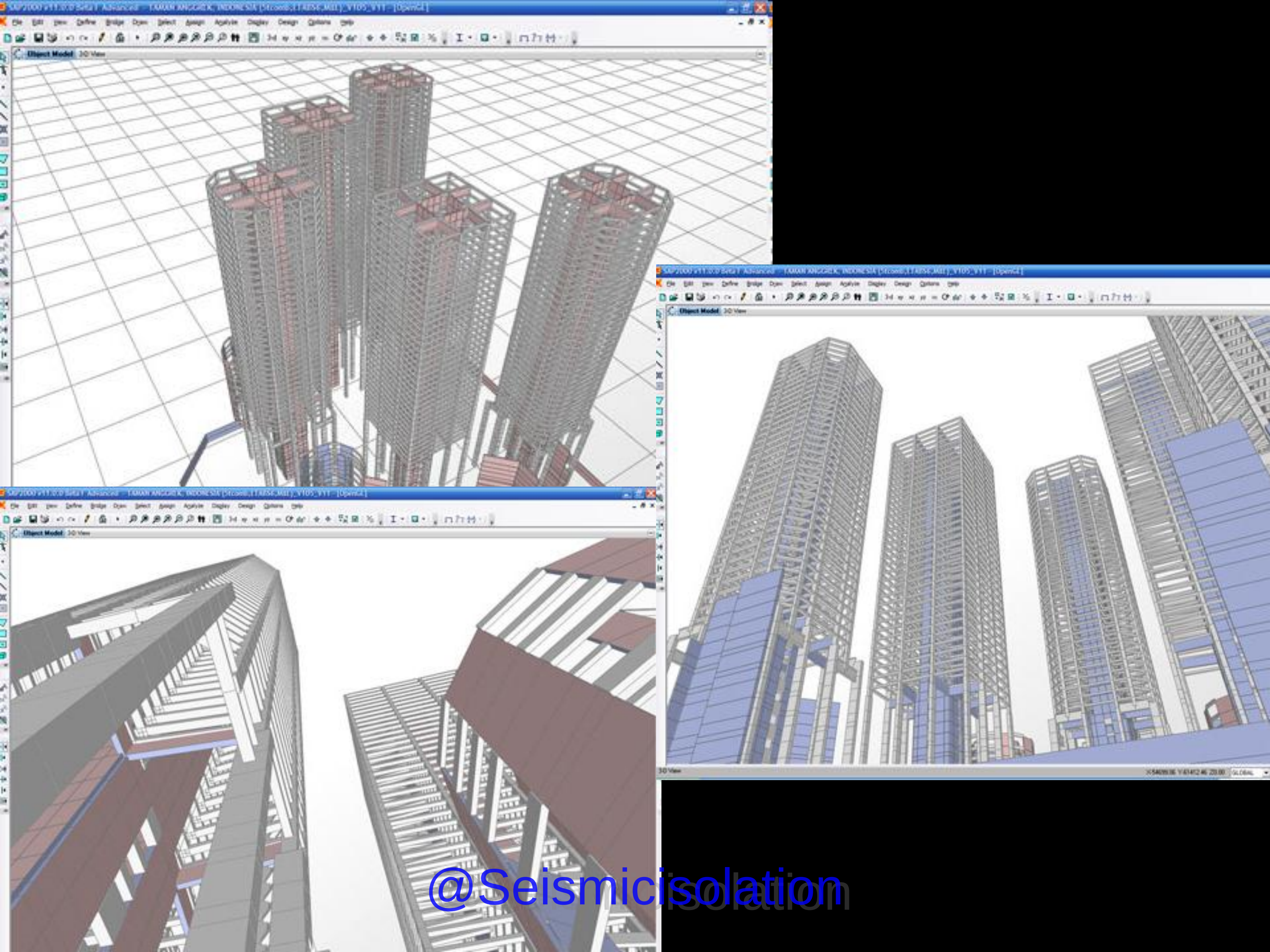


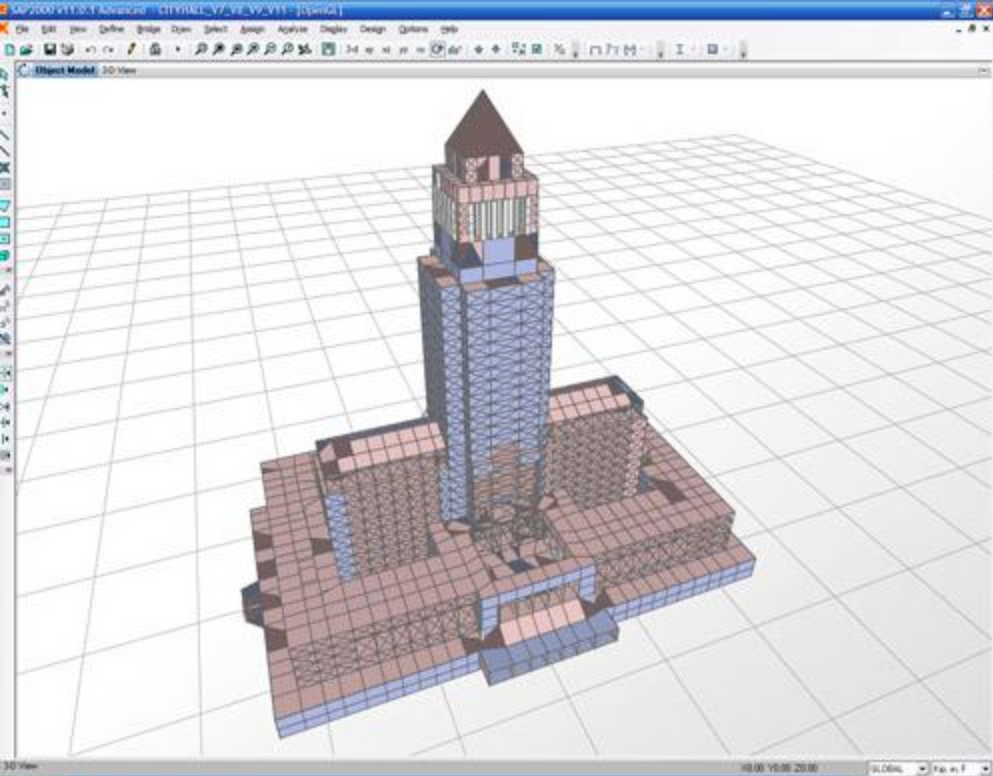
Beijing Jian Wai SOHO, Beijing, 2004, Riken Yamamoto

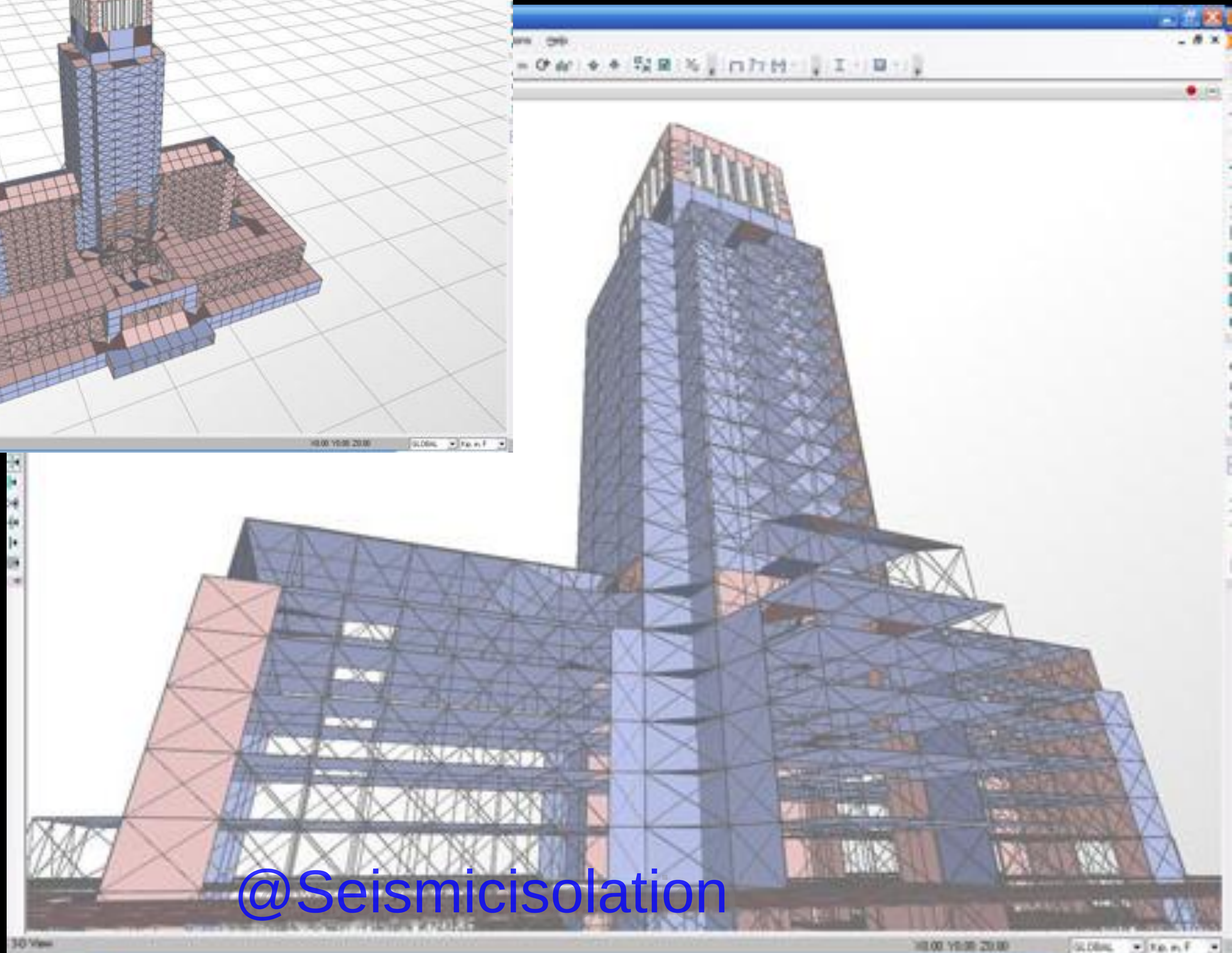

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3D View

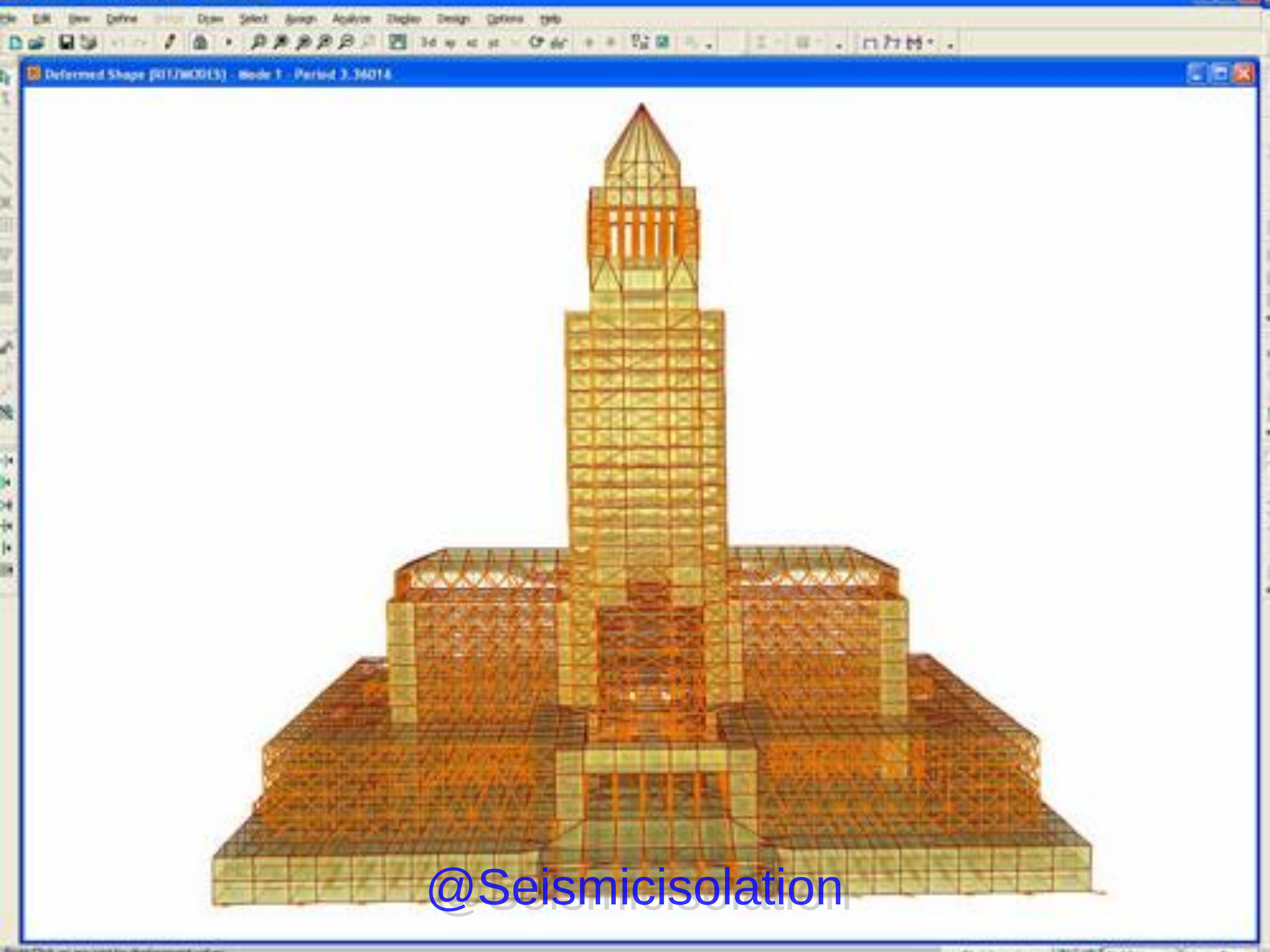


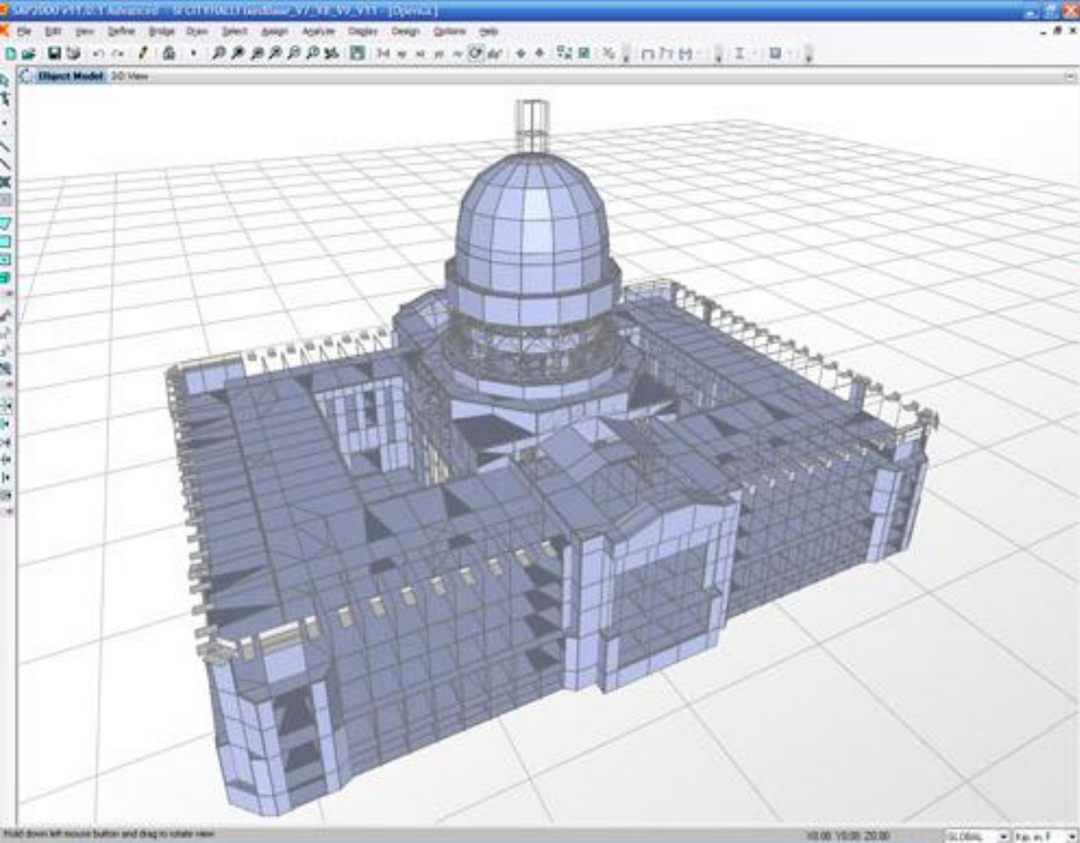

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model

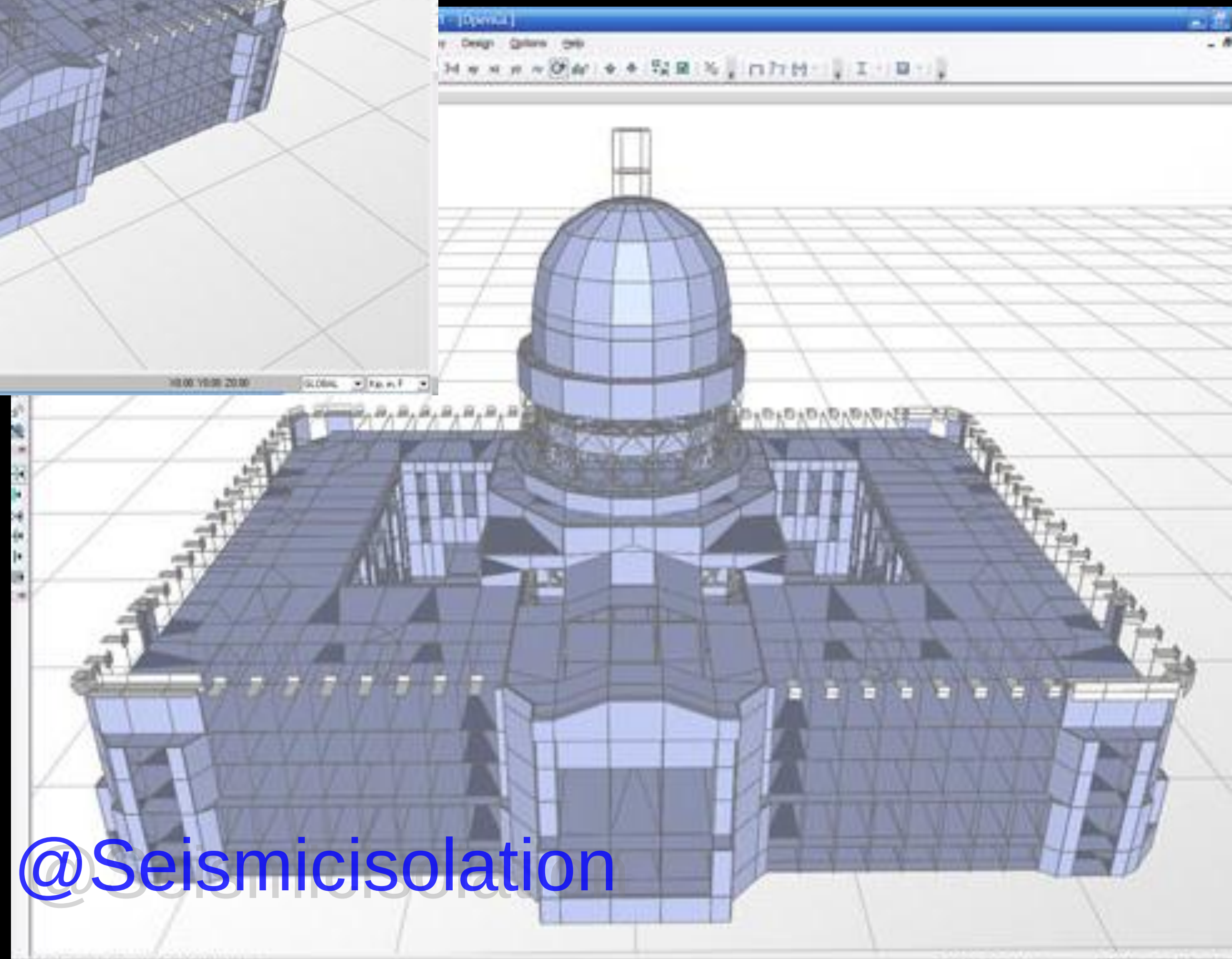


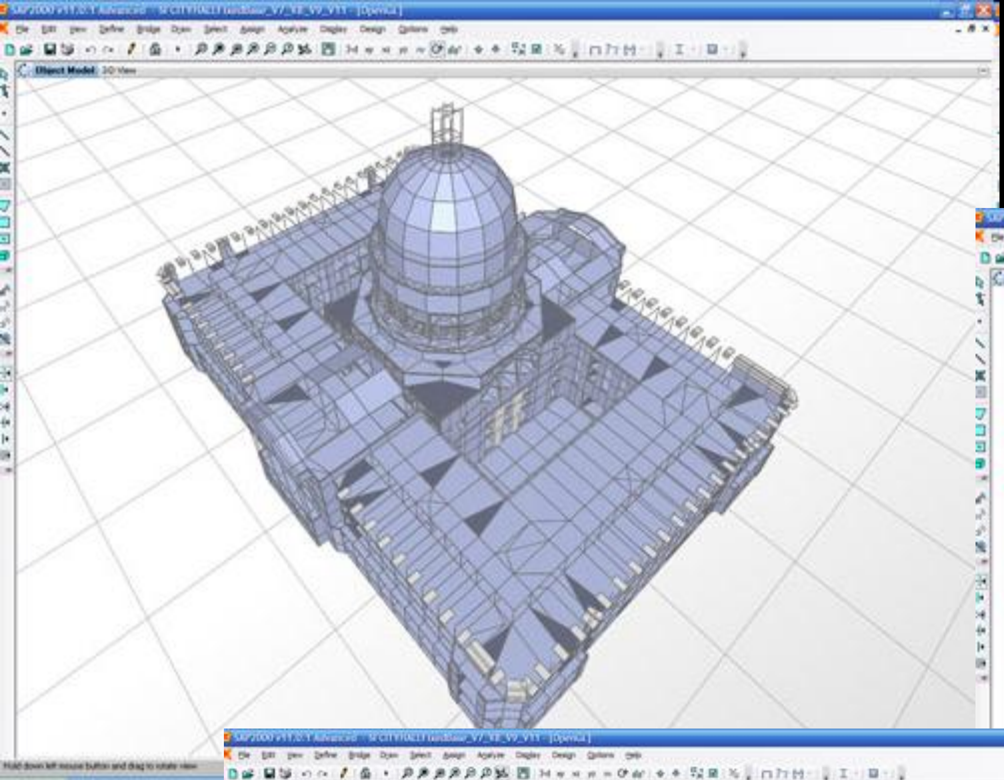


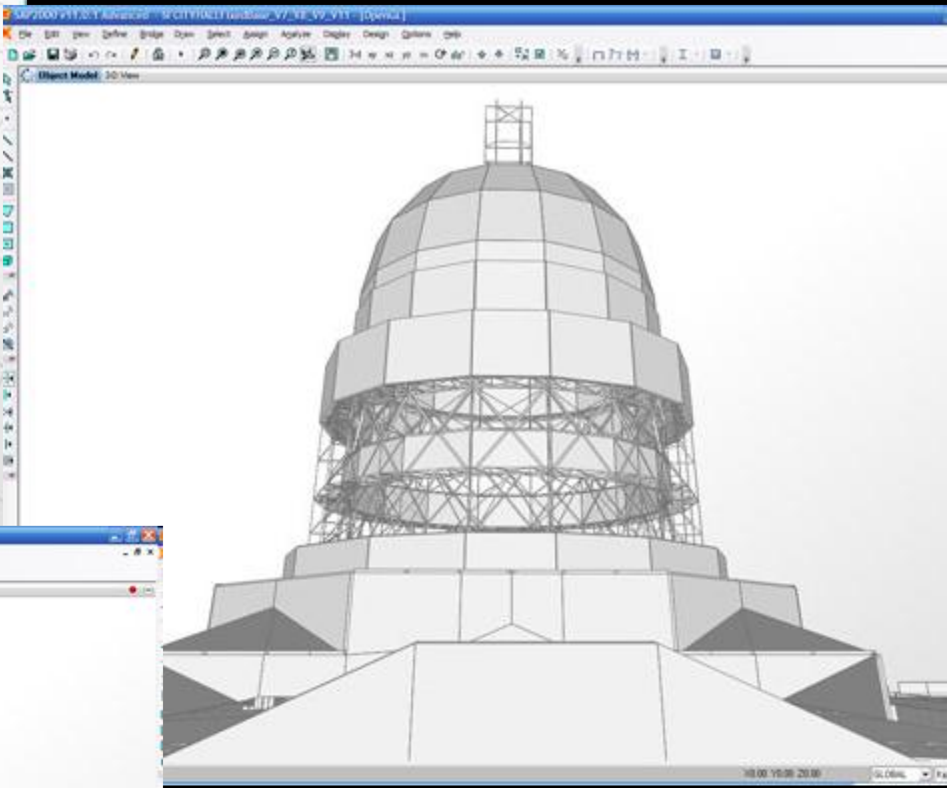


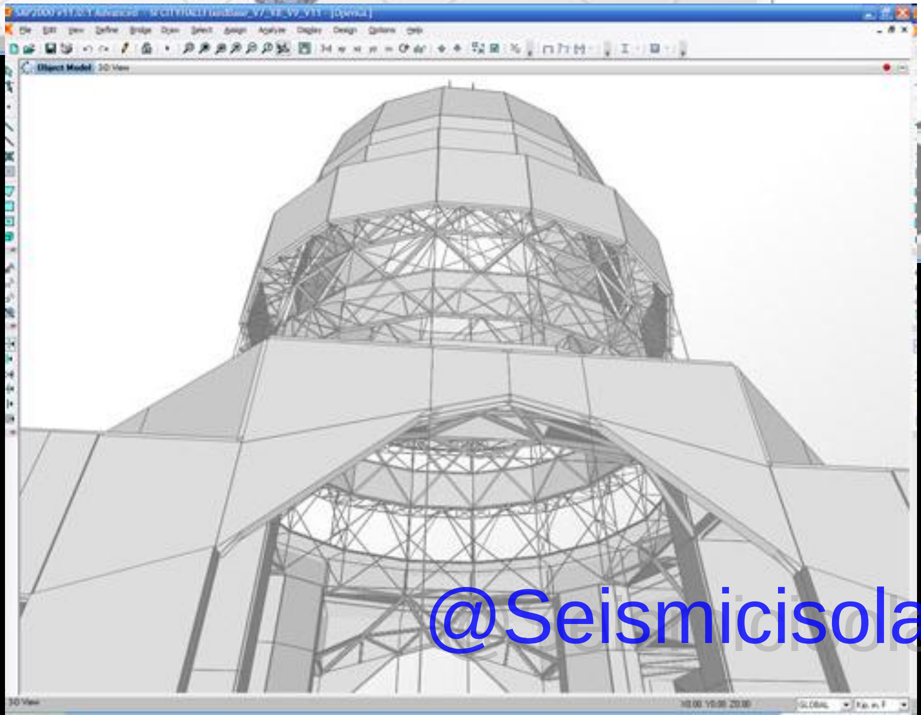

File Edit View Define Draw Select Assign Analyze Display Design Options Help
Deformed Shape - Mode 1 - Period 3.36014

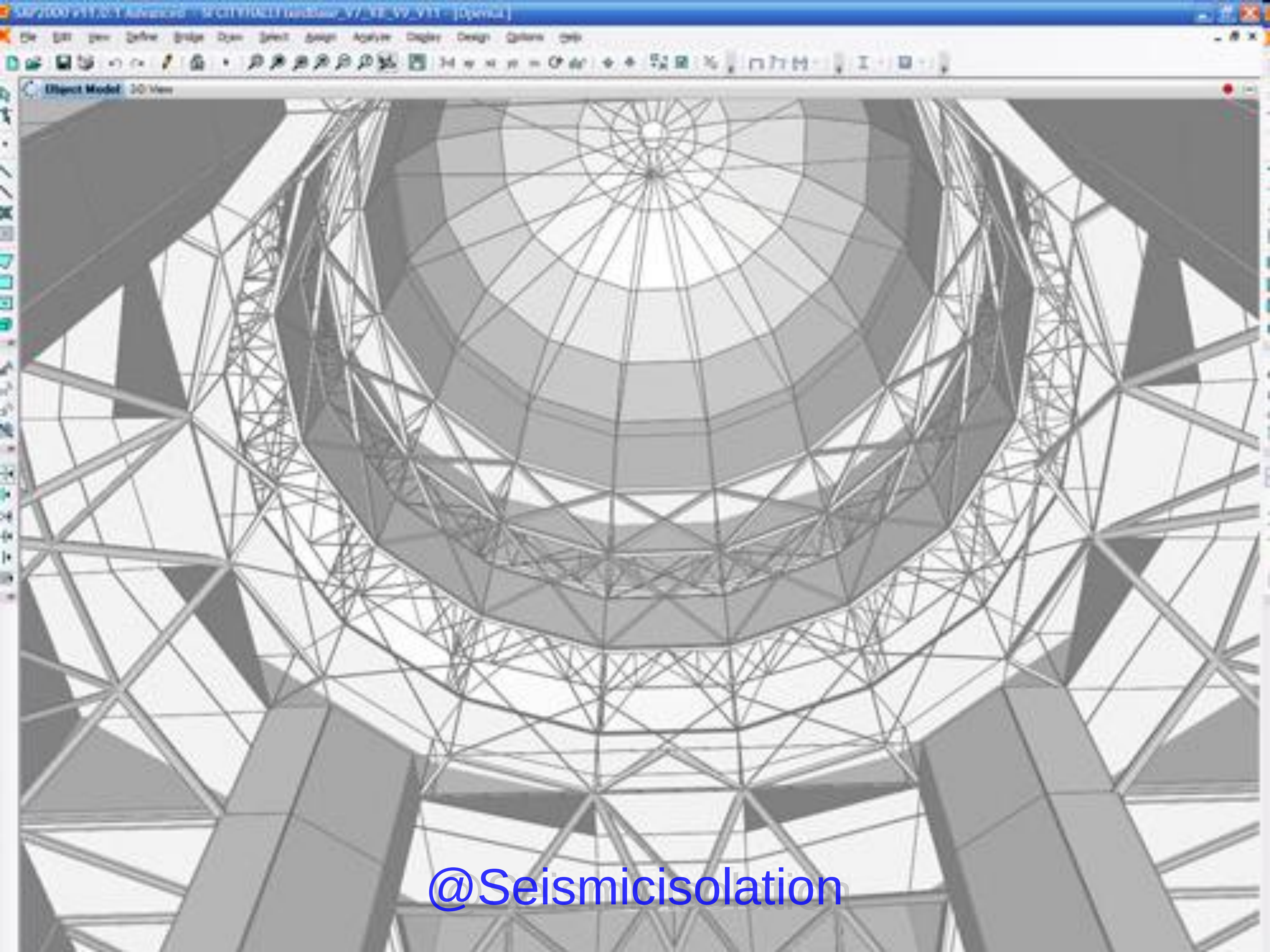


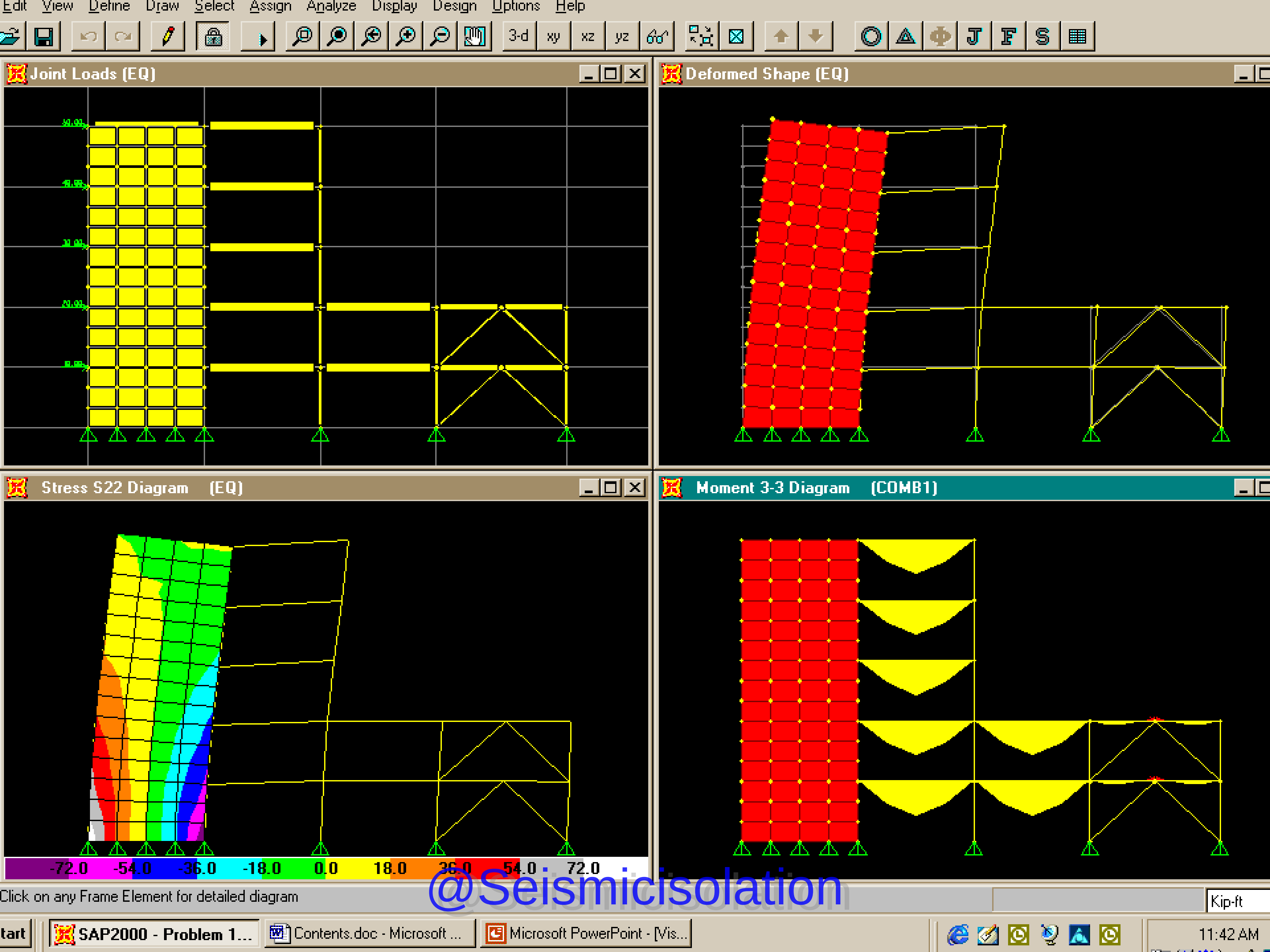




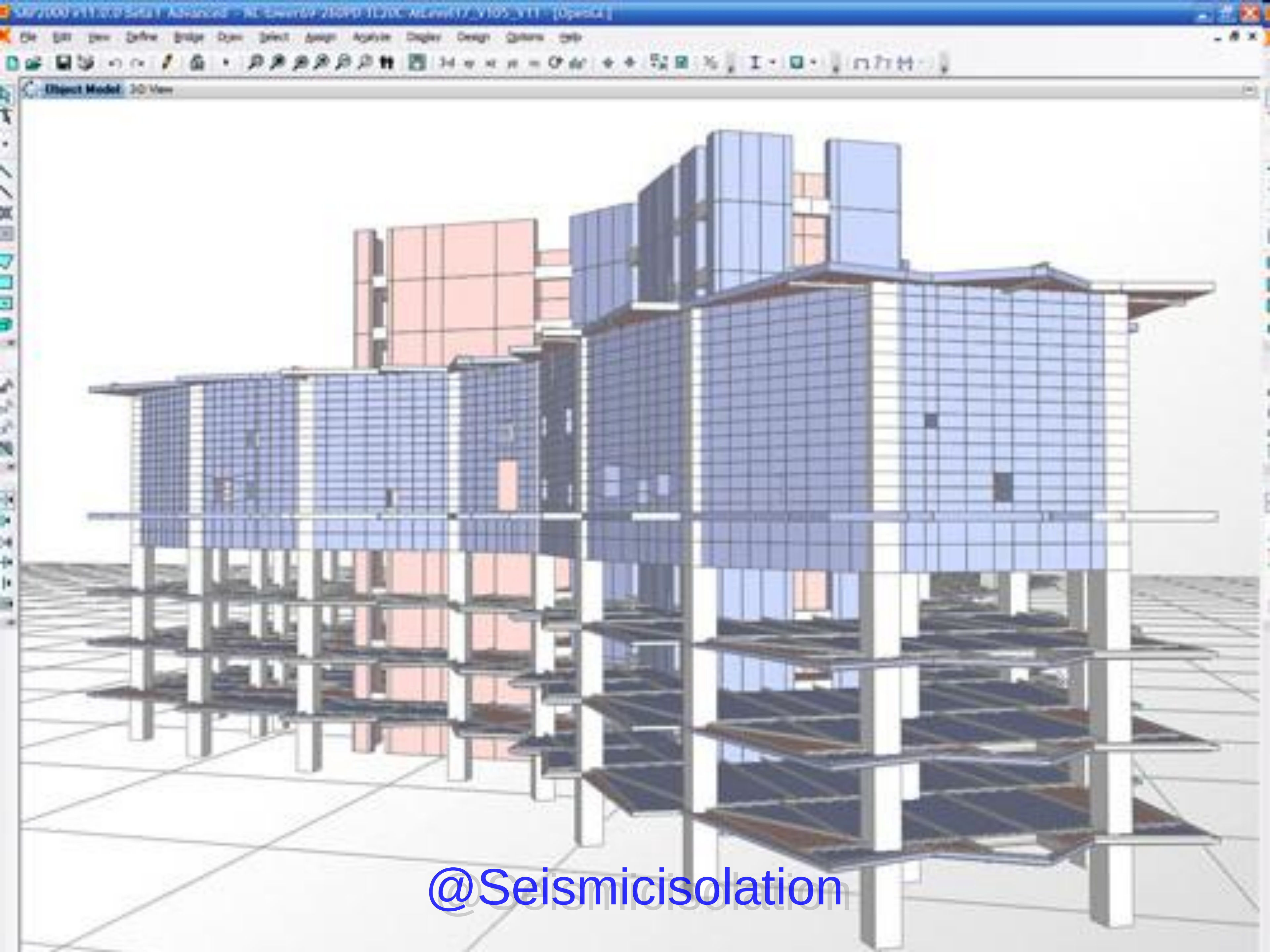


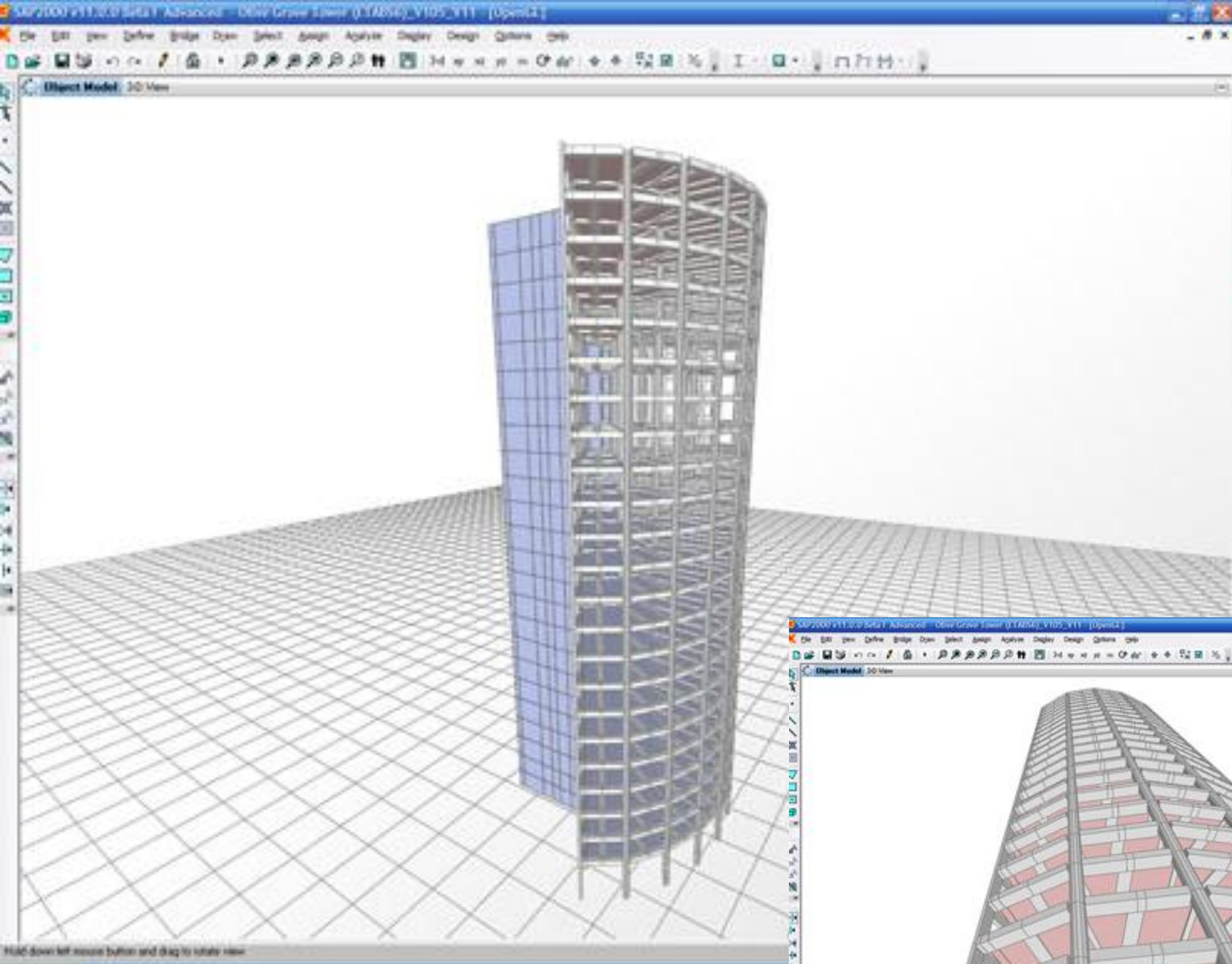


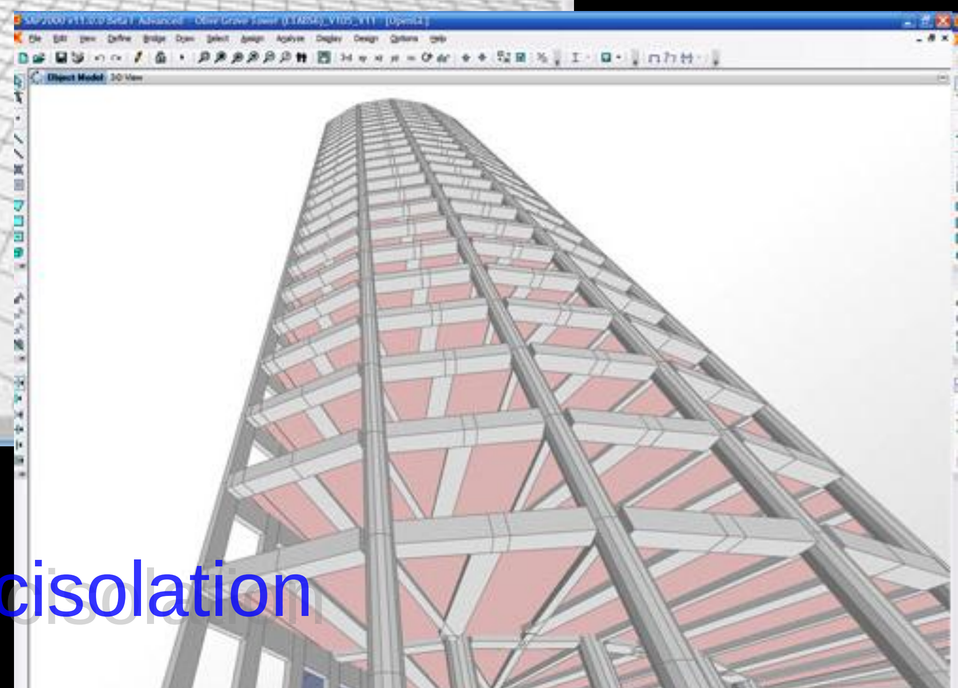


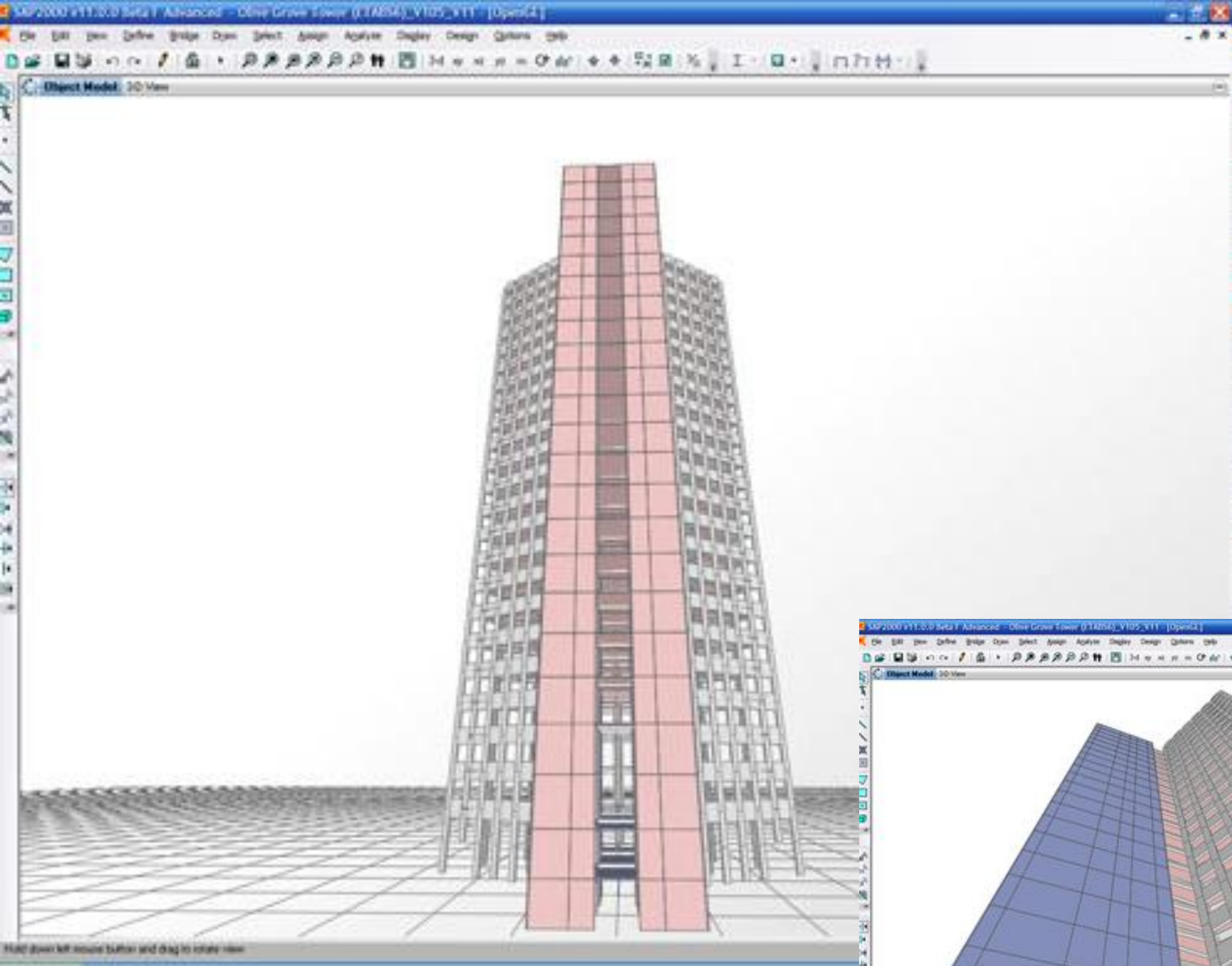


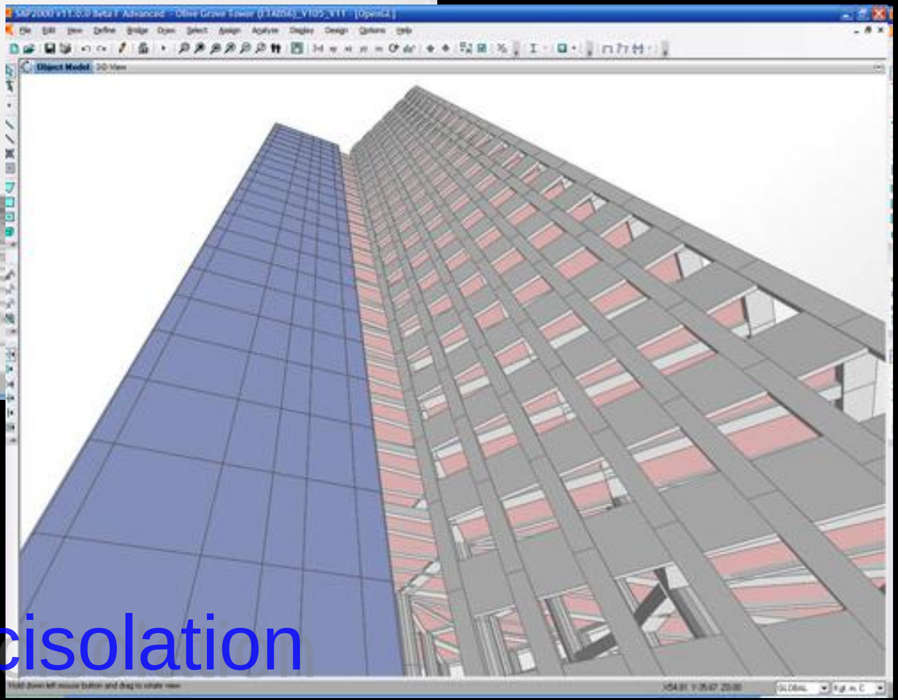
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Joint Loads (EQ)
Deformed Shape (EQ)
Stress S22 Diagram (EQ)
Moment 3-3 Diagram (COMB1)
-72.0 -54.0 -36.0 -18.0 0.0 18.0 36.0 54.0 72.0
@Seismicisolation
Click on any Frame Element for detailed diagram
Kip-ft
tart
SAP2000 - Problem 1...
Contents.doc - Microsoft ...
Microsoft PowerPoint - [Vis...
11:42 AM

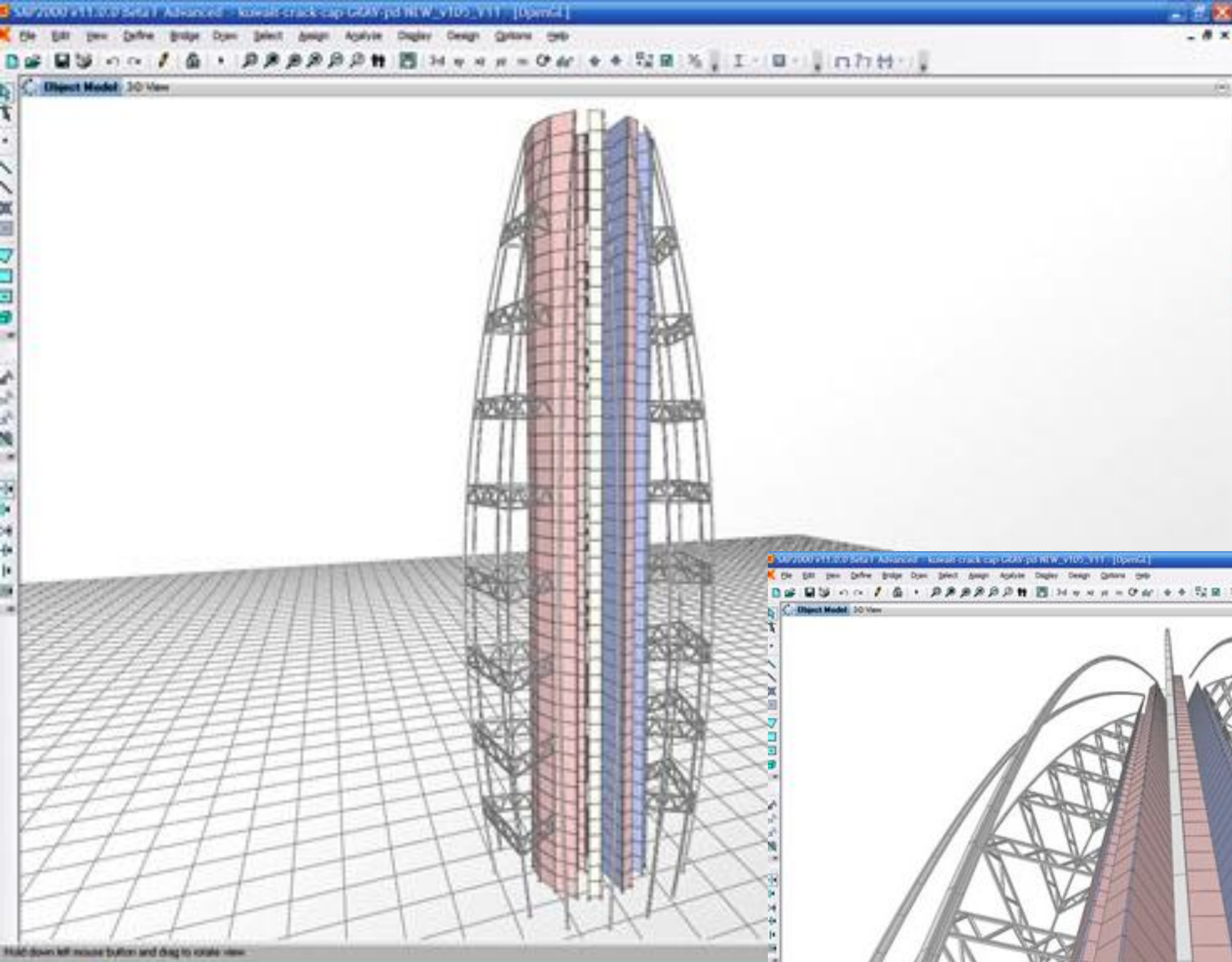

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model - 3D View

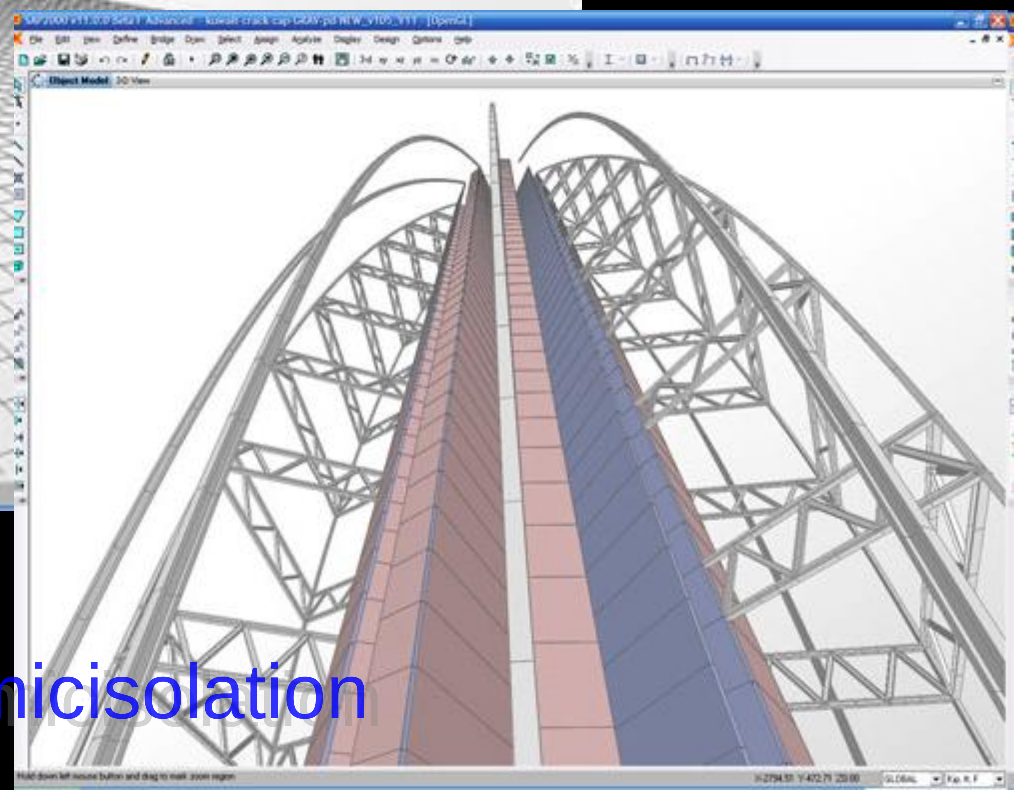


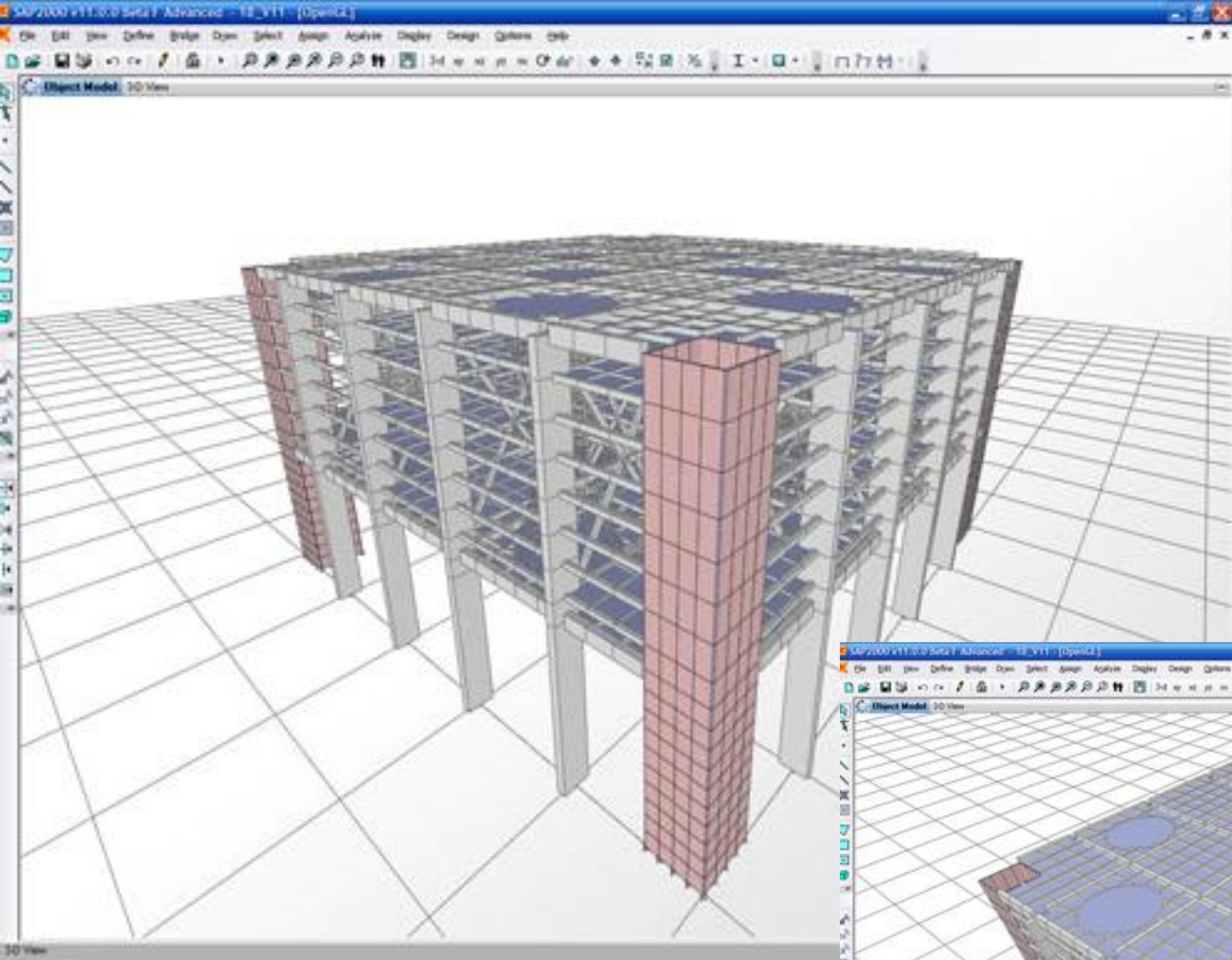


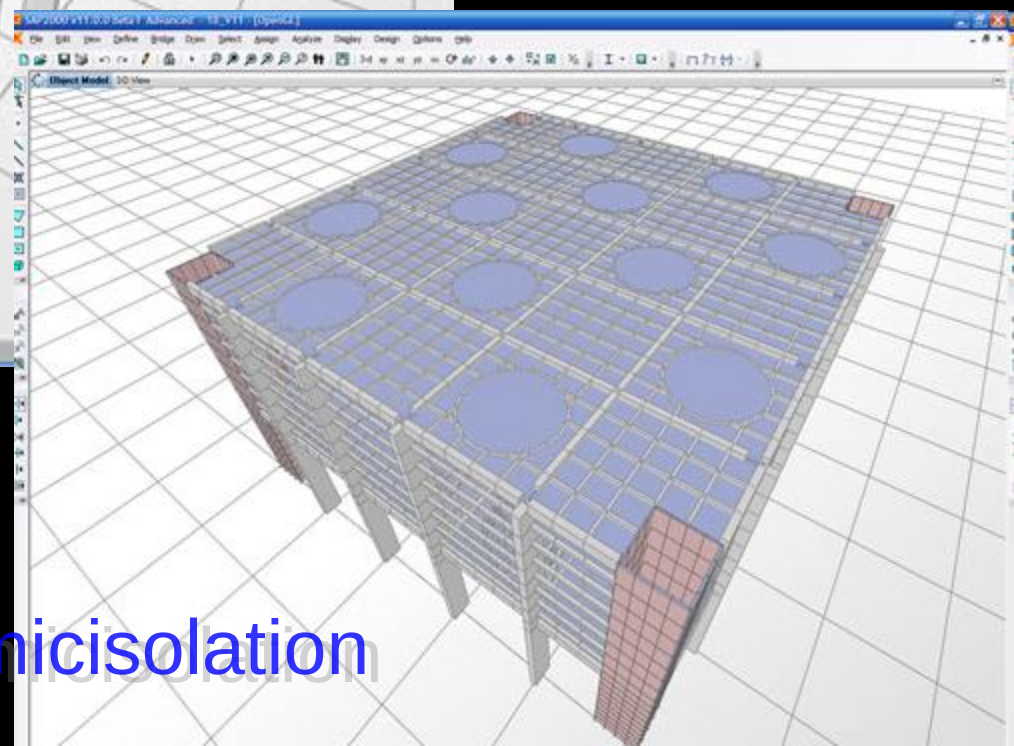

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help

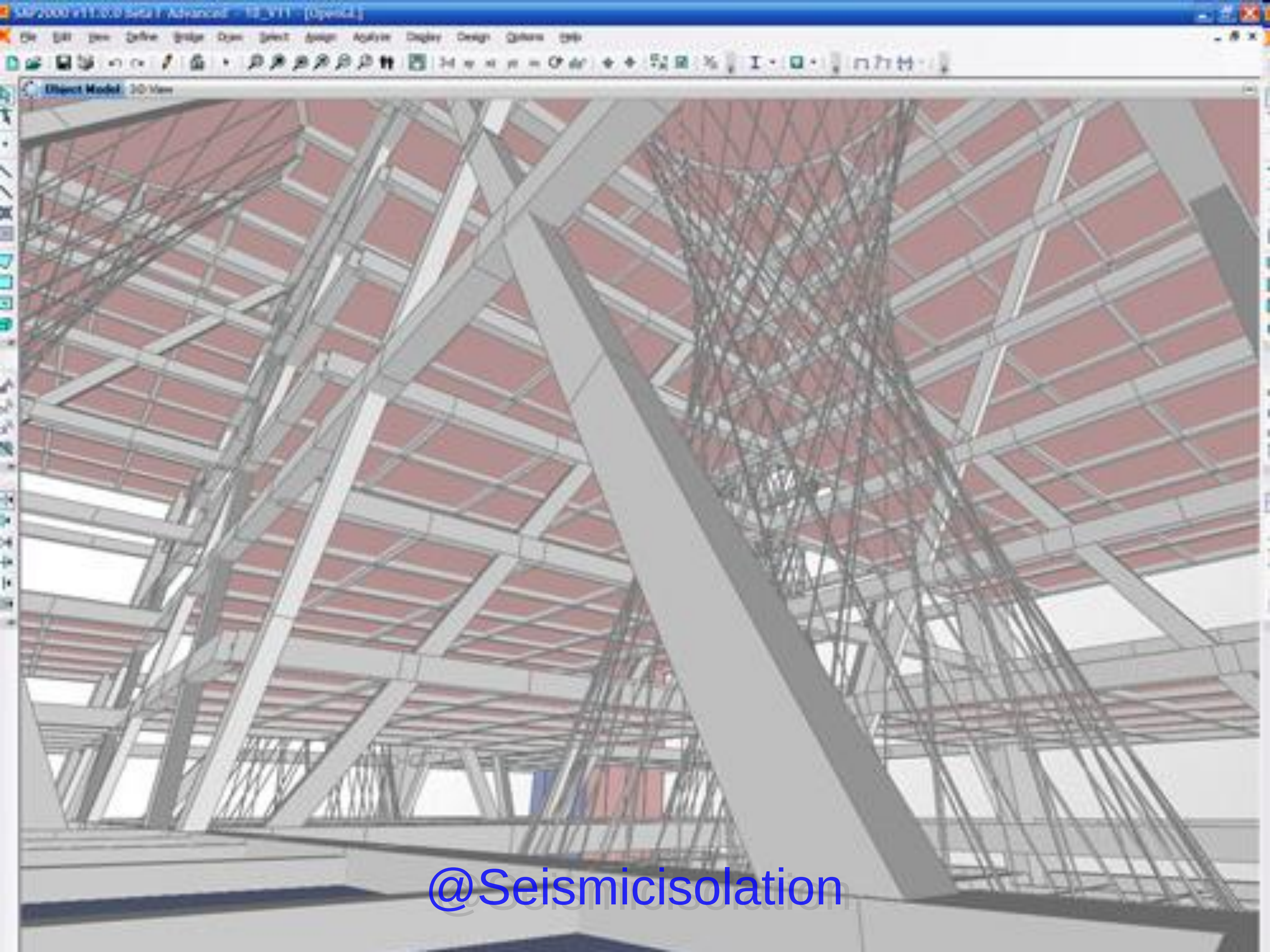


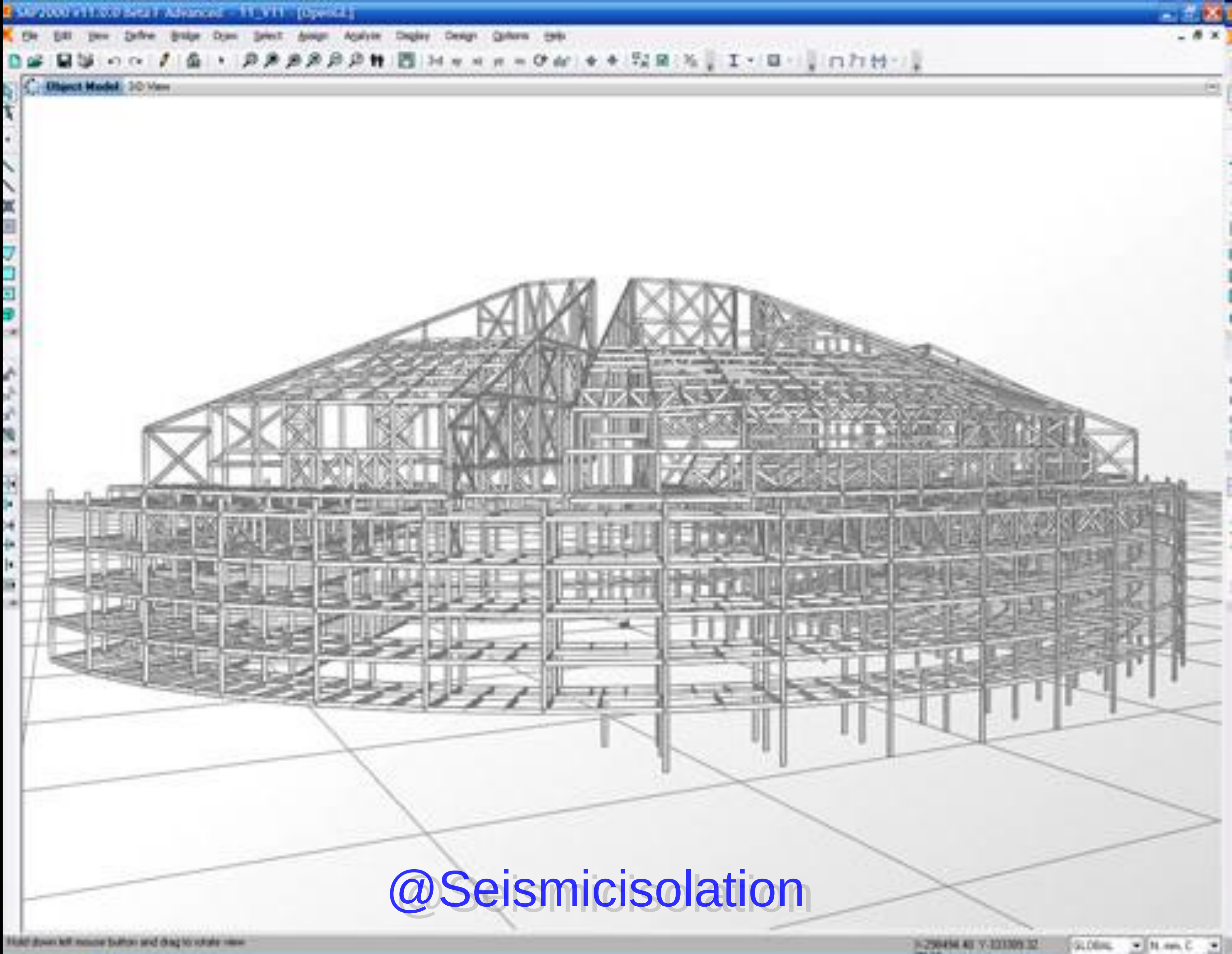


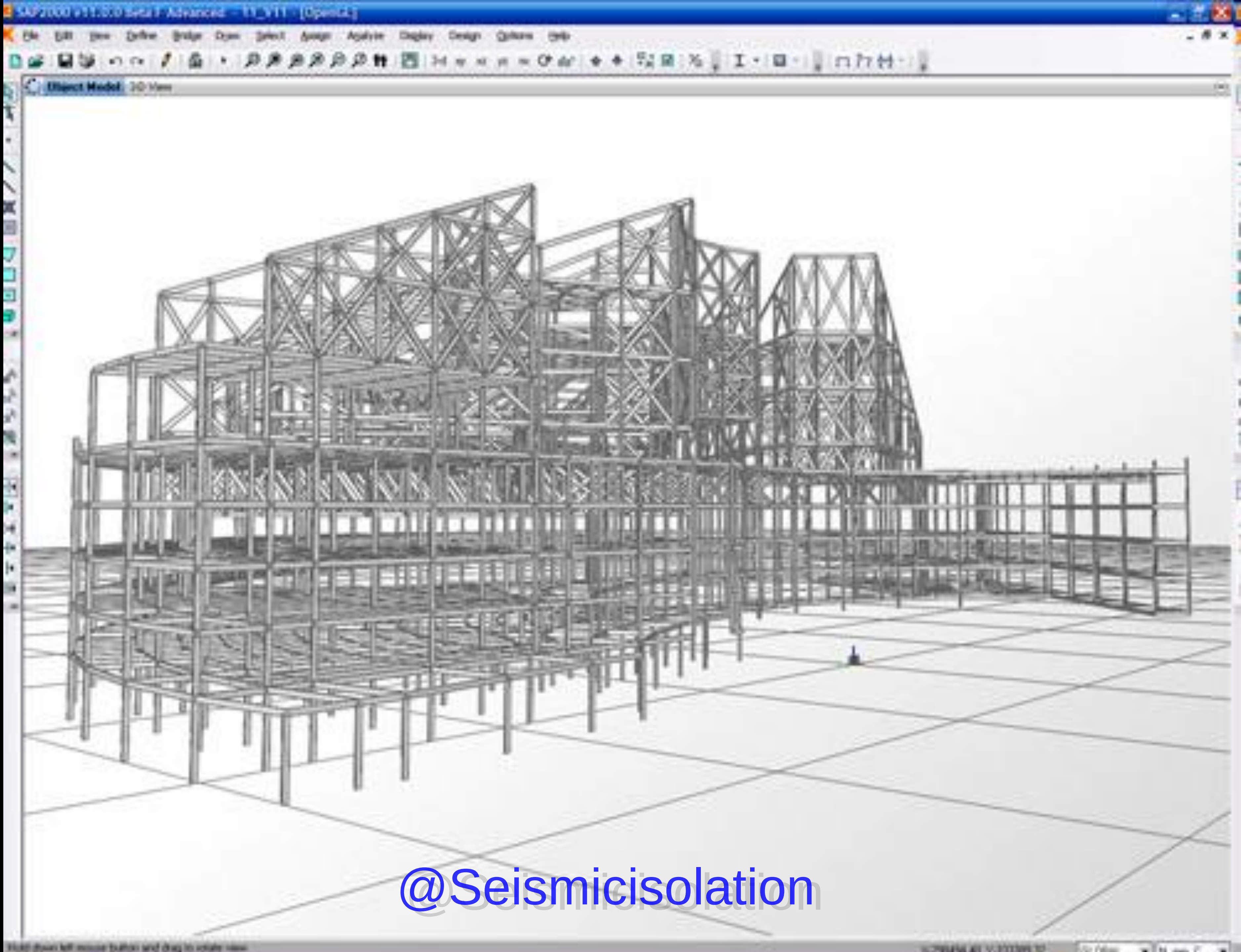


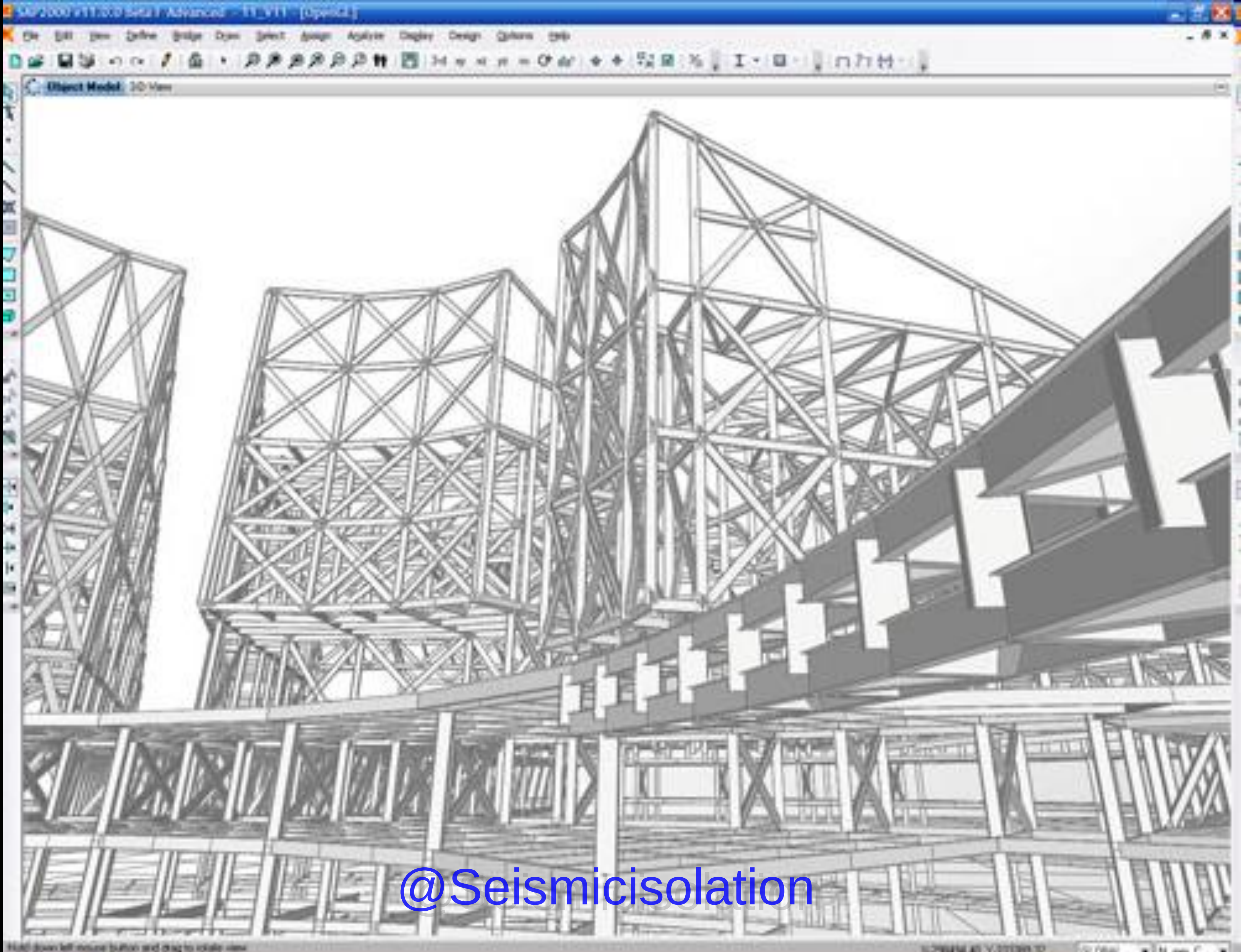


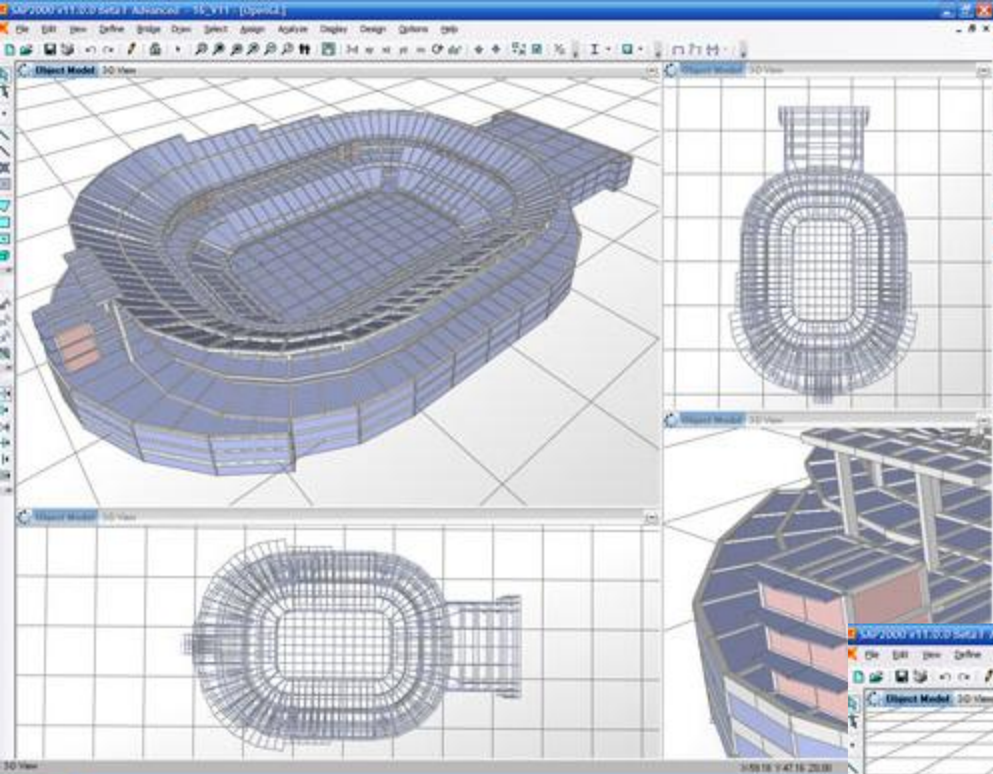


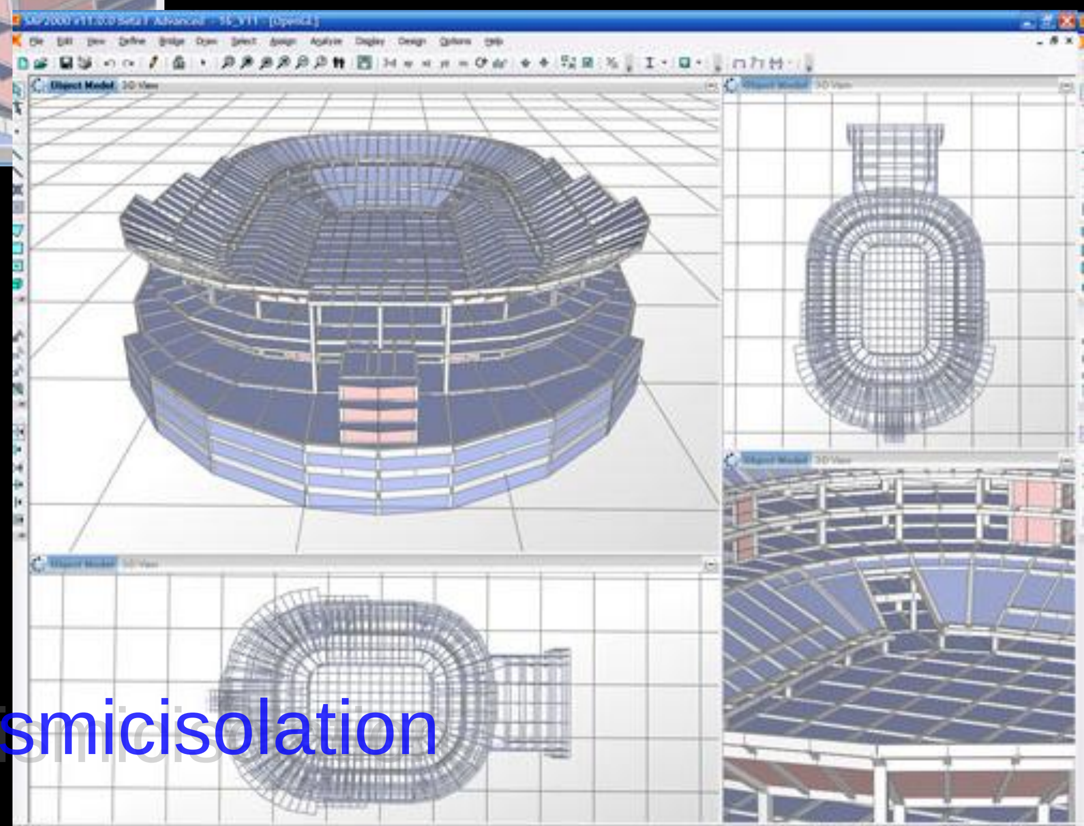

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model - 3D View

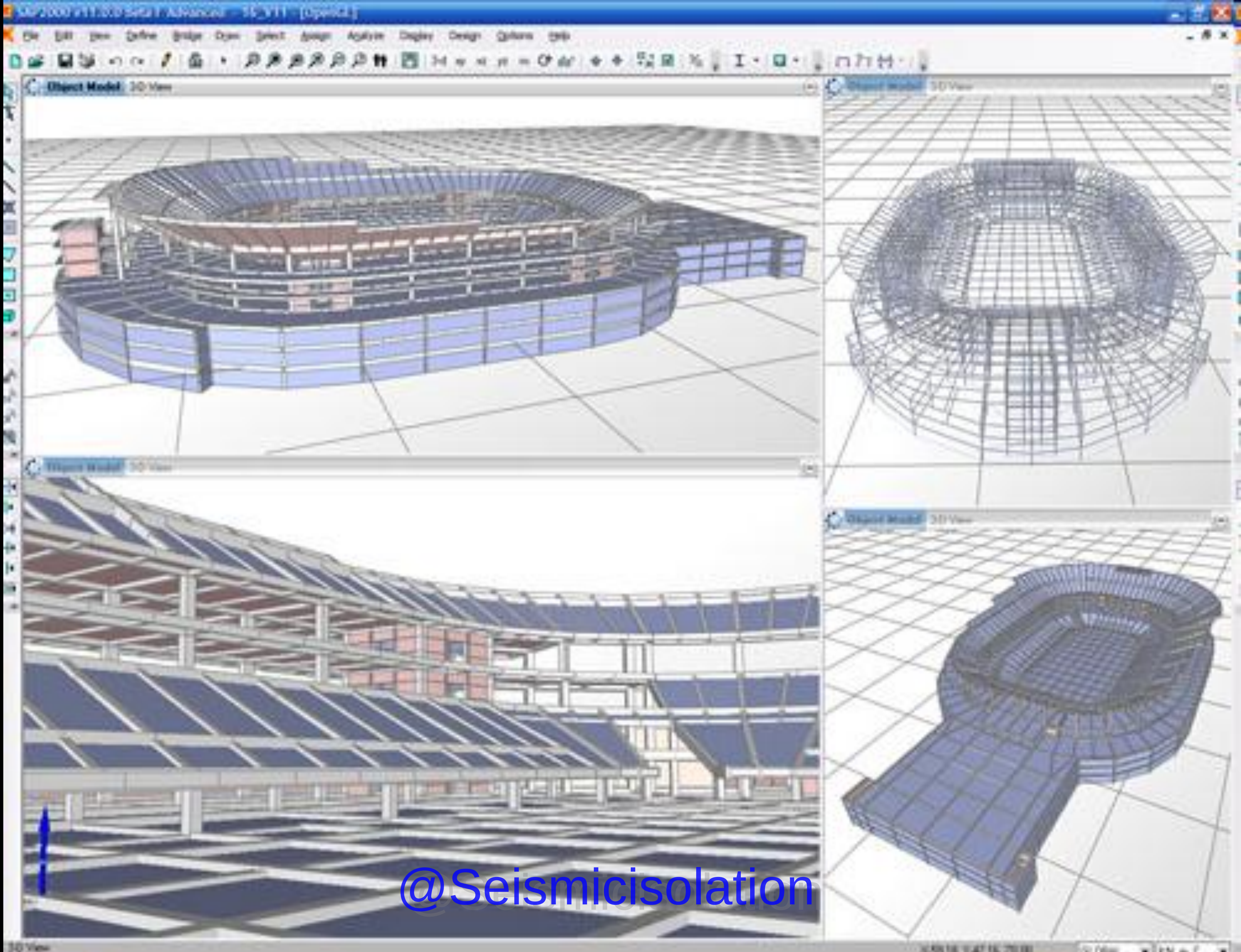

File
Edit
View
Define
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
Object Model




File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model



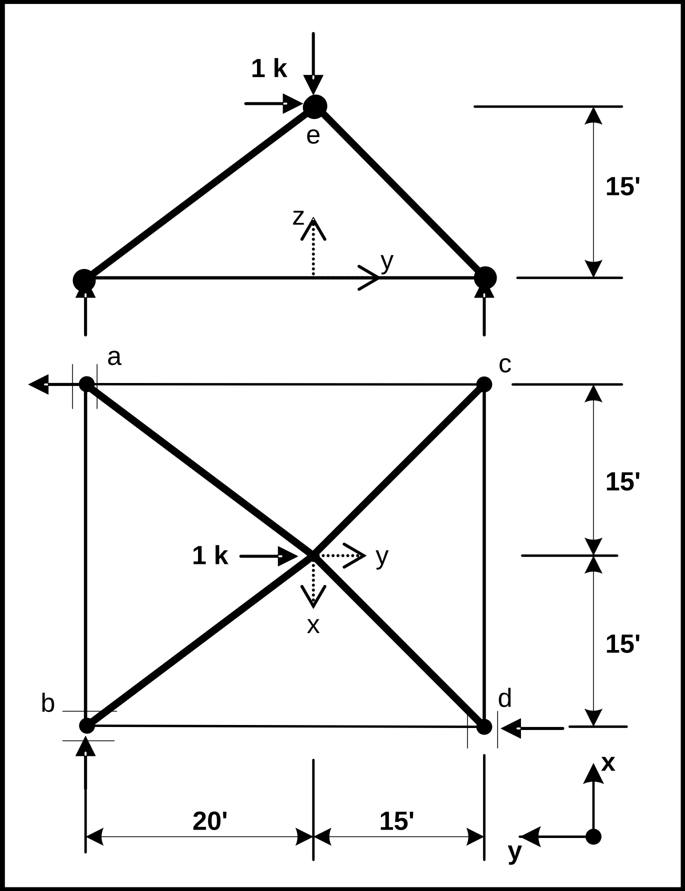


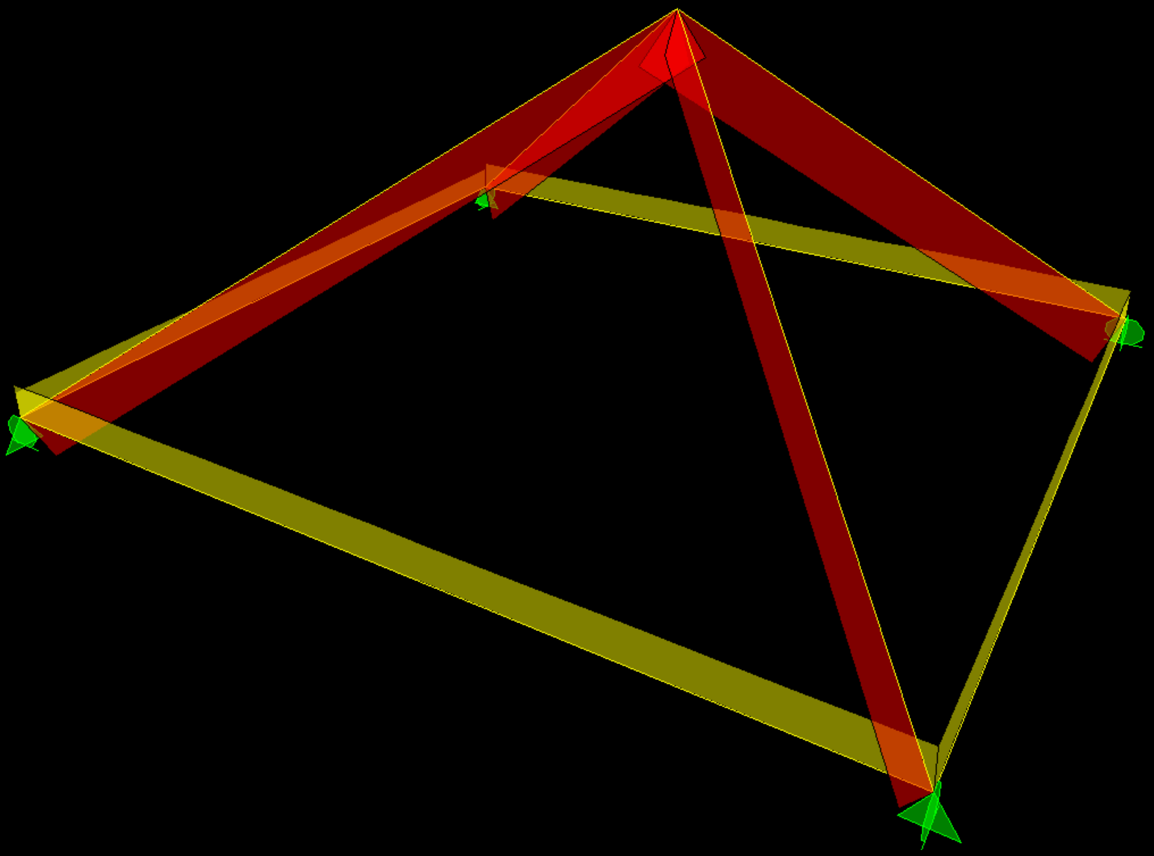

Object Model
3D View

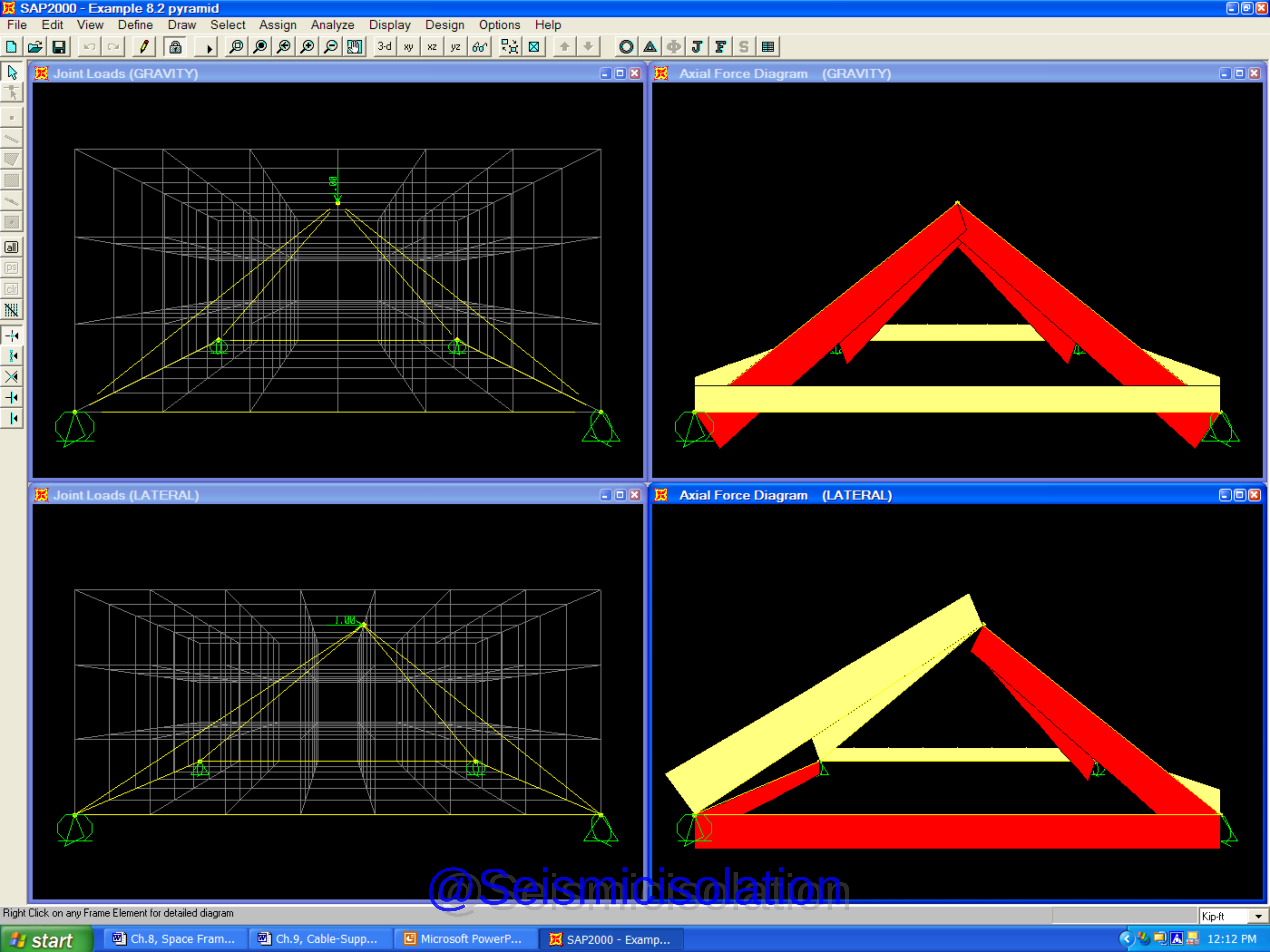
SPACE FRAMES

Cologne/Bonn Airport, Germany, 2000, Helmut Jahn Arch., Ove Arup USA Struct. Eng.


REISEMARKT
15 8 2001

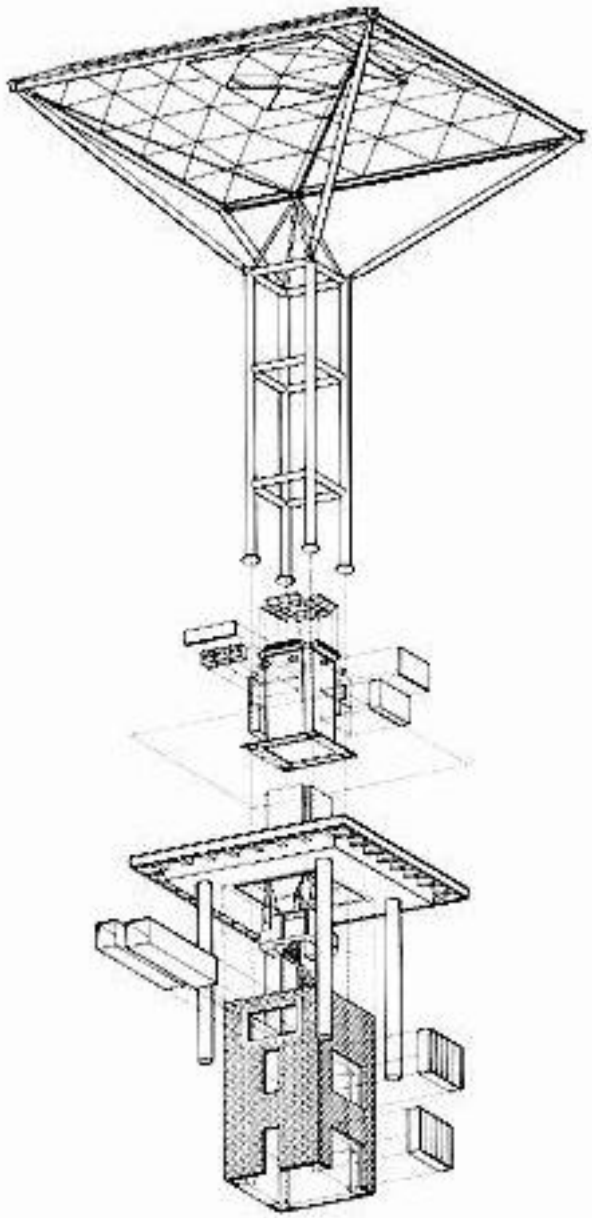

1 k
e
15'
z
y
a
c
15'
1 k
y
x
15'
b
d
x
20'
15'
y

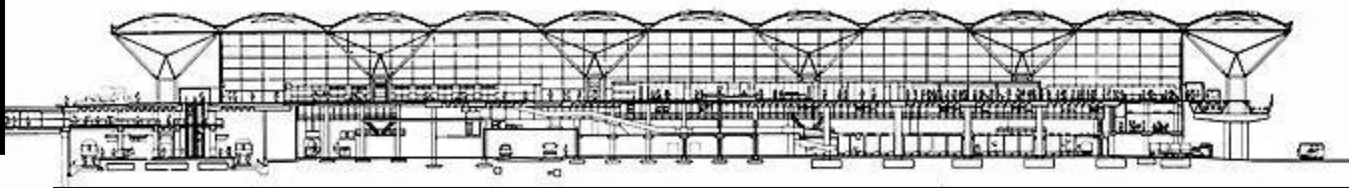


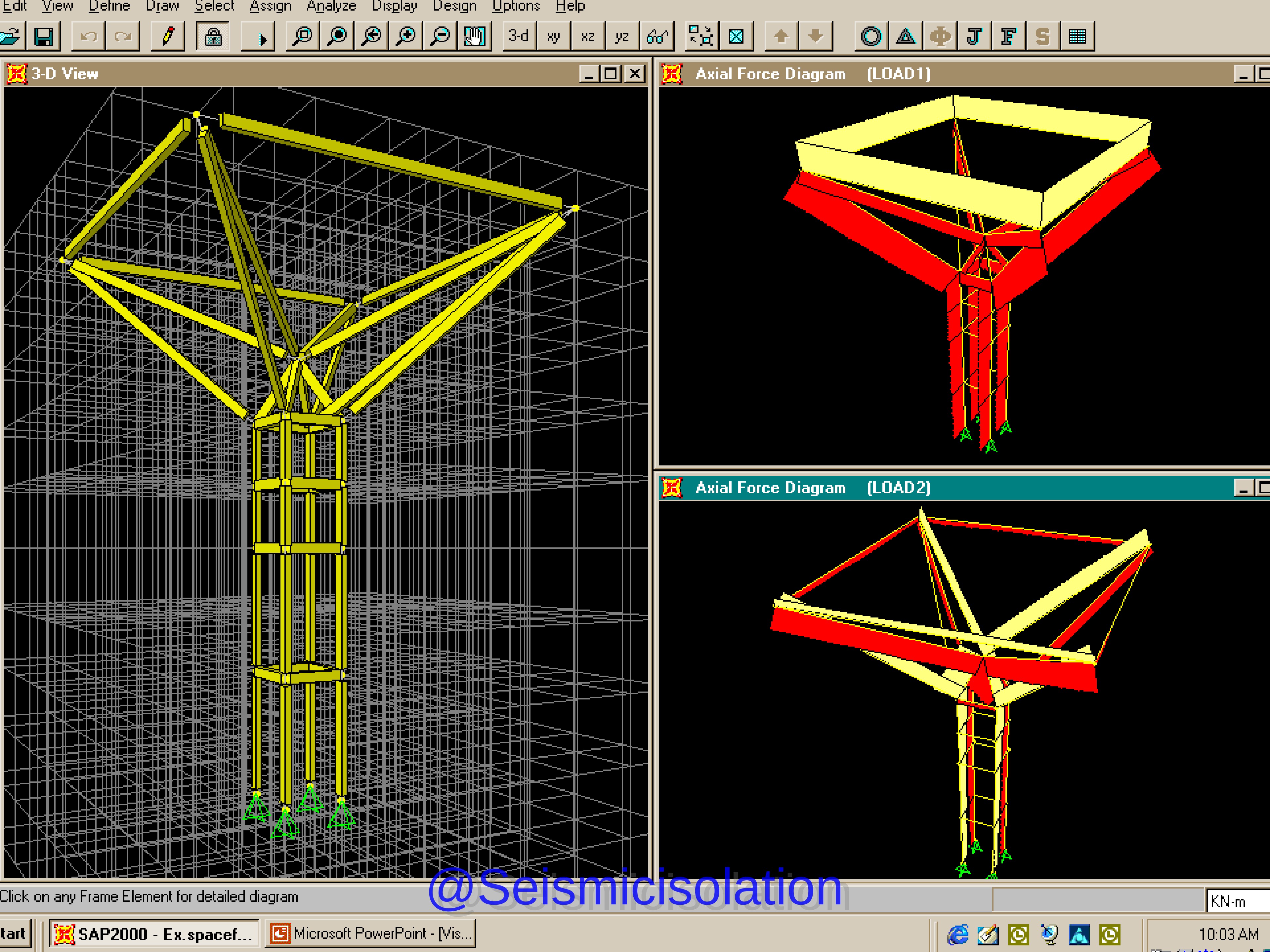


SAP2000 - Example 8.2 pyramid
File Edit View Define Draw Select Assign Analyze Display Design Options Help
Joint Loads (GRAVITY)
Axial Force Diagram (GRAVITY)
Joint Loads (LATERAL)
1.00
Axial Force Diagram (LATERAL)
Right Click on any Frame Element for detailed diagram
Kip-ft
start
Ch.8, Space Fram...
Ch.9, Cable-Supp...
Microsoft PowerP...
SAP2000 - Examp...
12:12 PM

Stansted Airport, London, UK, 1991, Foster/Arup

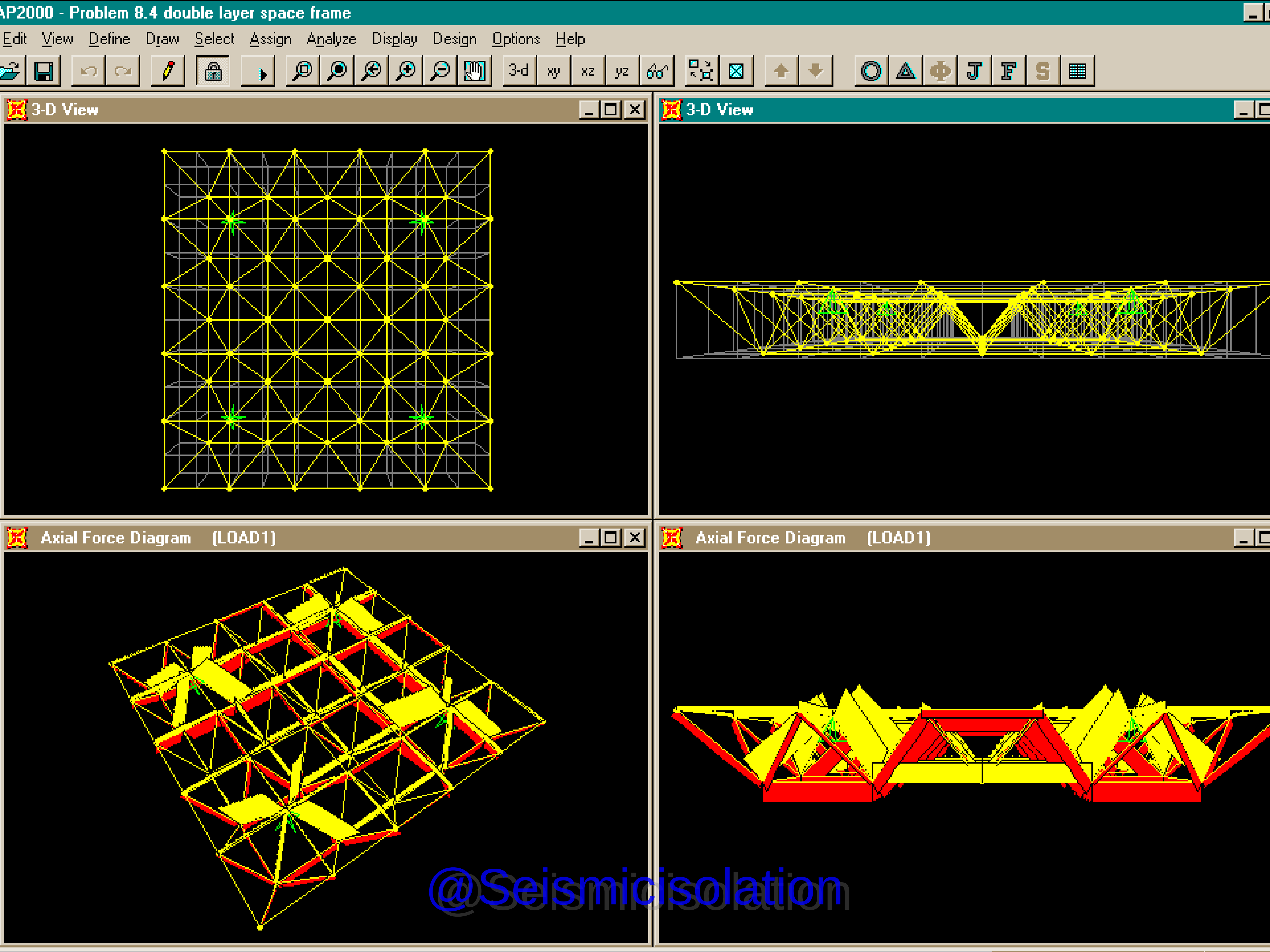


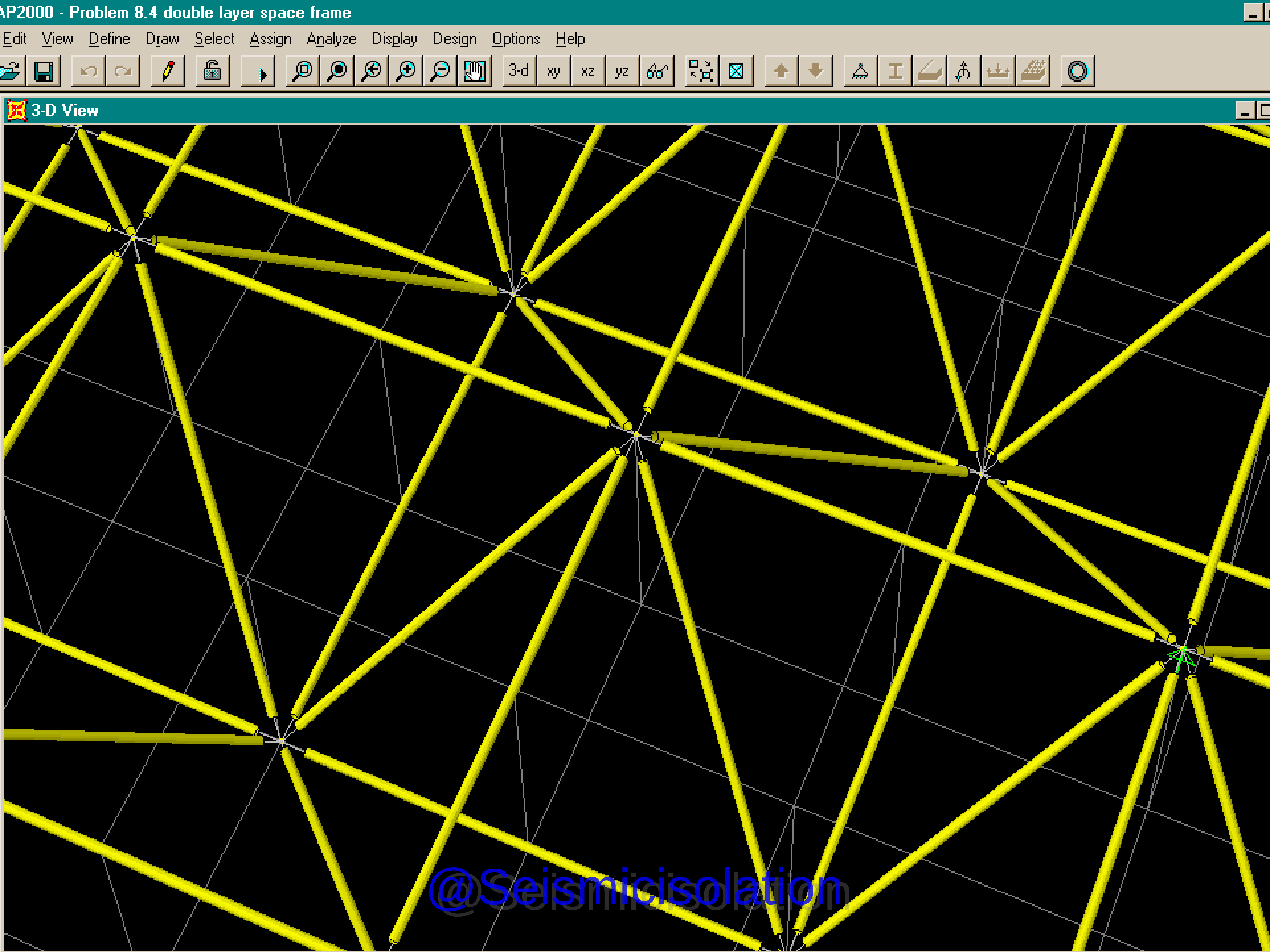


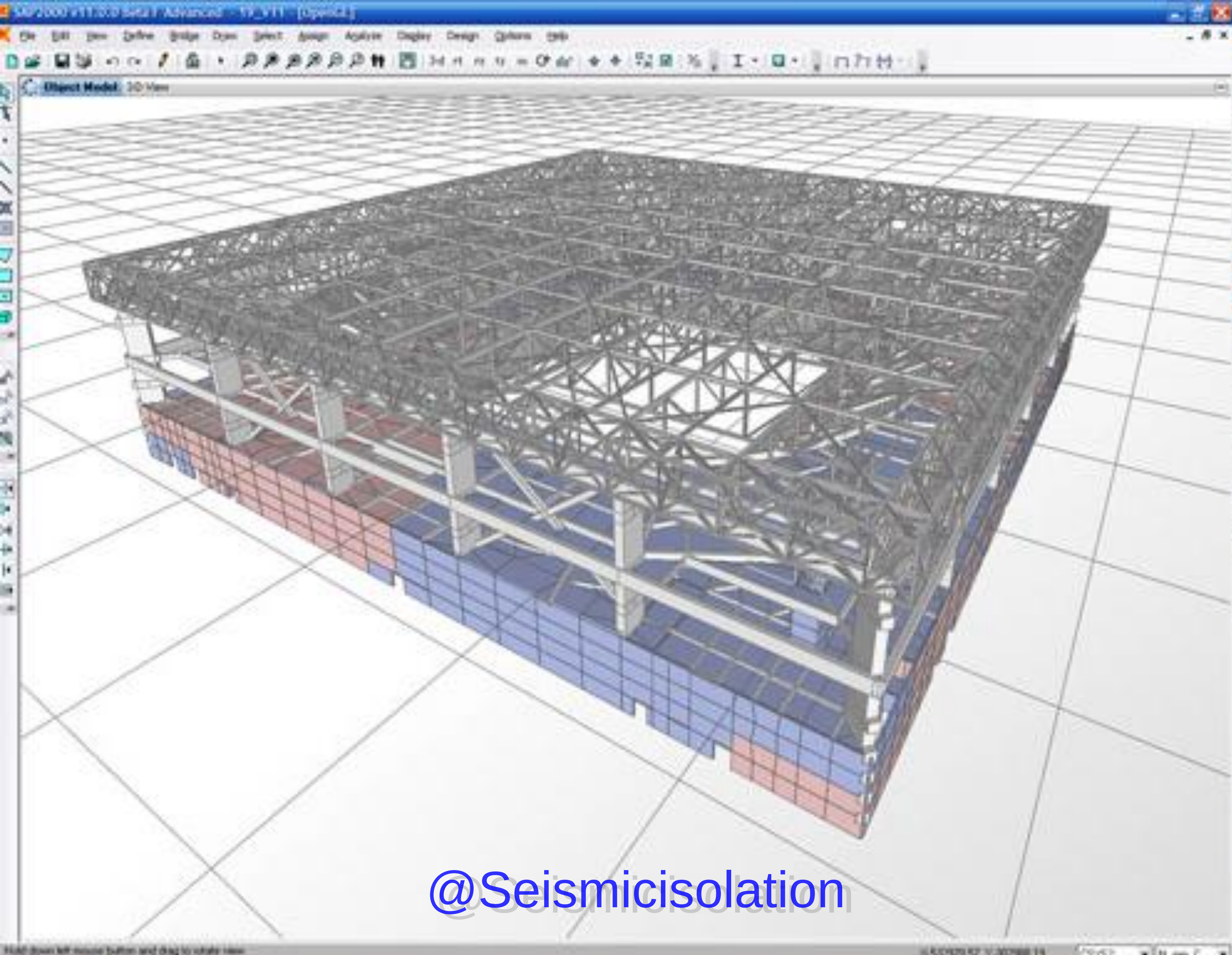

Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
3-D View
Axial Force Diagram (LOAD1)
Axial Force Diagram (LOAD2)
Click on any Frame Element for detailed diagram
KN-m
tart
SAP2000 - Ex.spacef...
Microsoft PowerPoint - [Vis...
10:03 AM

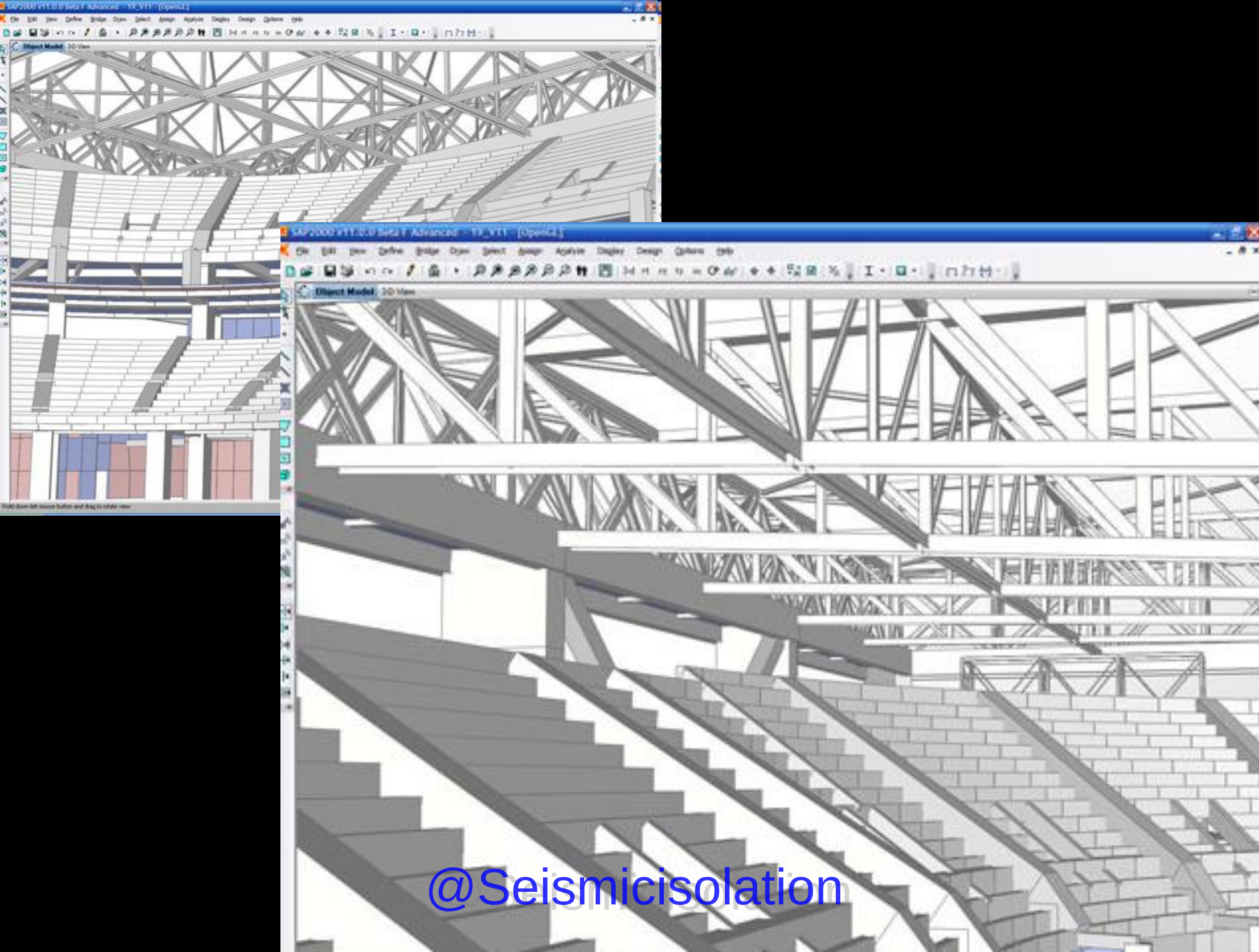
Cologne/Bonn Airport, Germany, 2000, Helmut Jahn Arch., Ove Arup USA Struct. Eng.

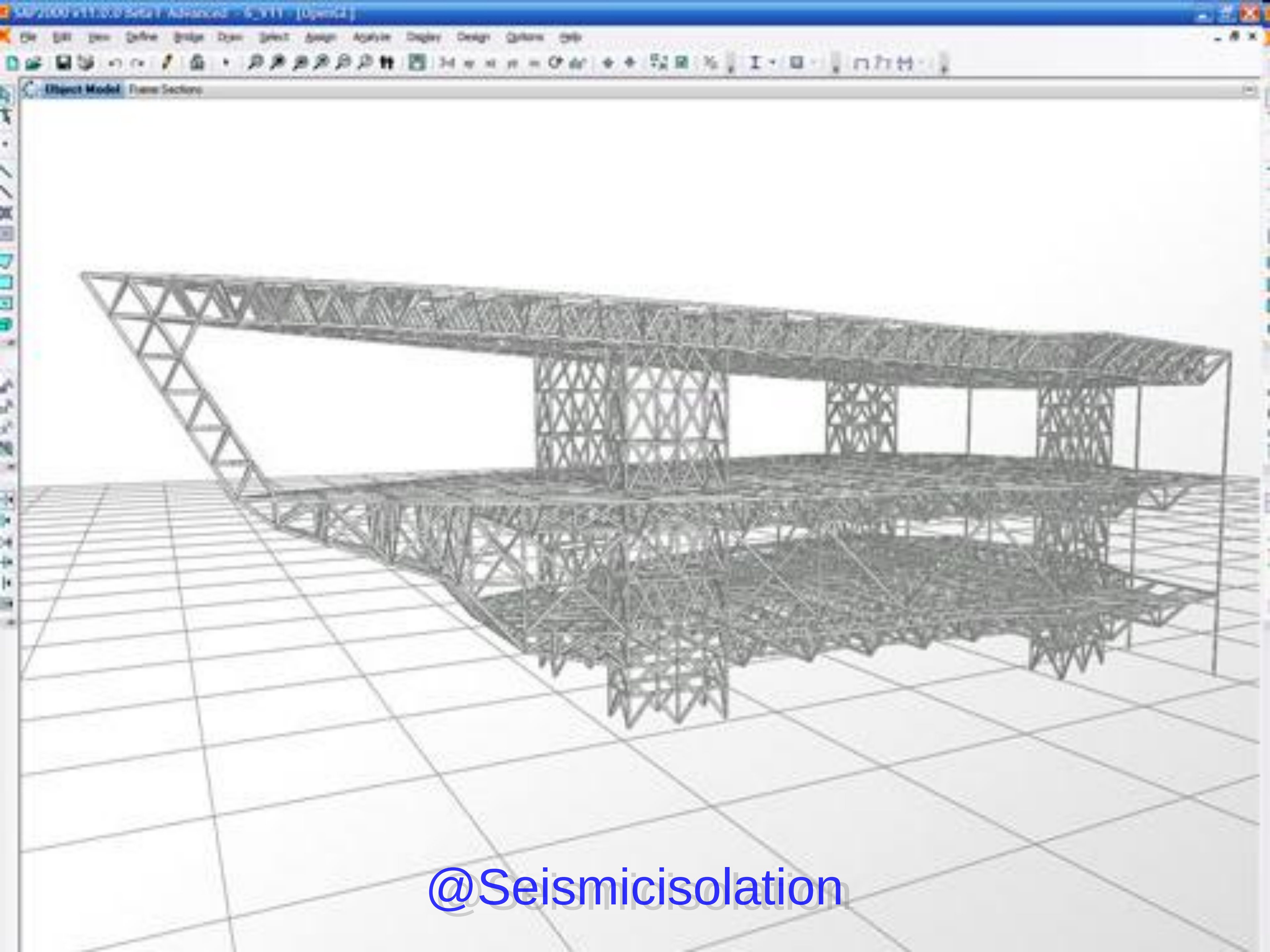

MARKT
15 8 2001

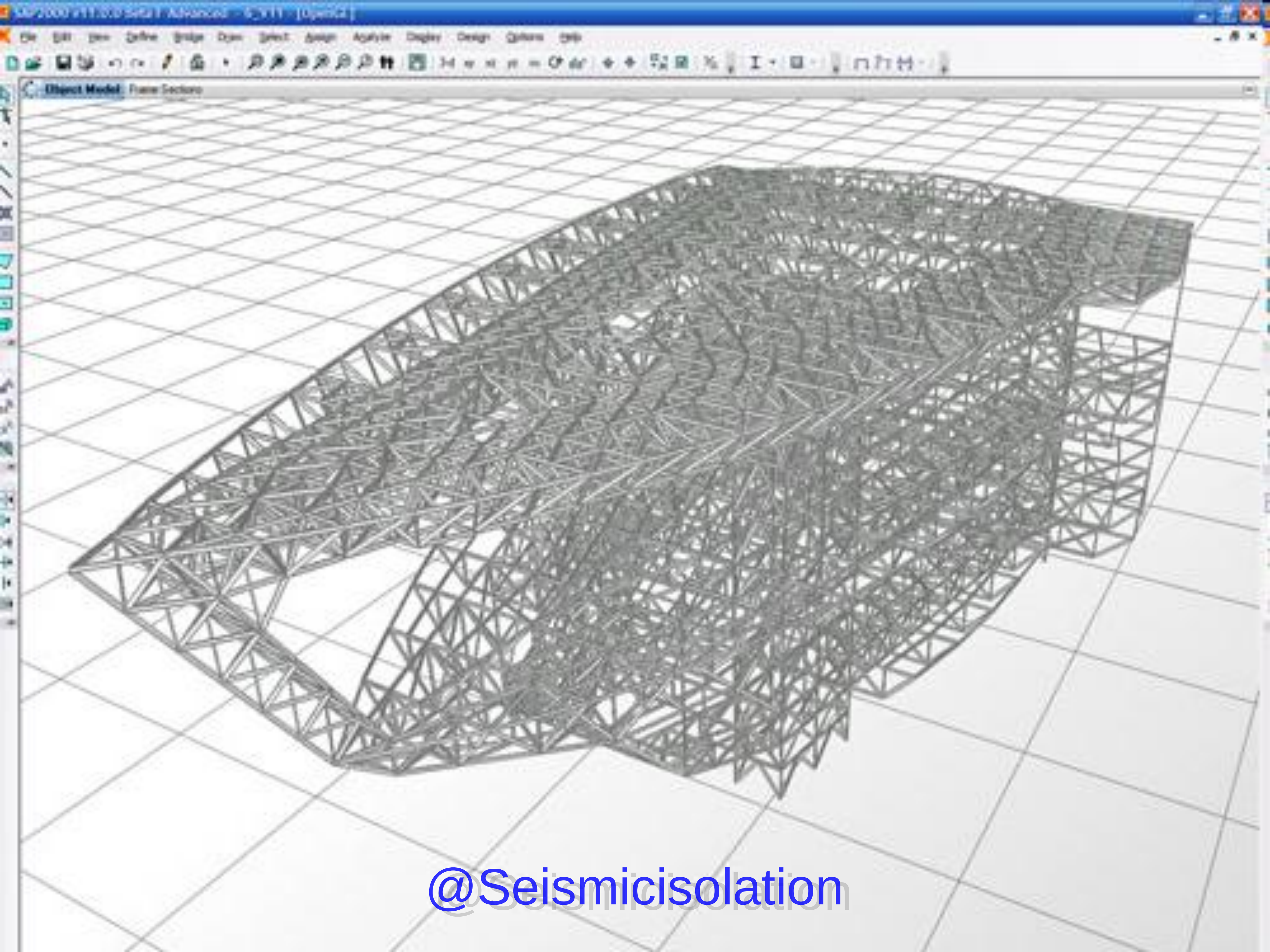

AP2000 - Problem 8.4 double layer space frame
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
3-D View
3-D View
Axial Force Diagram (LOAD1)
Axial Force Diagram (LOAD1)

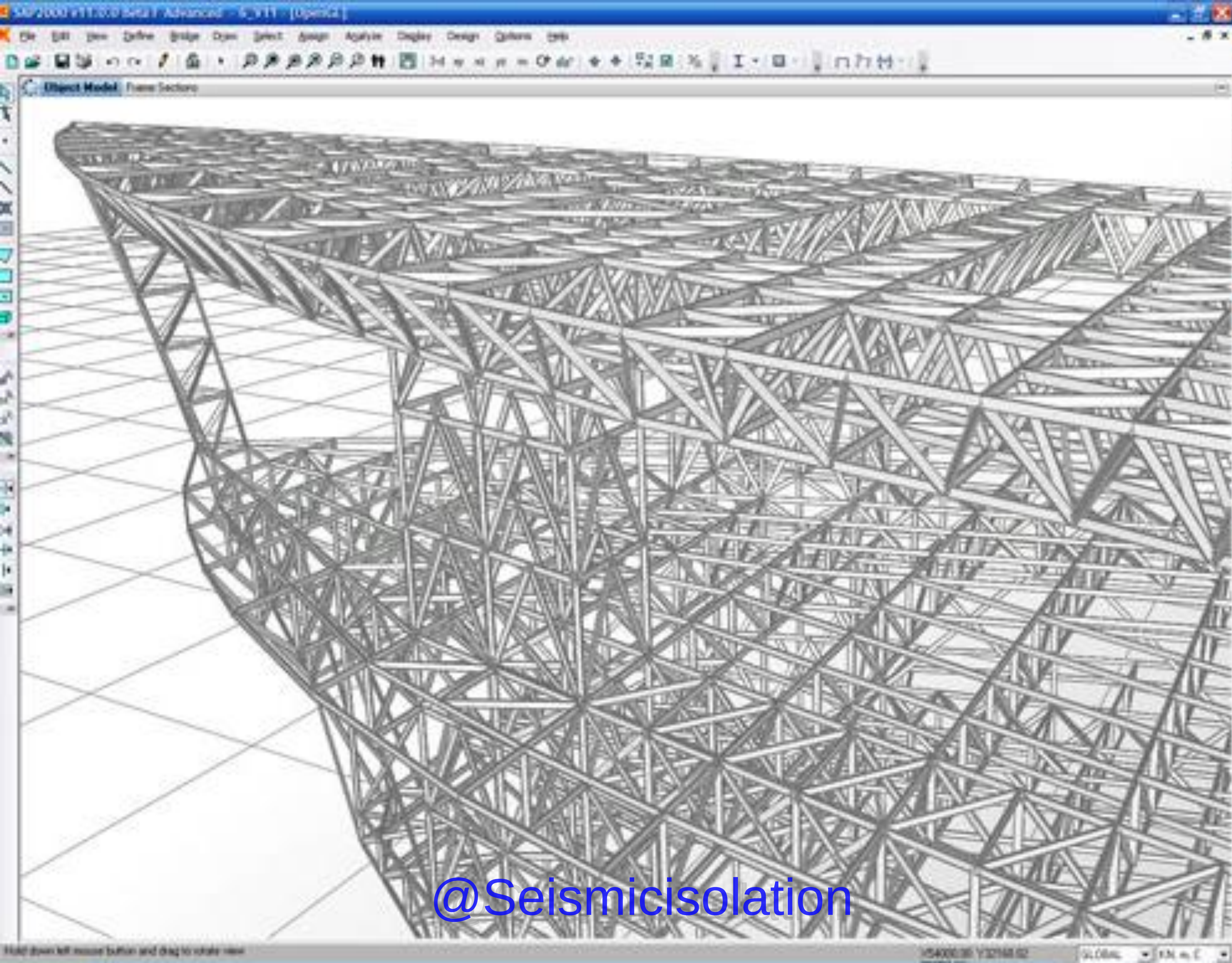

AP2000 - Problem 8.4 double layer space frame
Edit
View
Define
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
3-D View

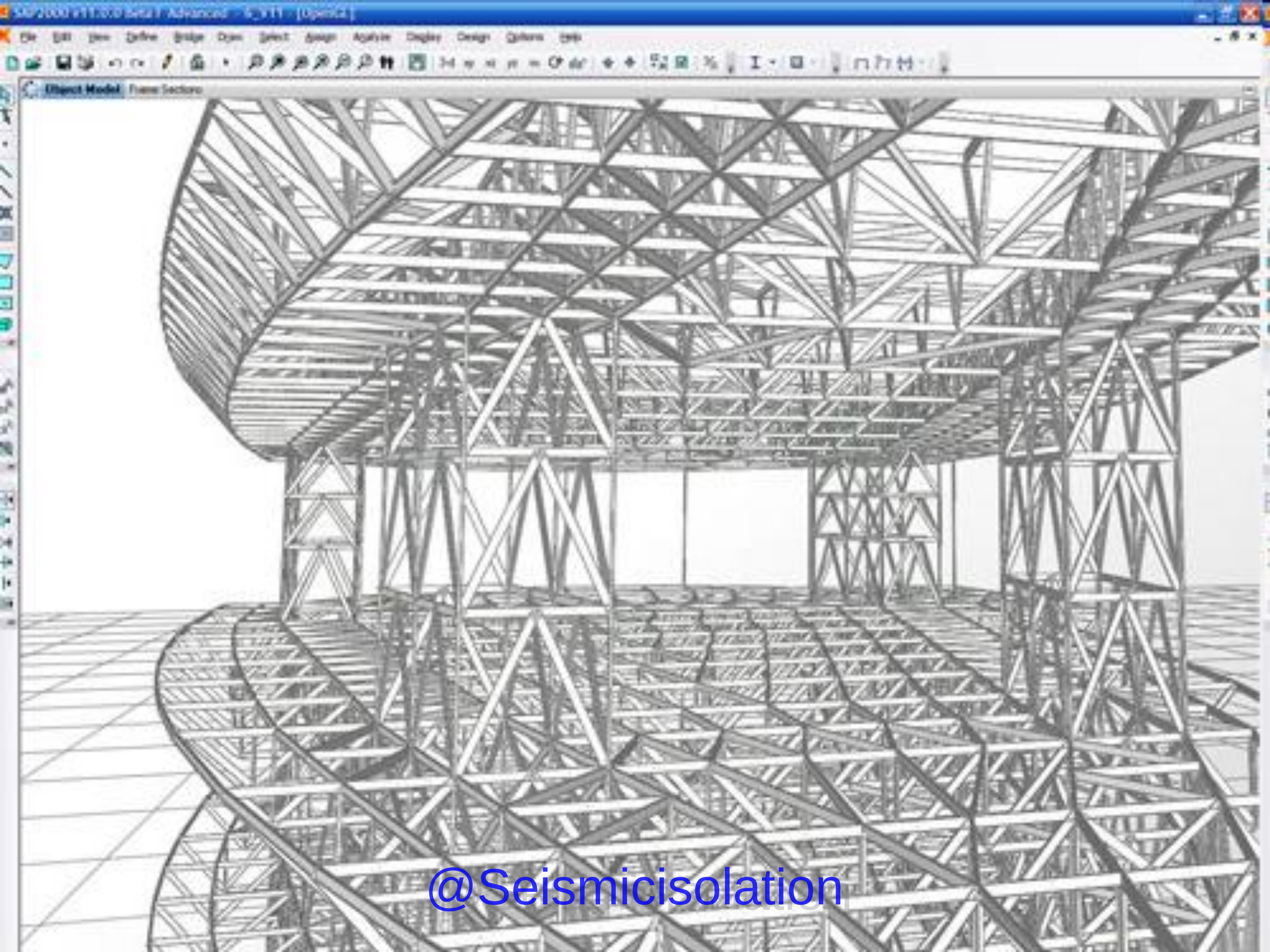

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3D View

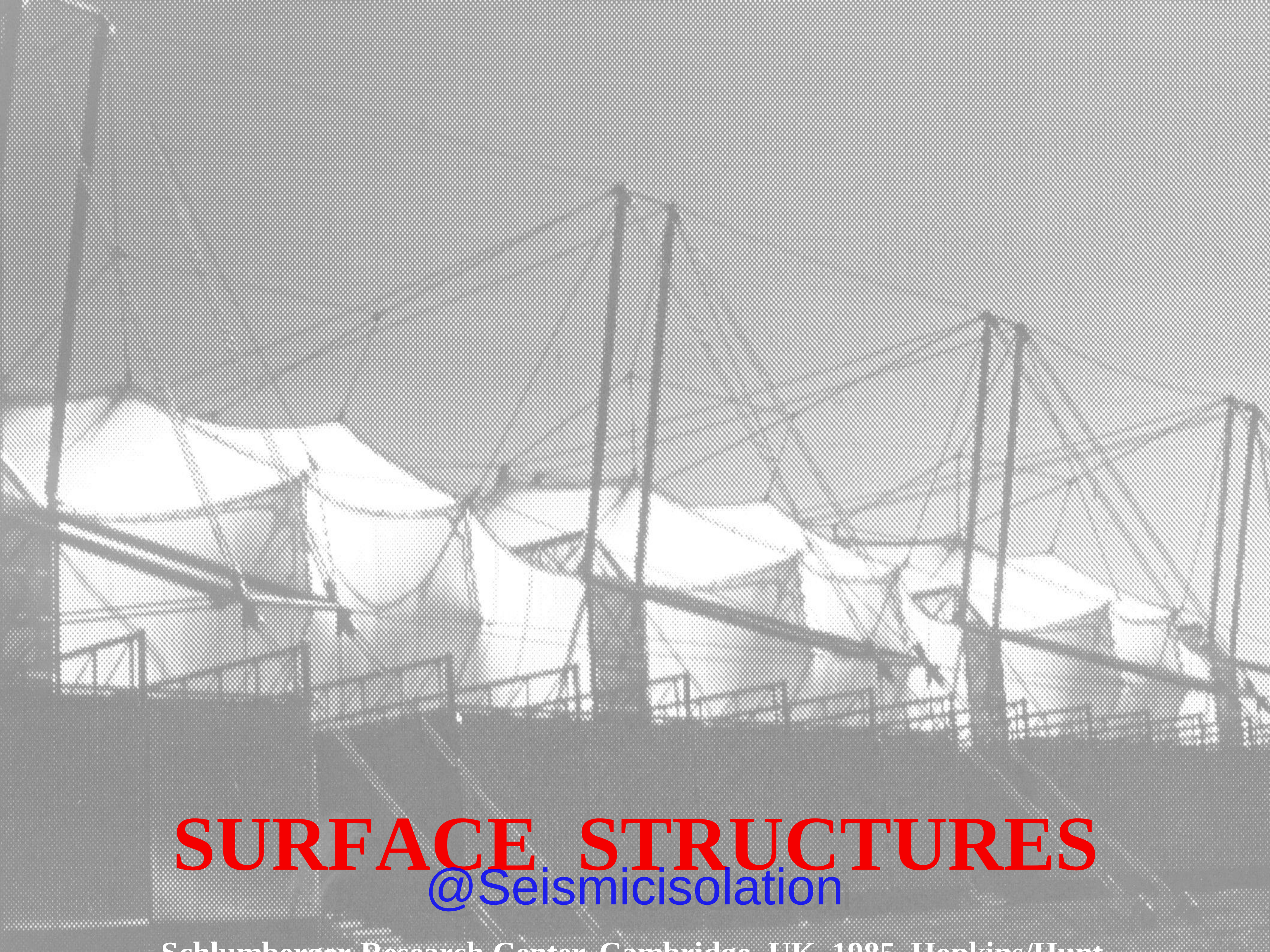

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model
3-D View




File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model Frame Sections


Object Model
Frame Sections





SURFACE STRUCTURES

Schlumberger Research Center, Cambridge, UK, 1985, Hopkins/Hunt

MEMBRANES

BEAMS

BEARING WALLS and SHEAR WALLS

FOLDED SURFACES

RIBBED VAULTING

LINEAR and RADIAL ADDITIONS

parallel, triangular, and tapered folds

CURVILINEAR FOLDS

SHELLS: solid shells, grid shells

CYLINDRICAL SHELLS

THIN SHELL DOMES

HYPERBOLIC PARABOLOIDS

TENSILE MEMBRANE STUCTURES

Pneumatic structures

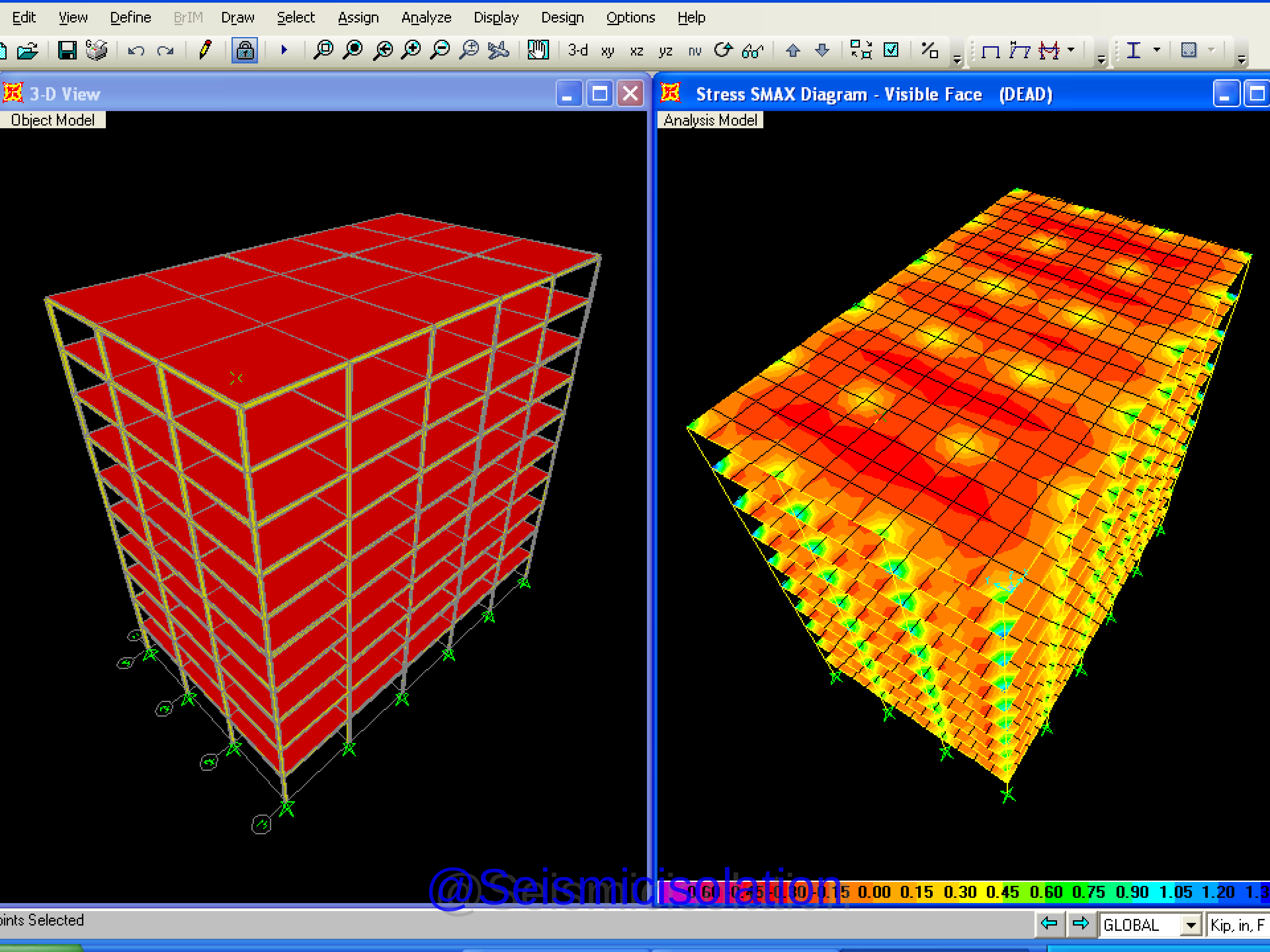
Air-supported structures
Air-inflated structures (i.e. air members)
Hybrid air structures

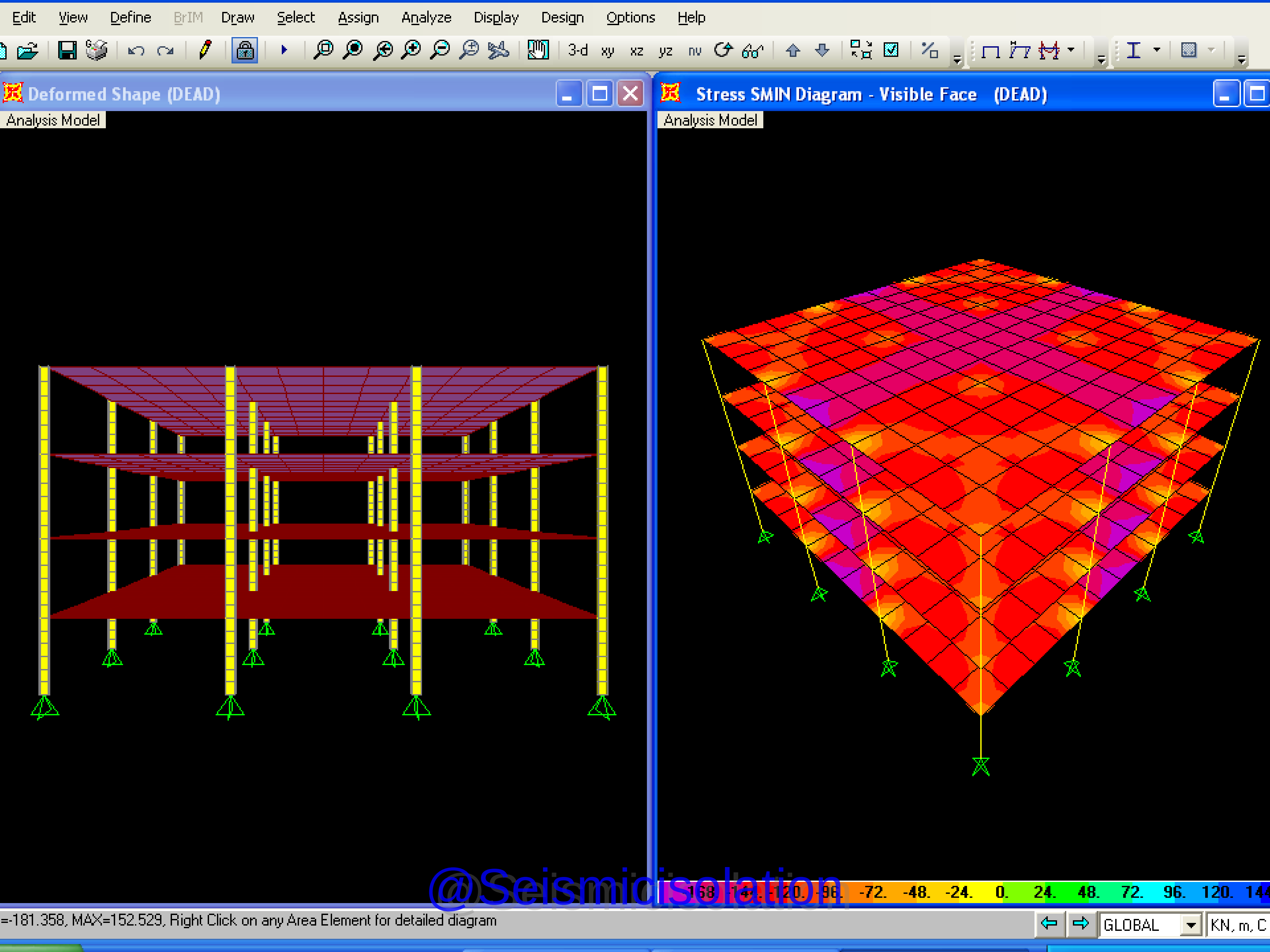
Anticlastic prestressed membrane structures

Edge-supported saddle roofs
Mast-supported conical saddle roofs
Arch-supported saddle roofs

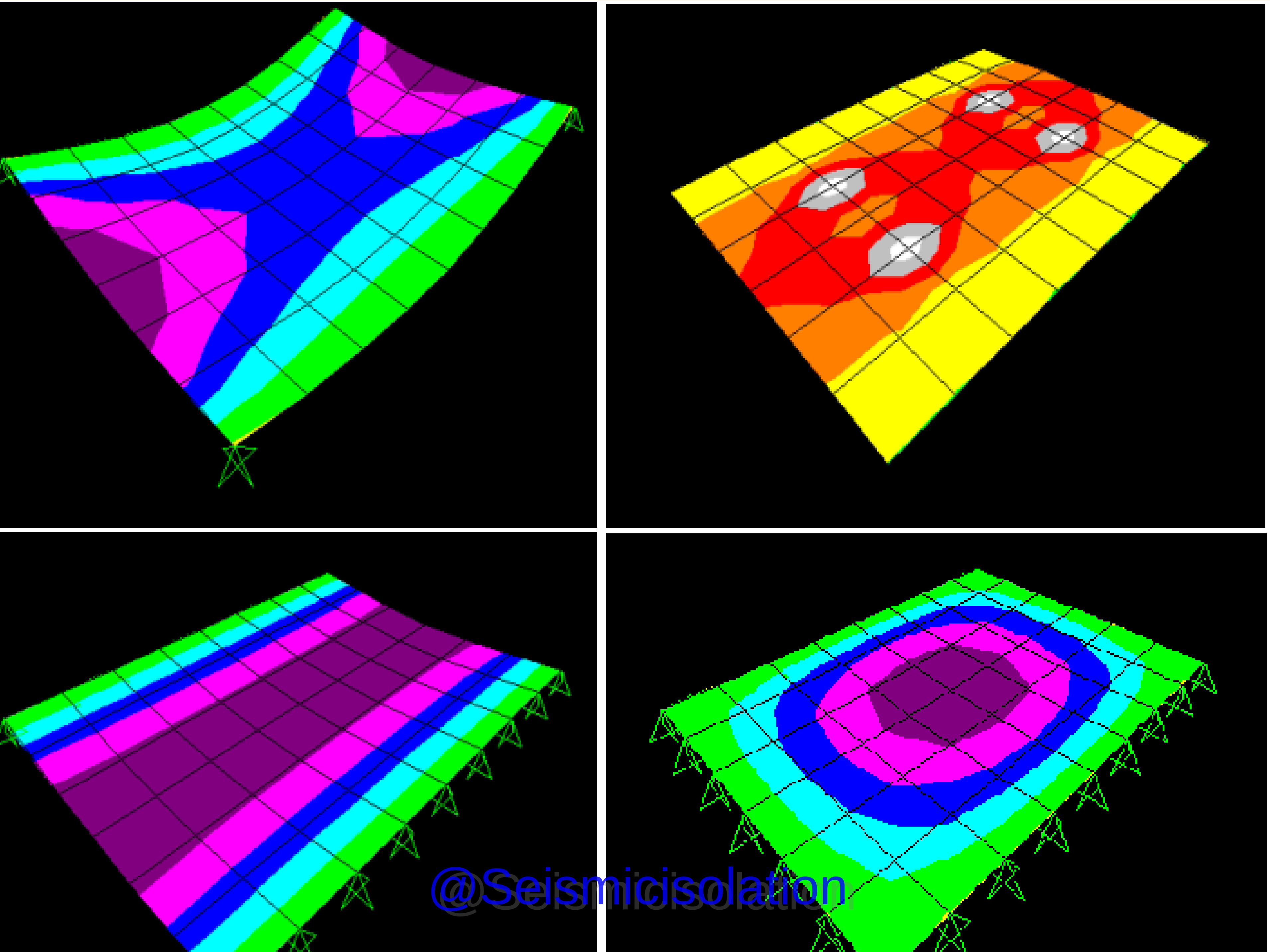
Hybrid tensile surface structures

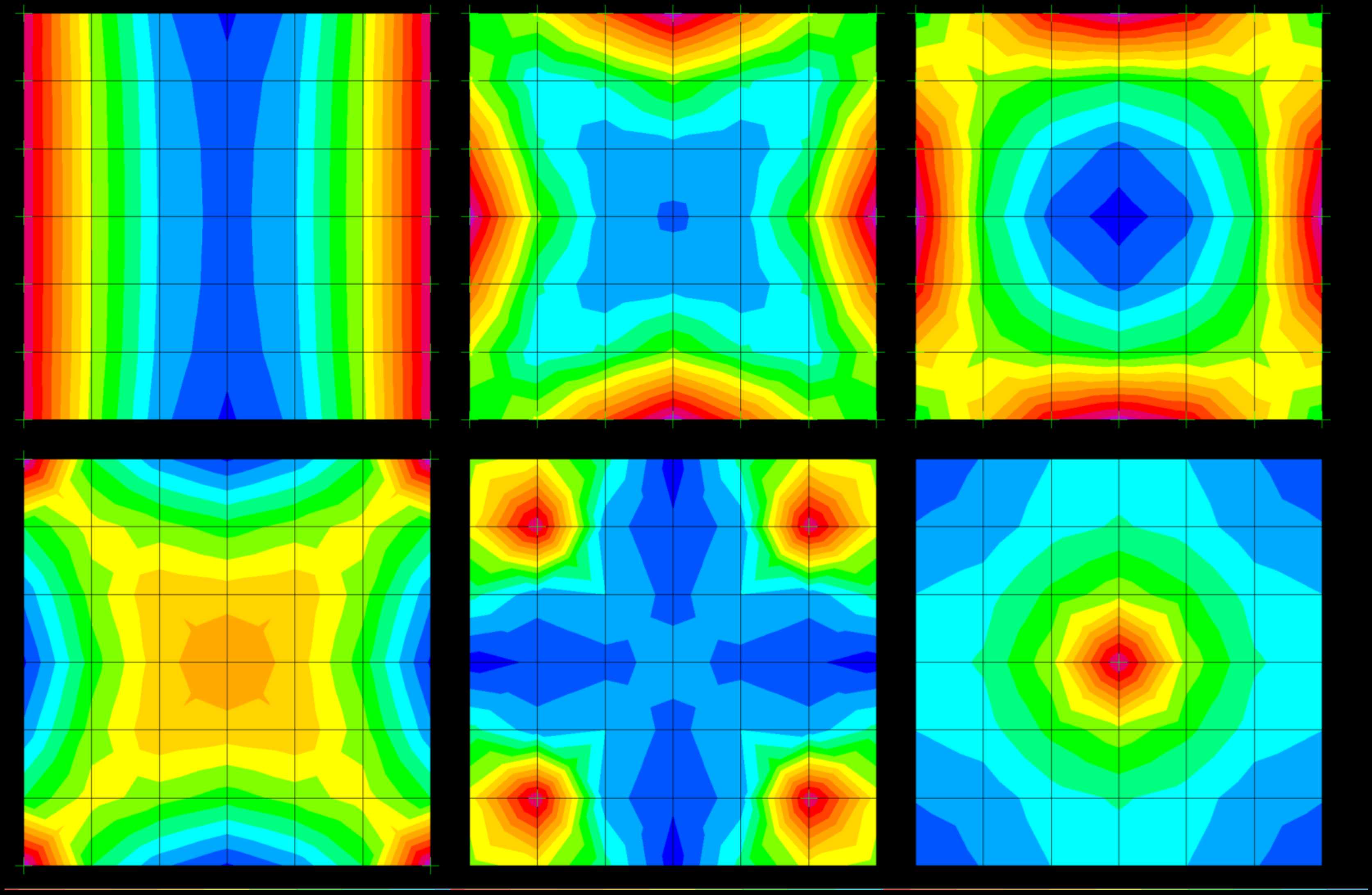
@Seismicisolation

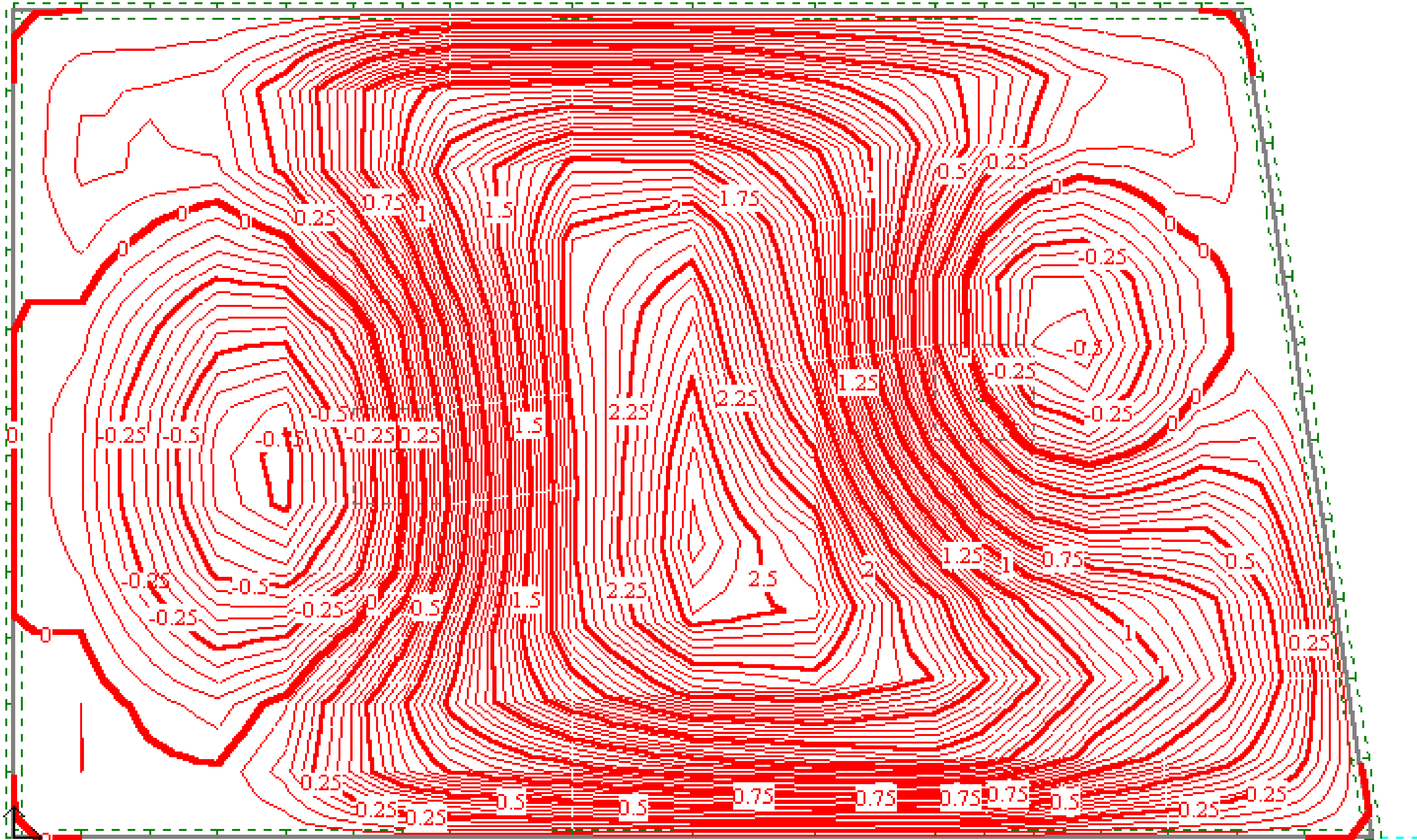

Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
3-D View
Object Model
Stress SMAX Diagram - Visible Face (DEAD)
Analysis Model
0.60 -0.45 -0.30 -0.15 0.00 0.15 0.30 0.45 0.60 0.75 0.90 1.05 1.20
GLOBAL

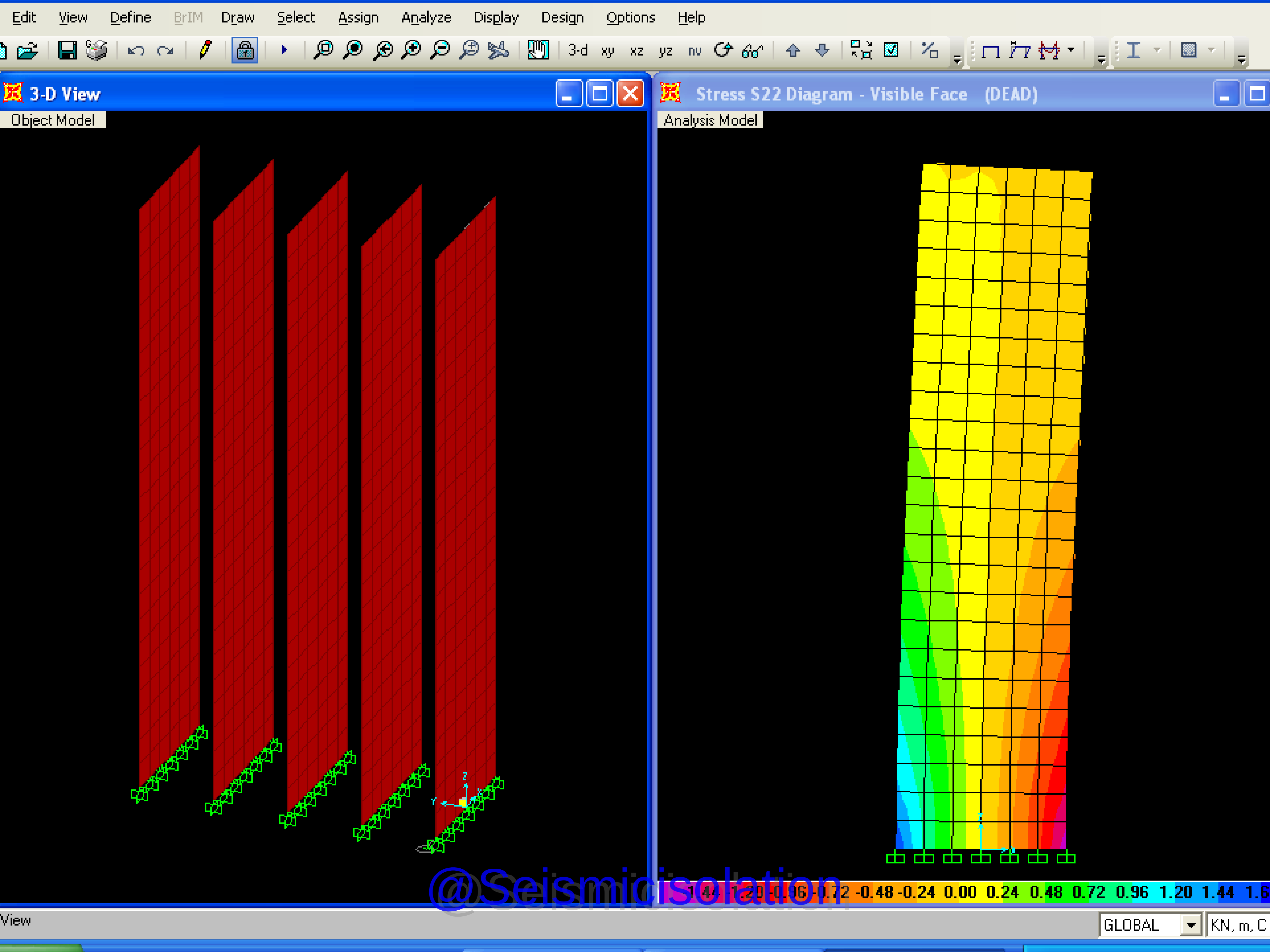


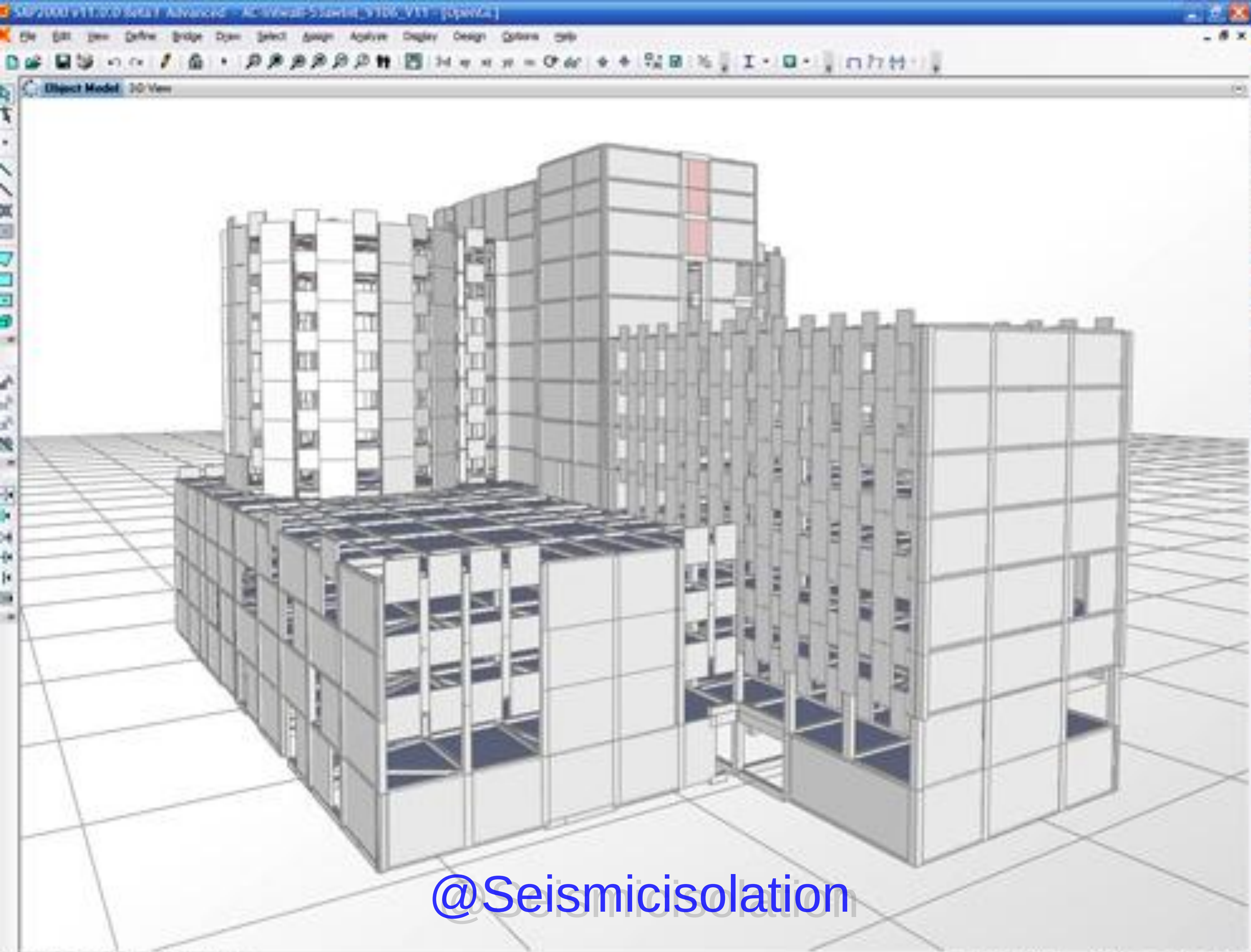
Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
Deformed Shape (DEAD)
Analysis Model
Stress SMIN Diagram - Visible Face (DEAD)
Analysis Model
-72. -48. -24. 0. 24. 48. 72. 96. 120.
=-181.358, MAX=152.529, Right Click on any Area Element for detailed diagram
GLOBAL
KN, m, C
@Seismicisolation

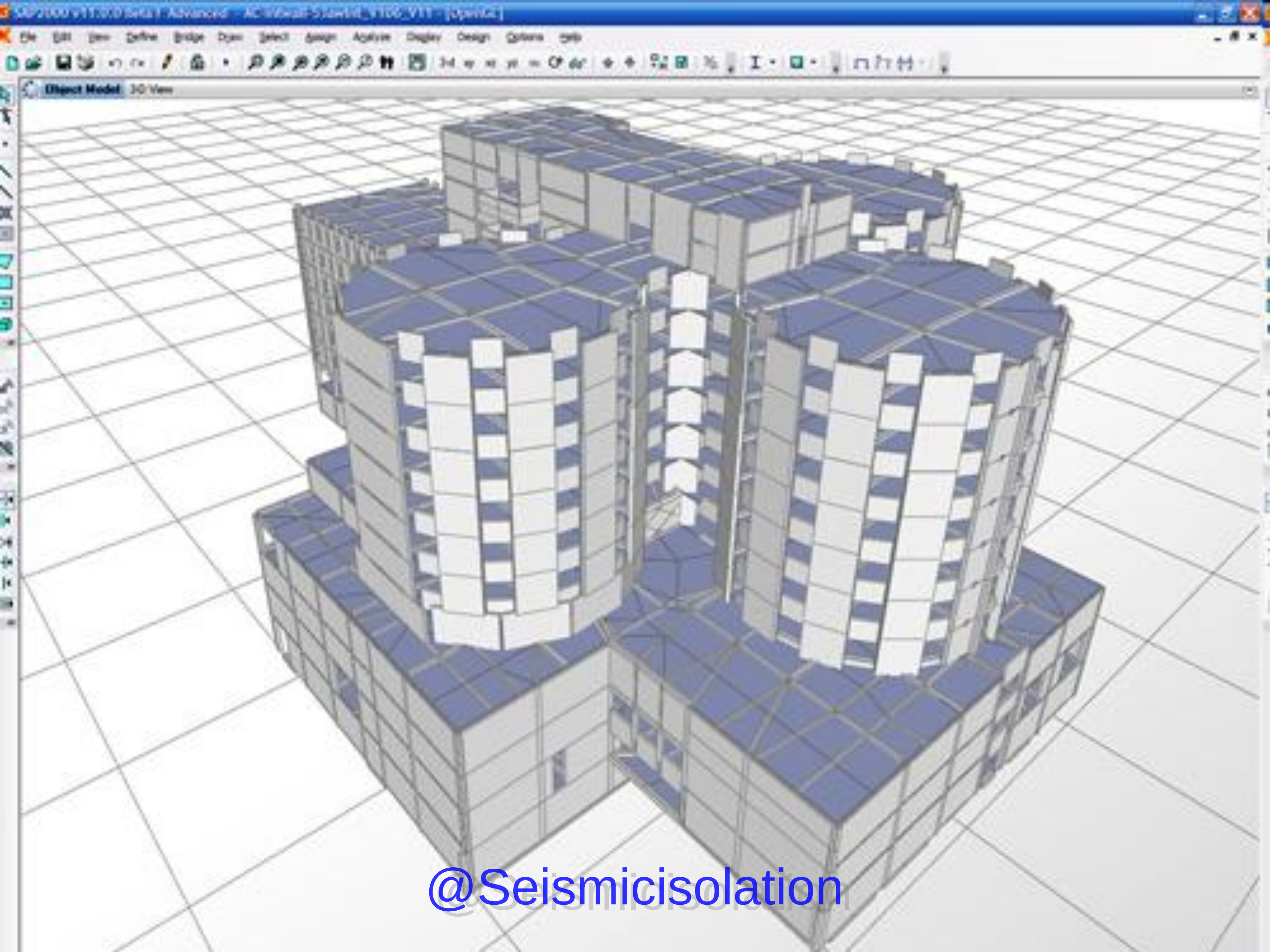


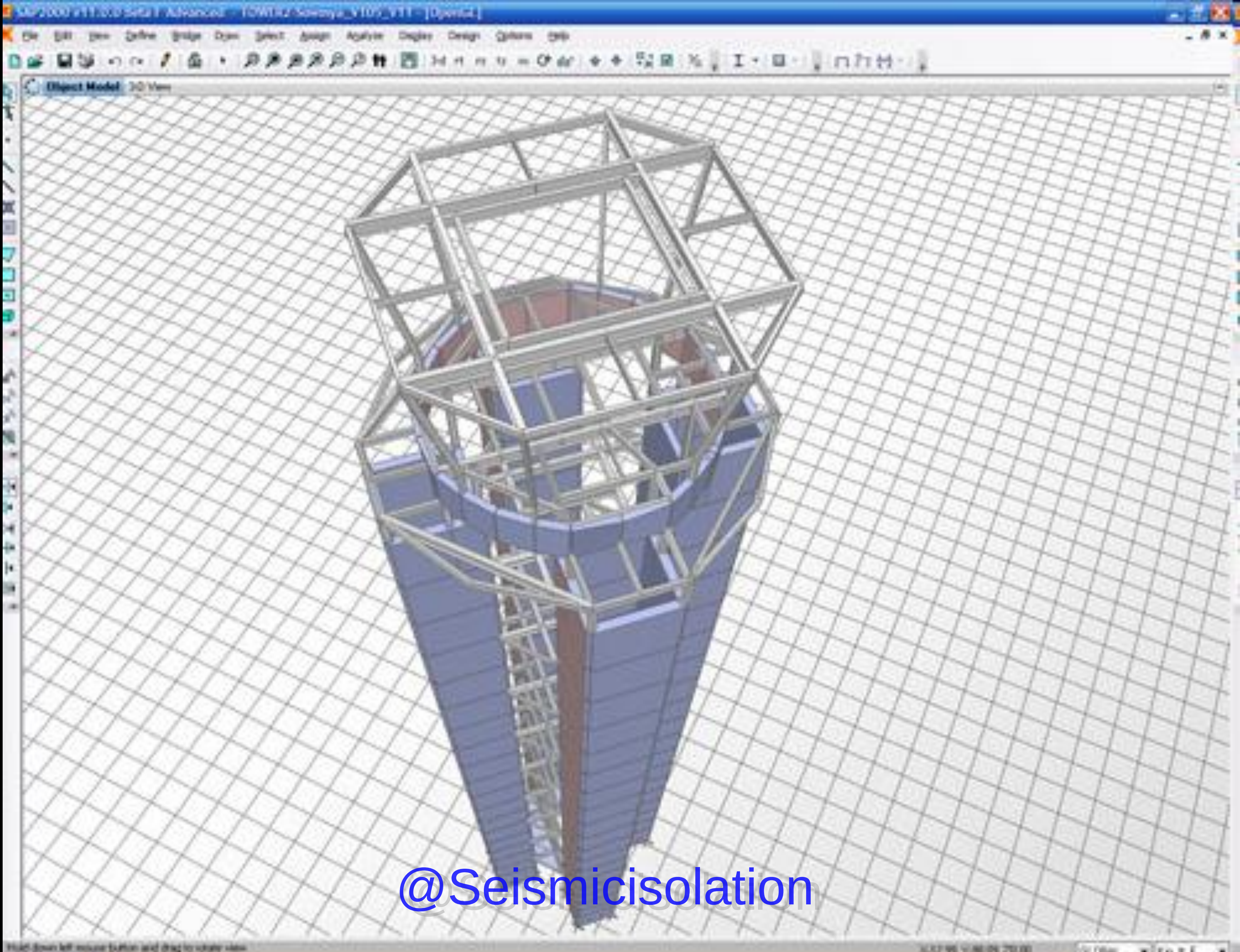


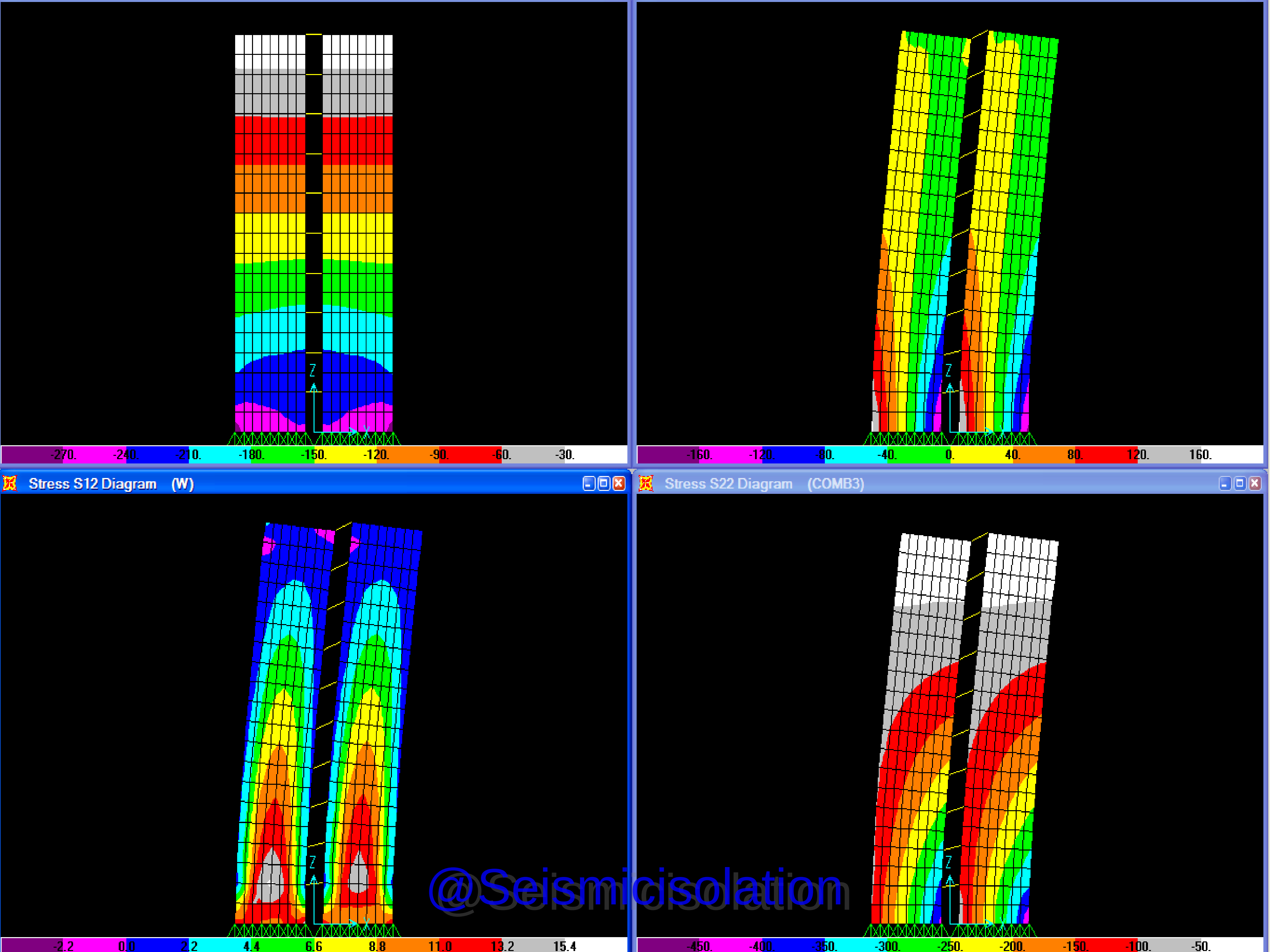

0.25
0.75
1
1.5
2
1.75
0.5
0.25
0
-0.25
-0.5
-0.75
-0.5
-0.25
0.25
1.25
2.25
2.5
0.75
0.5
0.25


Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
3-D View
Object Model
Stress S22 Diagram - Visible Face (DEAD)
Analysis Model
-1.44 -1.20 -0.96 -0.72 -0.48 -0.24 0.00 0.24 0.48 0.72 0.96 1.20 1.44 1.6
View
GLOBAL
KN, m, C

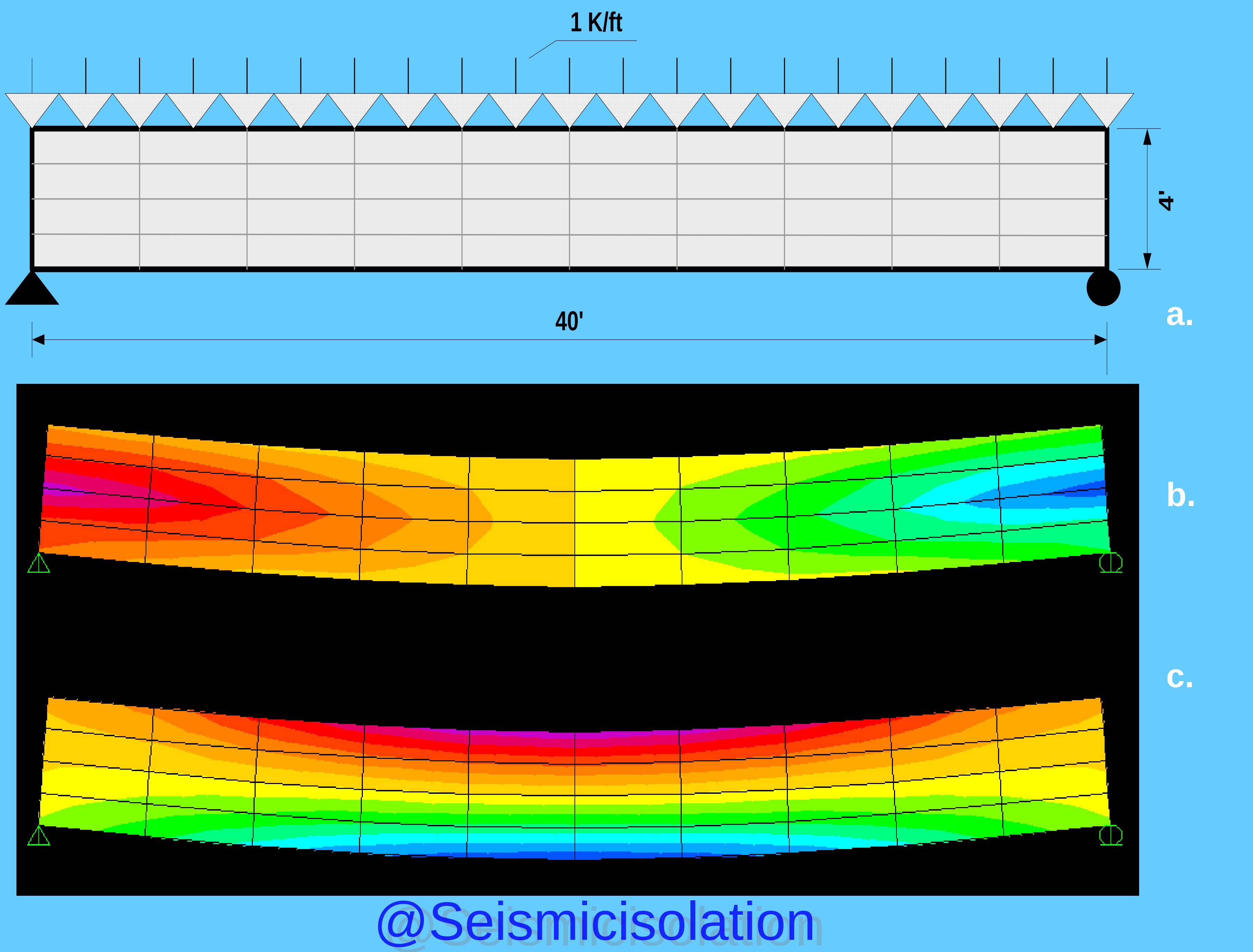

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3-D View


File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model 3-D View

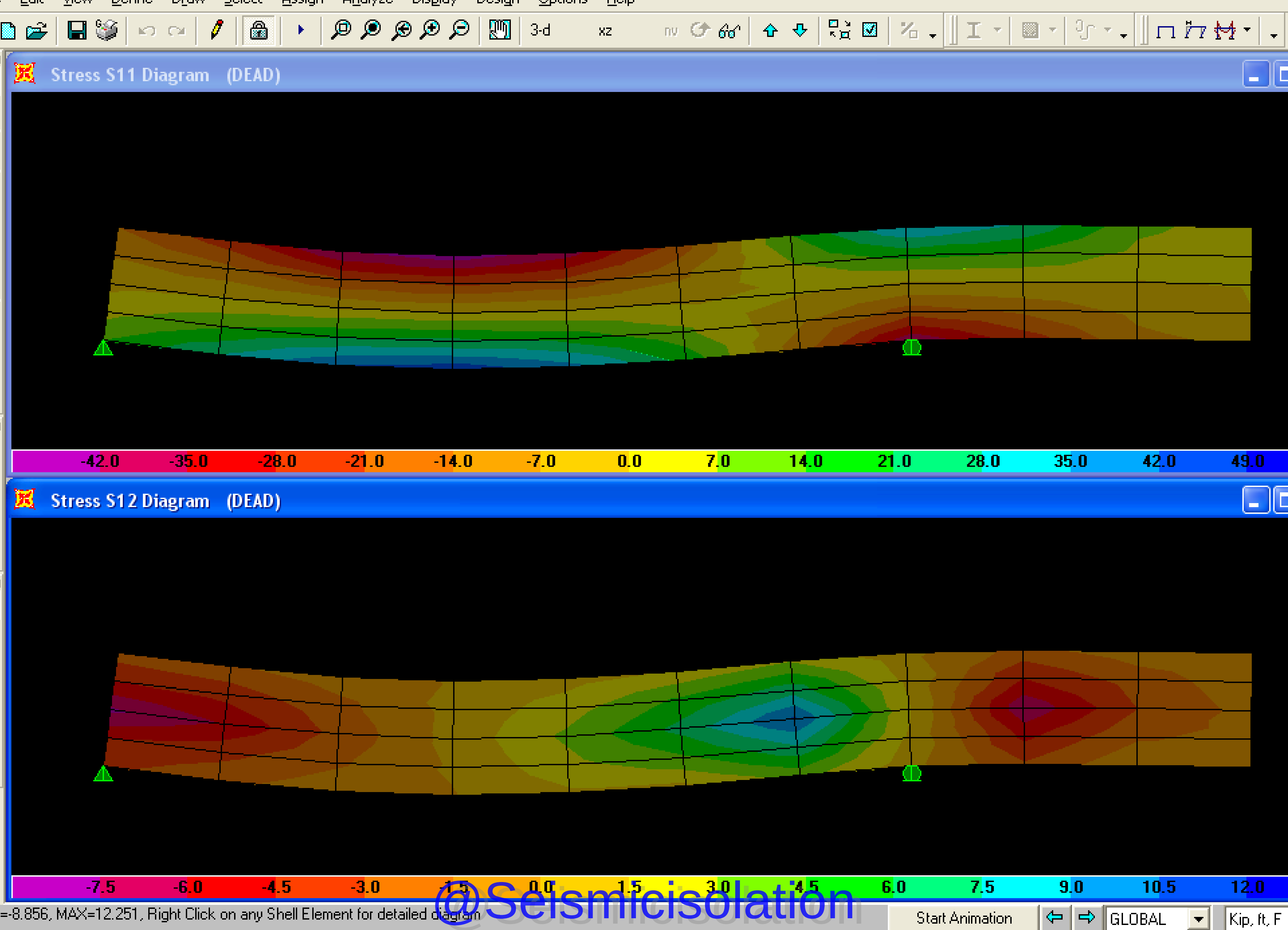

Object Model



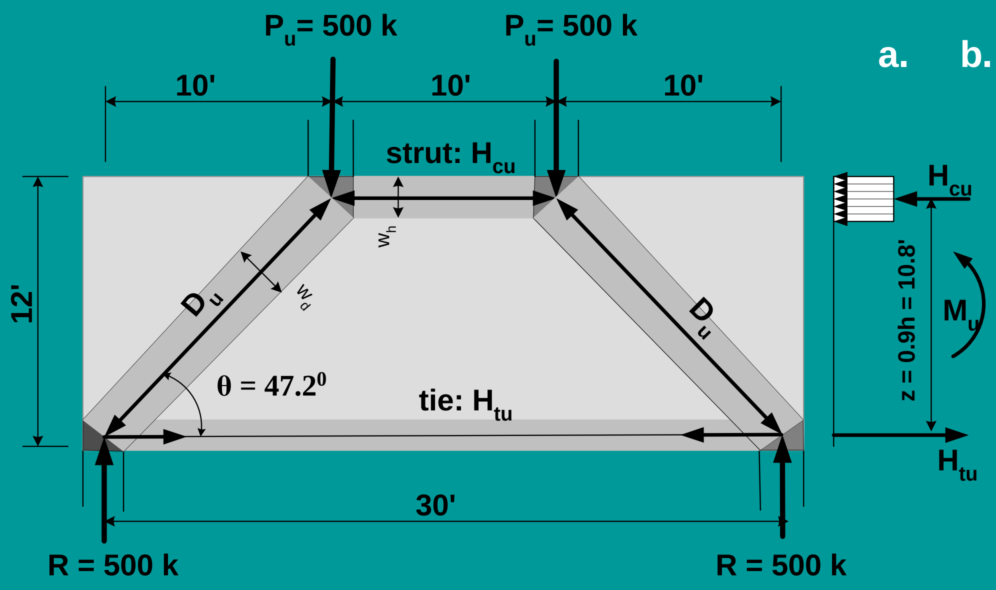
-270. -240. -210. -180. -150. -120. -90. -60. -30.
-160. -120. -80. -40. 0. 40. 80. 120. 160.
Stress S12 Diagram (W)
Stress S22 Diagram (COMB3)
-2.2 0.0 2.2 4.4 6.6 8.8 11.0 13.2 15.4
-450. -400. -350. -300. -250. -200. -150. -100. -50.



1 K/ft
4'
40'
a.
b.
c.
@Seismicisolation

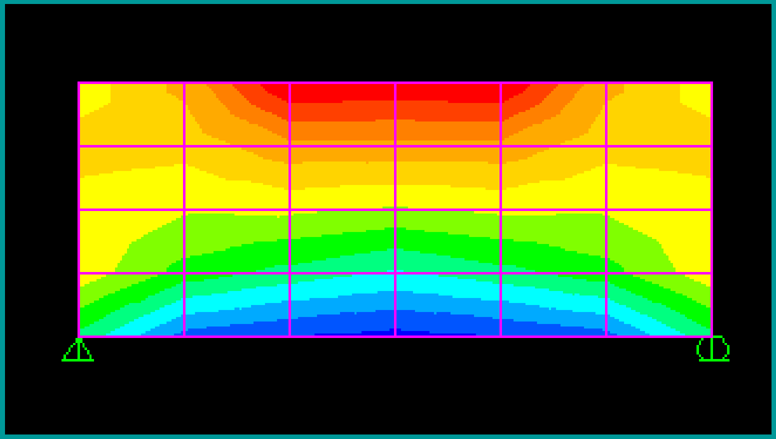


Stress S11 Diagram (DEAD)
-42.0 -35.0 -28.0 -21.0 -14.0 -7.0 0.0 7.0 14.0 21.0 28.0 35.0 42.0 49.0
Stress S12 Diagram (DEAD)
-7.5 -6.0 -4.5 -3.0 -1.5 0.0 1.5 3.0 4.5 6.0 7.5 9.0 10.5 12.0
=-8.856, MAX=12.251, Right Click on any Shell Element for detailed diagram
Start Animation
GLOBAL
Kip, ft, F
@Seismicisolation

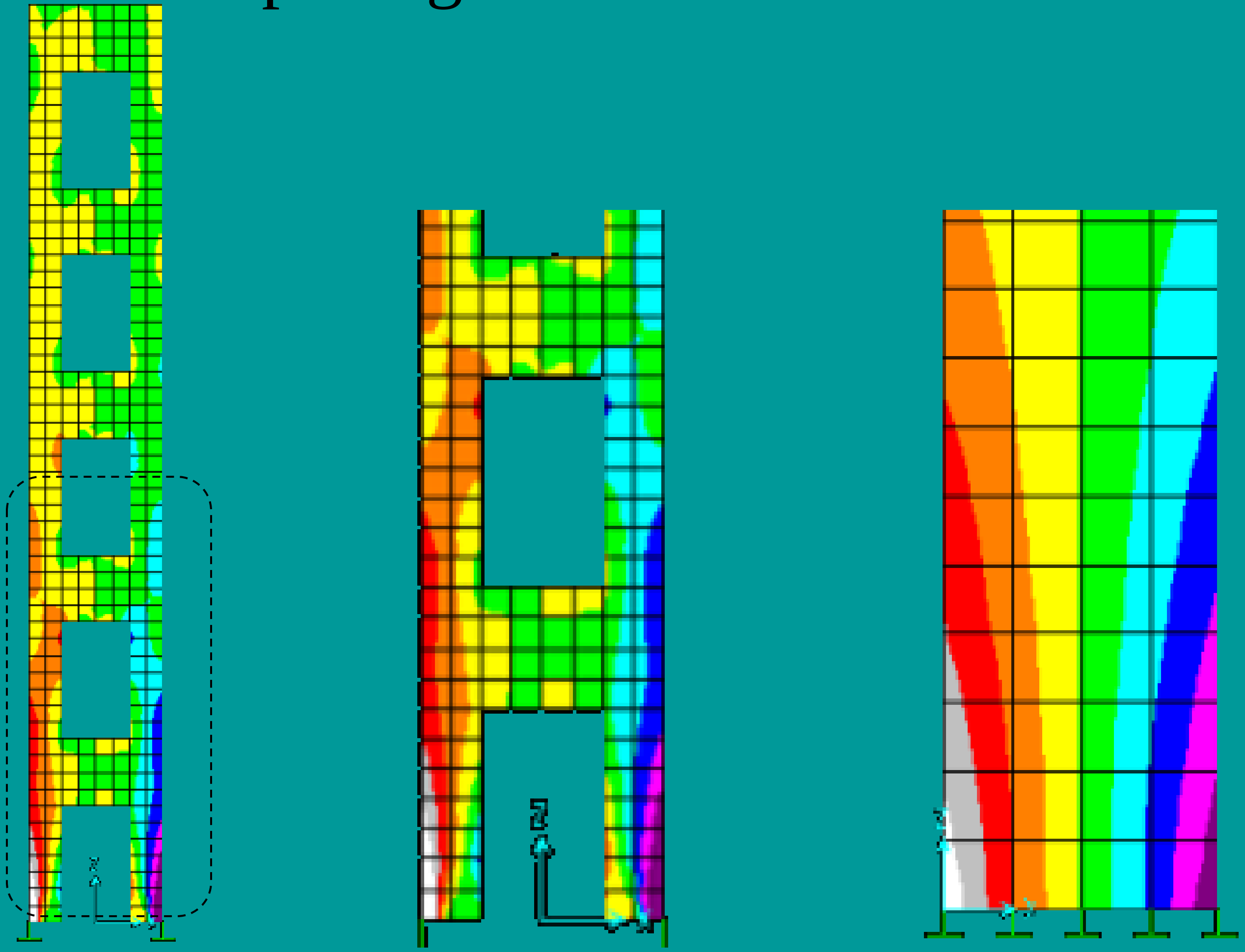


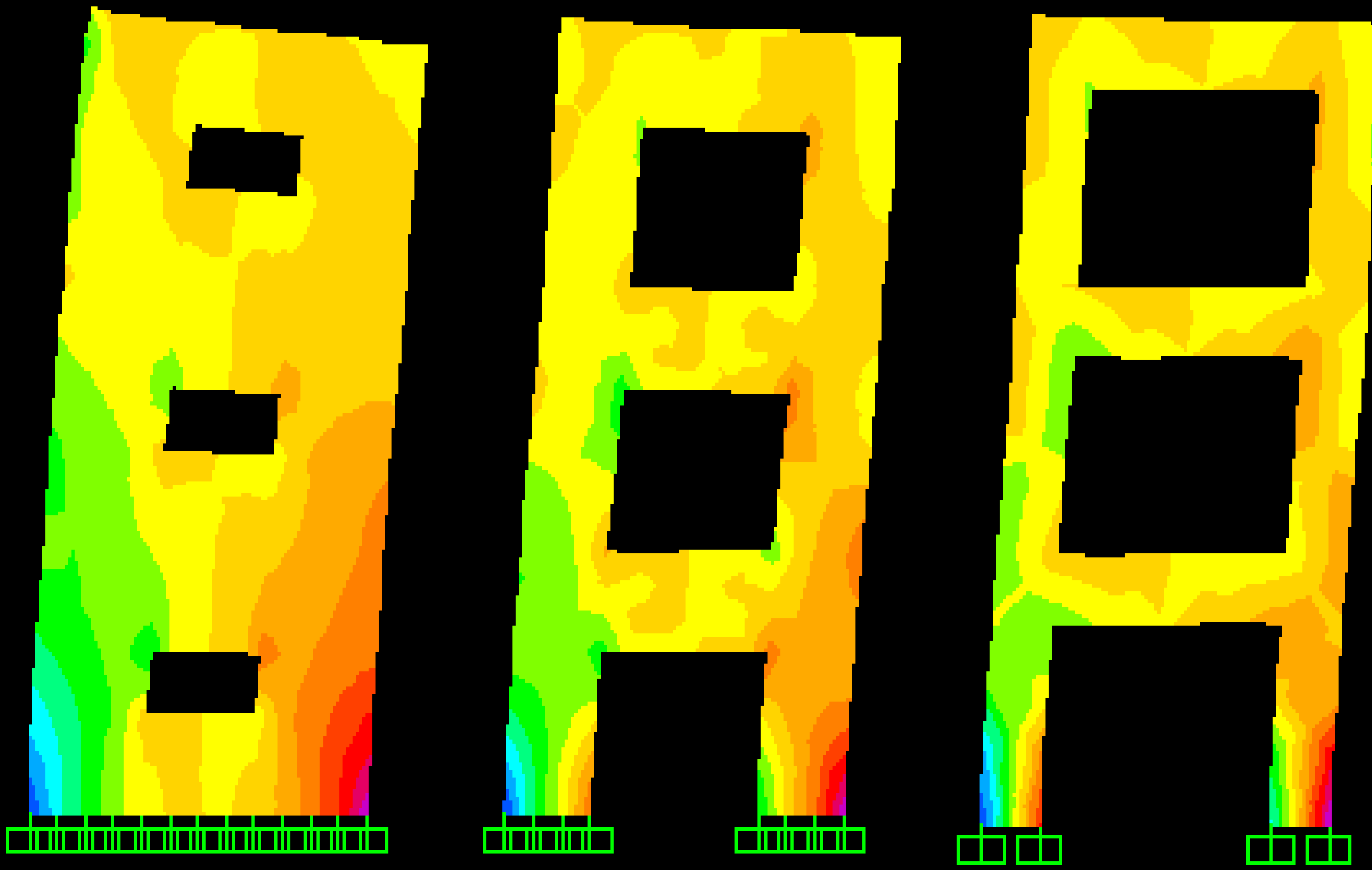
Pu= 500 k
Pu= 500 k
10'
10'
10'
strut: Hcu
12'
Du
wh
wd
Du
θ = 47.2⁰
tie: Htu
30'
R = 500 k
R = 500 k
Hcu
z = 0.9h = 10.8'
Mu
Htu
a.

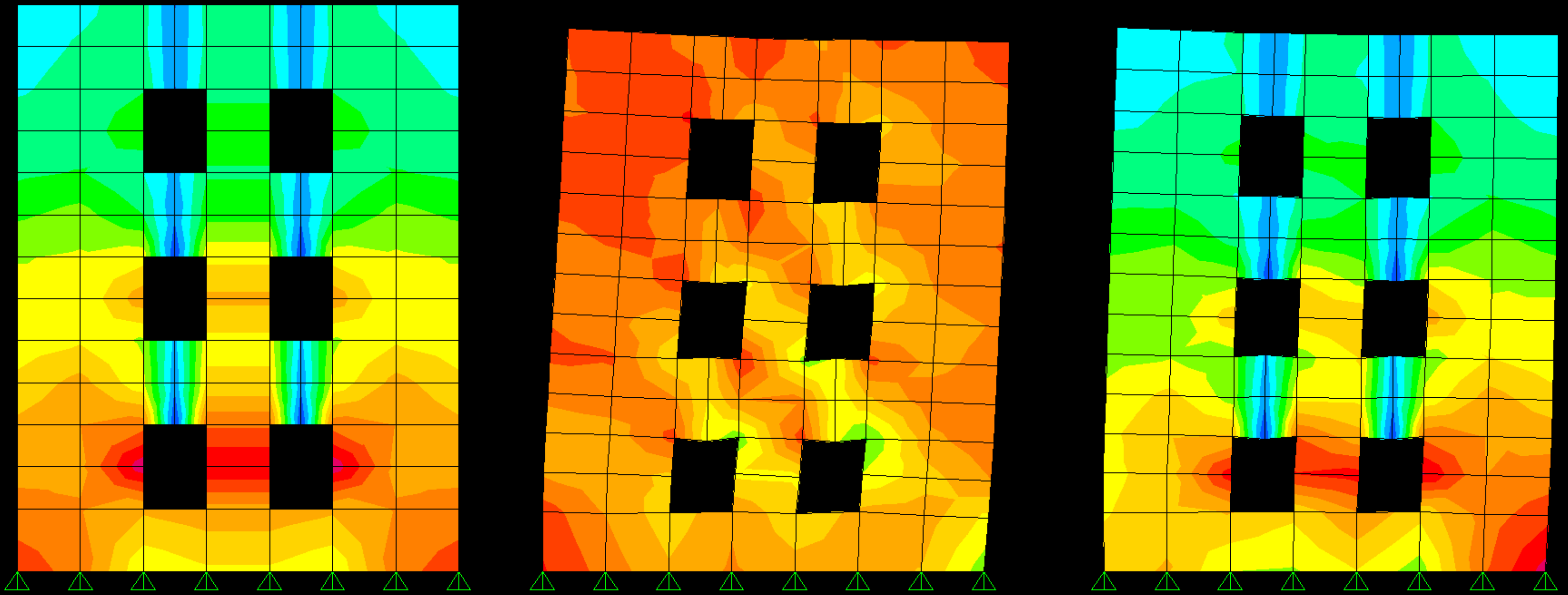
b.



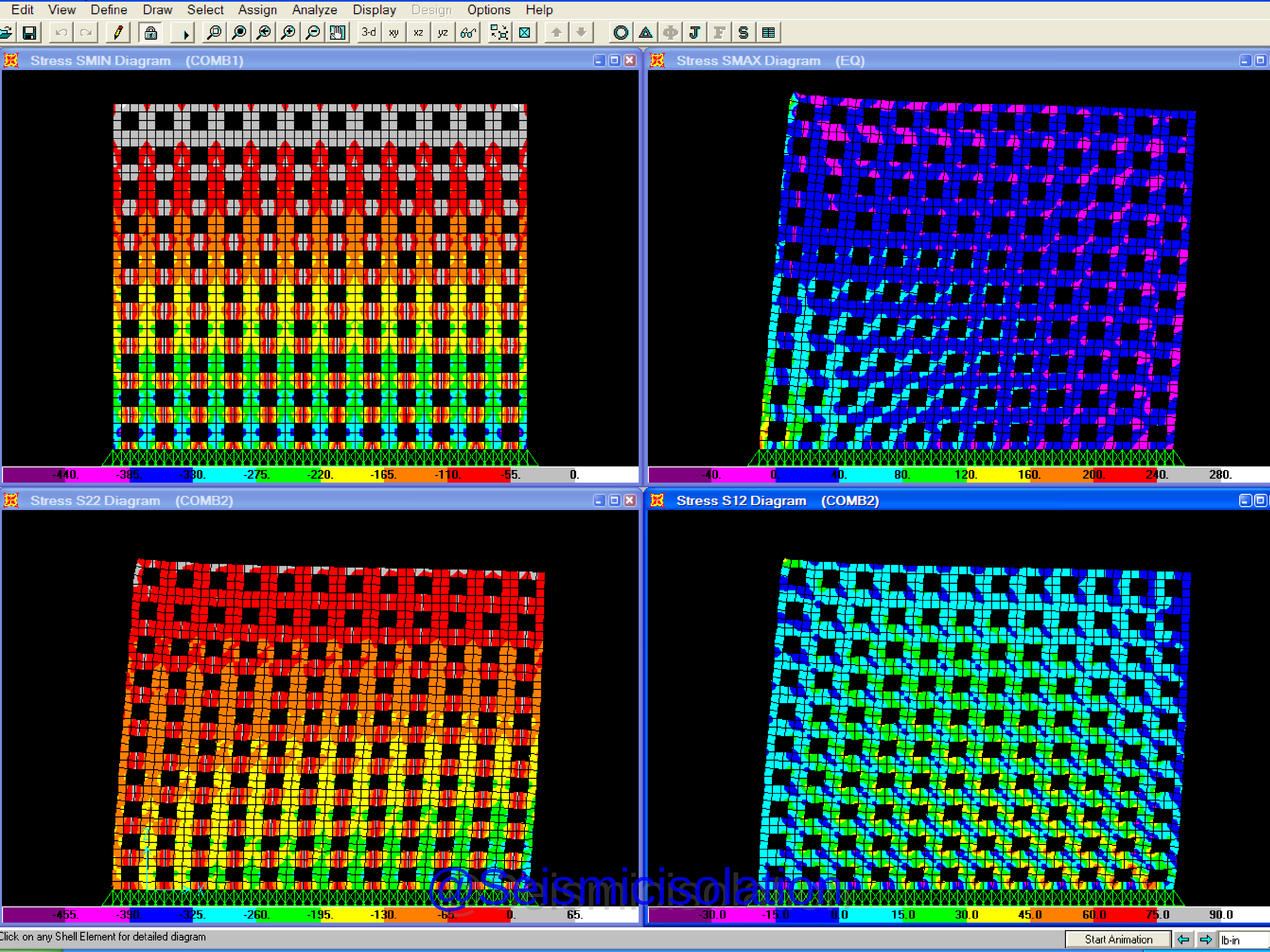
Openings in Shear Walls - Planer

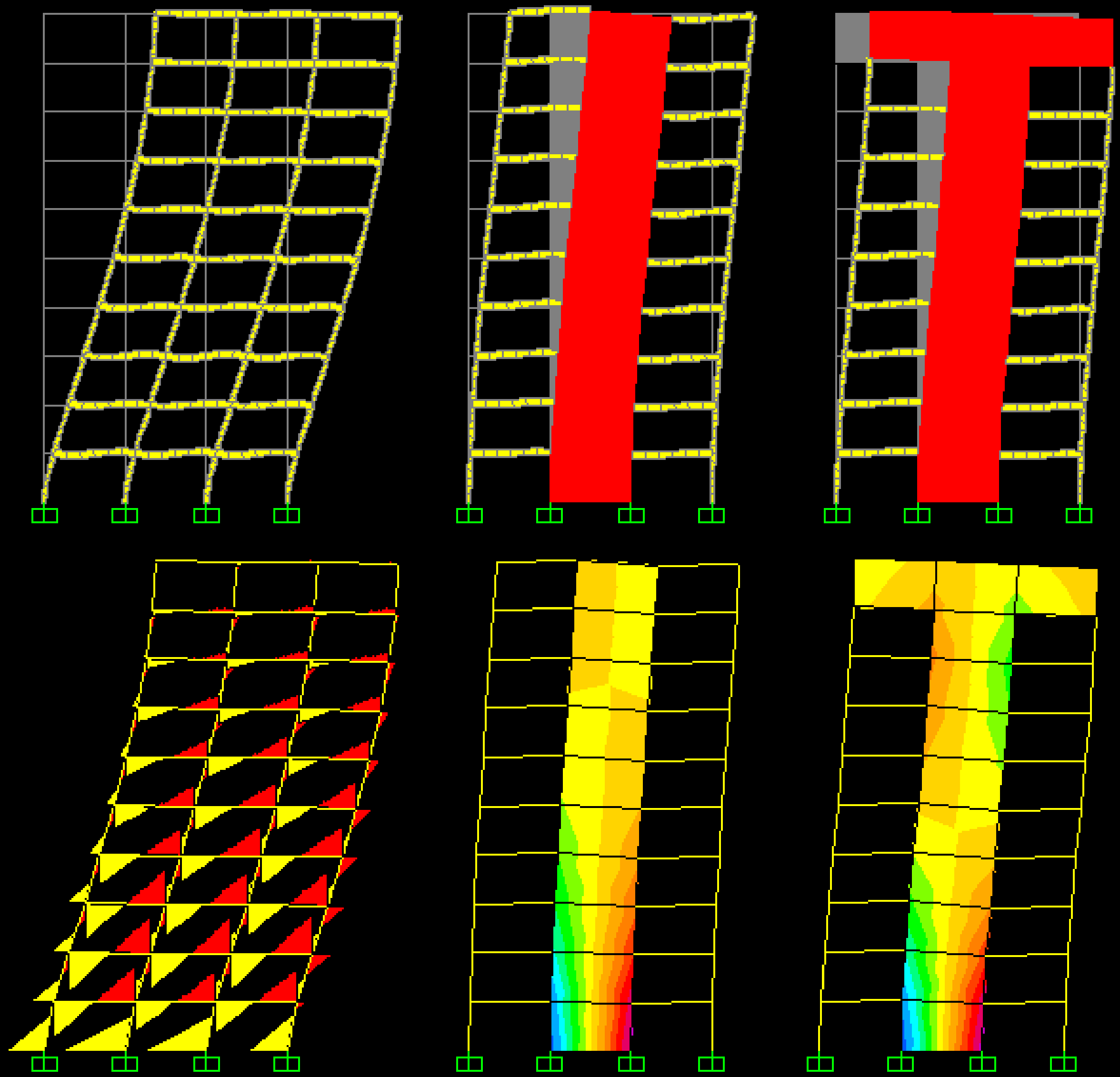






@Seismicisolation


Edit View Define Draw Select Assign Analyze Display Design Options Help
Stress SMIN Diagram (COMB1)
-440. -385. -330. -275. -220. -165. -110. -55. 0.
Stress SMAX Diagram (EQ)
-40. 0. 40. 80. 120. 160. 200. 240. 280.
Stress S22 Diagram (COMB2)
-455. -390. -325. -260. -195. -130. -65. 0. 65.
Stress S12 Diagram (COMB2)
-30.0 -15.0 0.0 15.0 30.0 45.0 60.0 75.0 90.0
Click on any Shell Element for detailed diagram
Start Animation
lb-in
@Seismicisolation



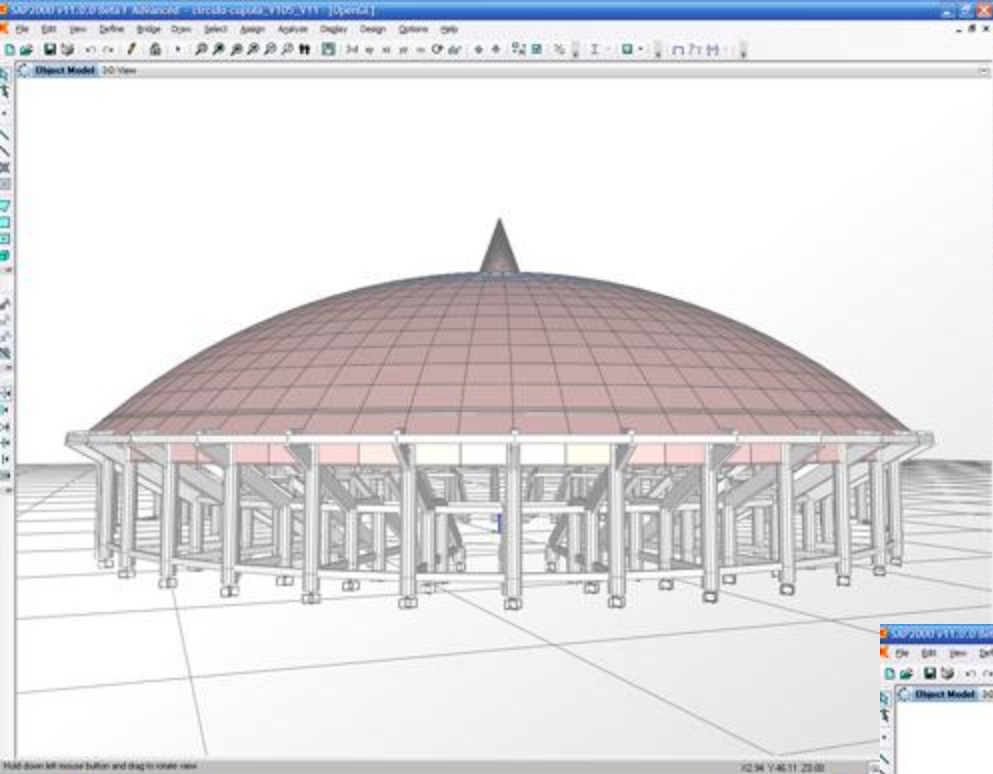
Shanghai Grand Theater, 1998, Jean-Marie Charpentier

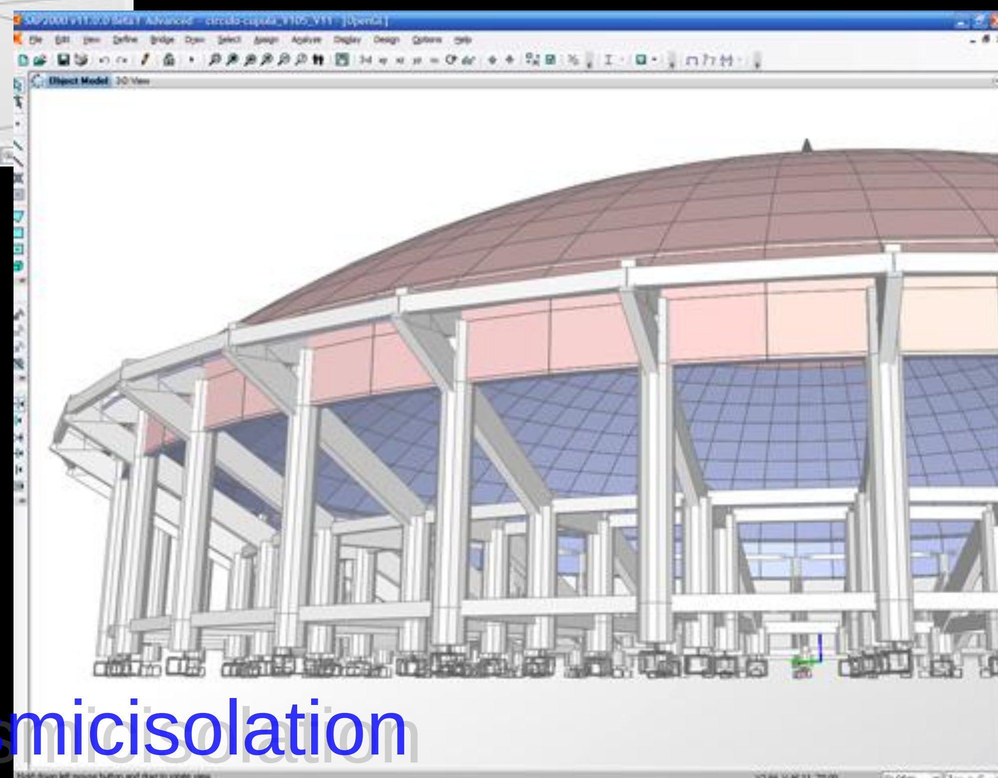
Reichstag, Berlin, Germany, 1999, Norman Foster Arch. Leonhardt & Andrae Struct. Eng

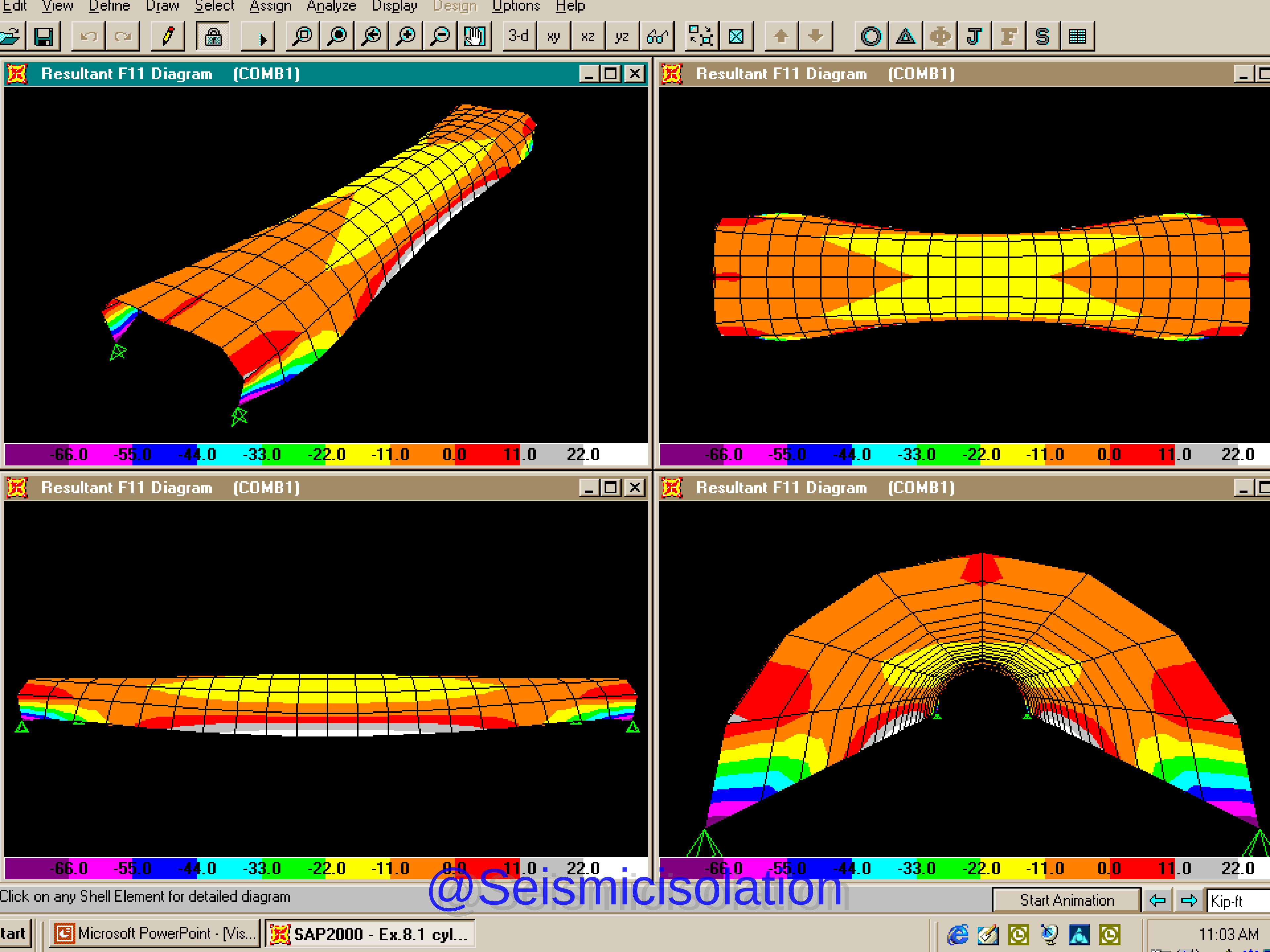

24 9 2001

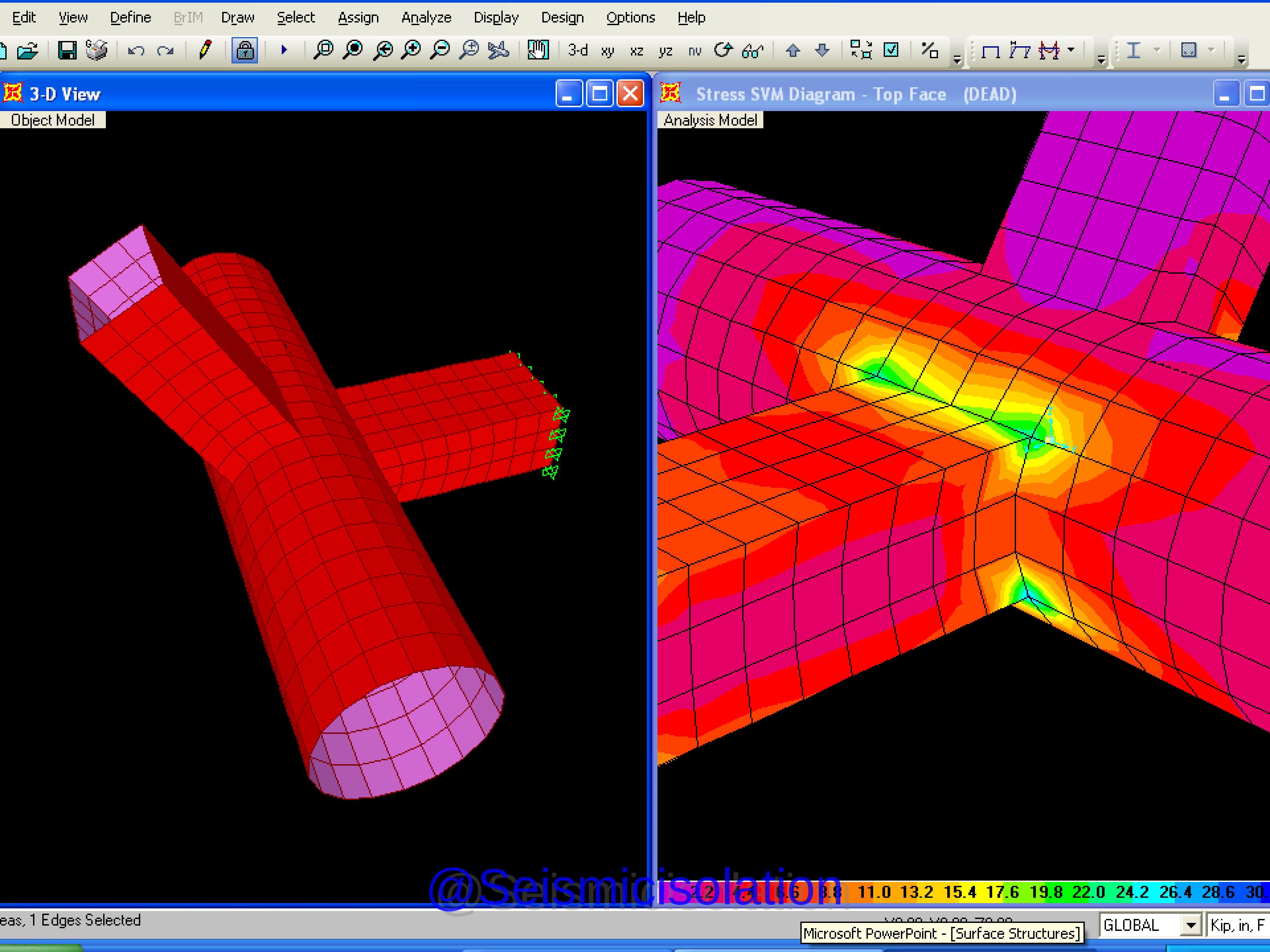

24 9 2001

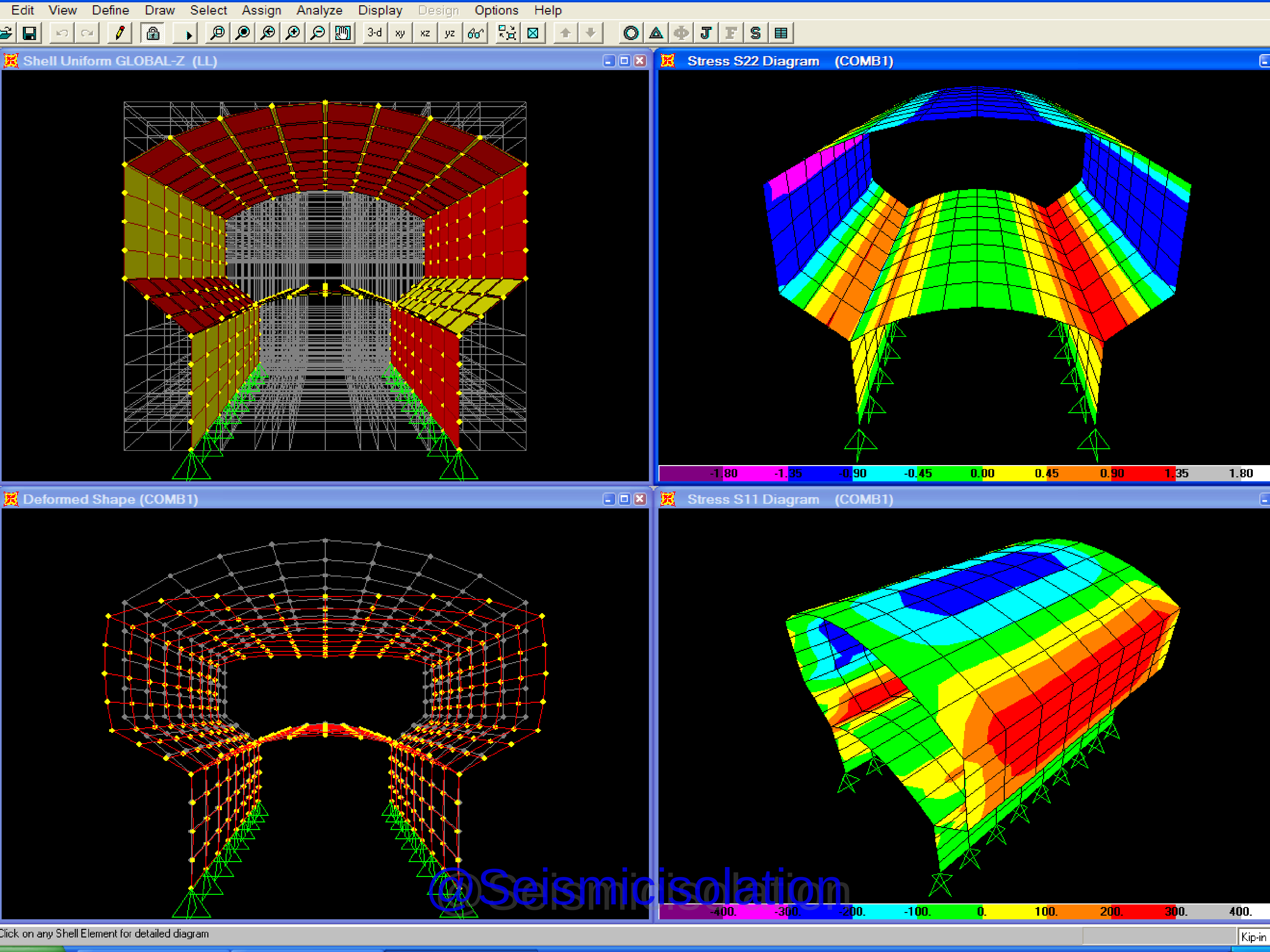

24 9 2001

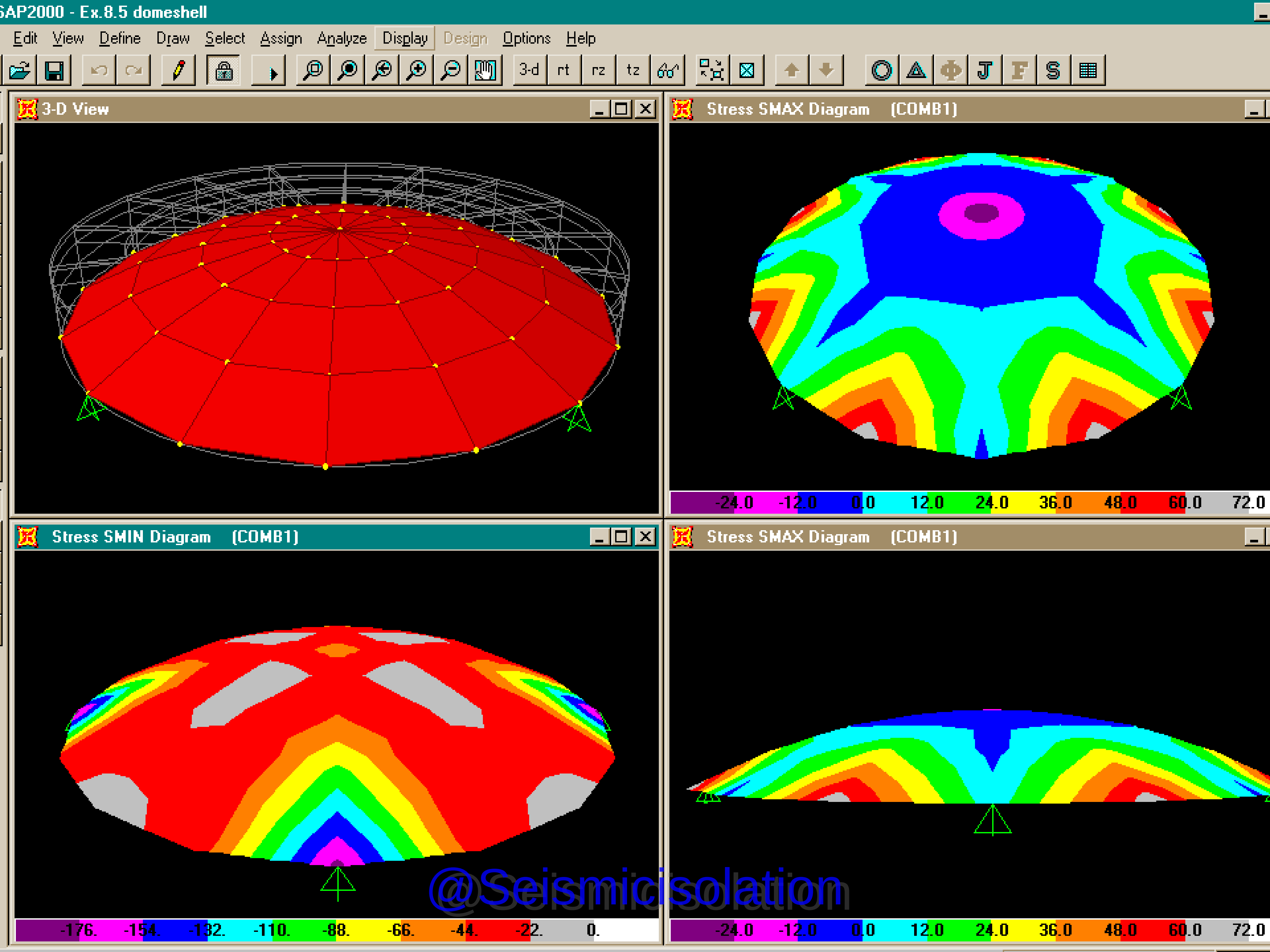


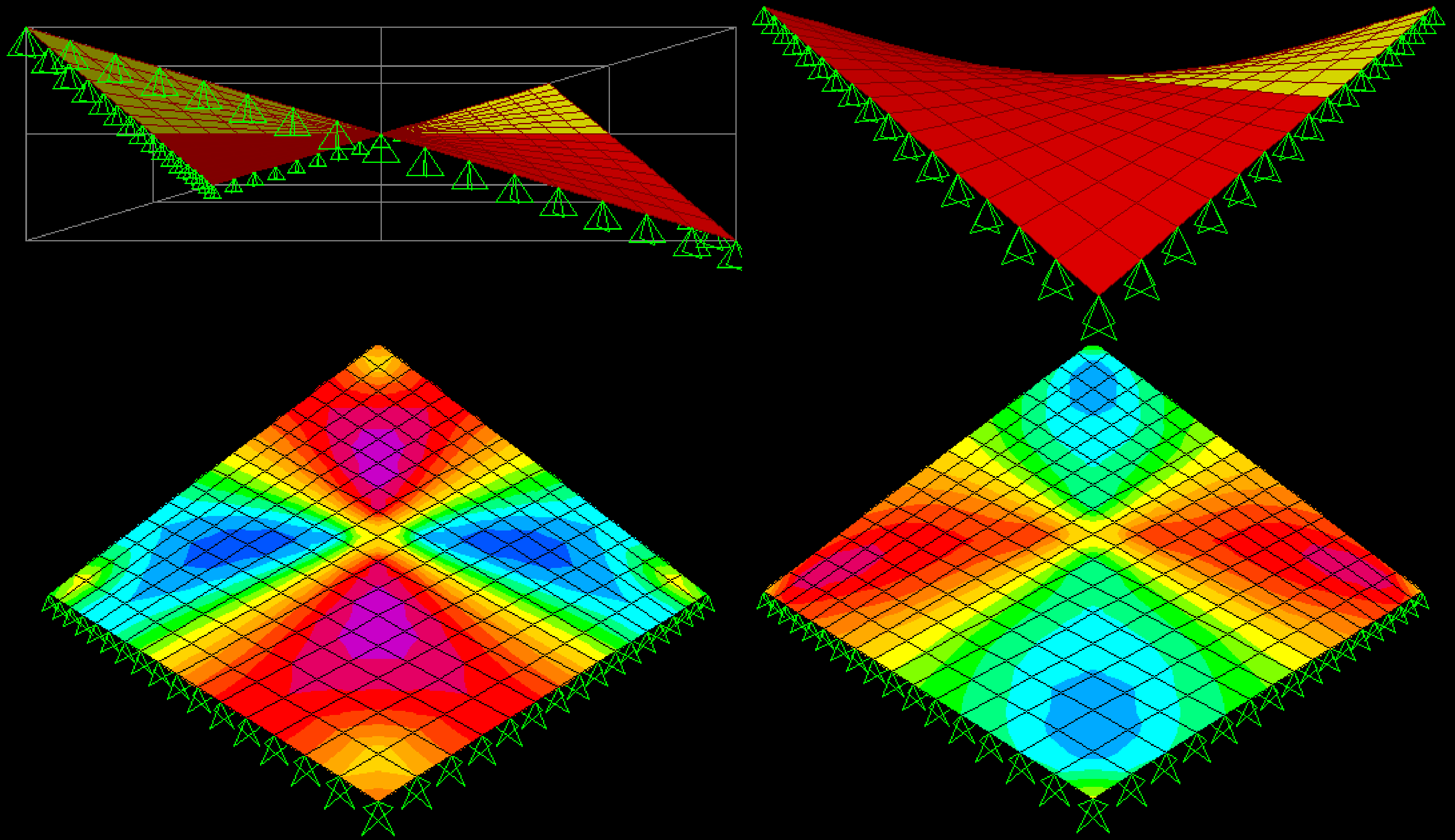


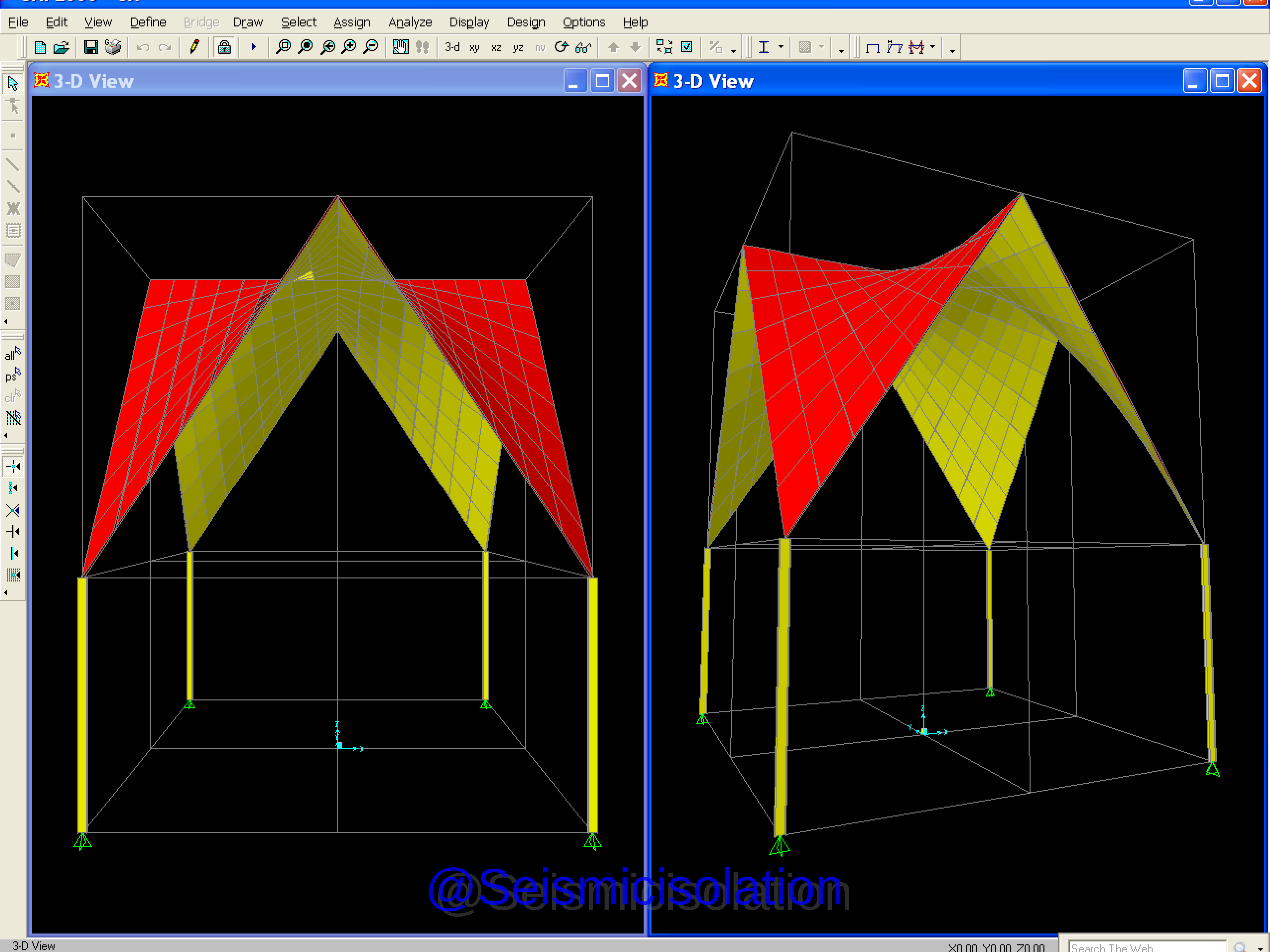

Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz
Resultant F11 Diagram (COMB1)
-66.0 -55.0 -44.0 -33.0 -22.0 -11.0 0.0 11.0 22.0
Resultant F11 Diagram (COMB1)
-66.0 -55.0 -44.0 -33.0 -22.0 -11.0 0.0 11.0 22.0
Resultant F11 Diagram (COMB1)
-66.0 -55.0 -44.0 -33.0 -22.0 -11.0 0.0 11.0 22.0
Resultant F11 Diagram (COMB1)
-66.0 -55.0 -44.0 -33.0 -22.0 -11.0 0.0 11.0 22.0
Click on any Shell Element for detailed diagram
Start Animation
Kip-ft
tart
Microsoft PowerPoint - [Vis...
SAP2000 - Ex.8.1 cyl...
11:03 AM

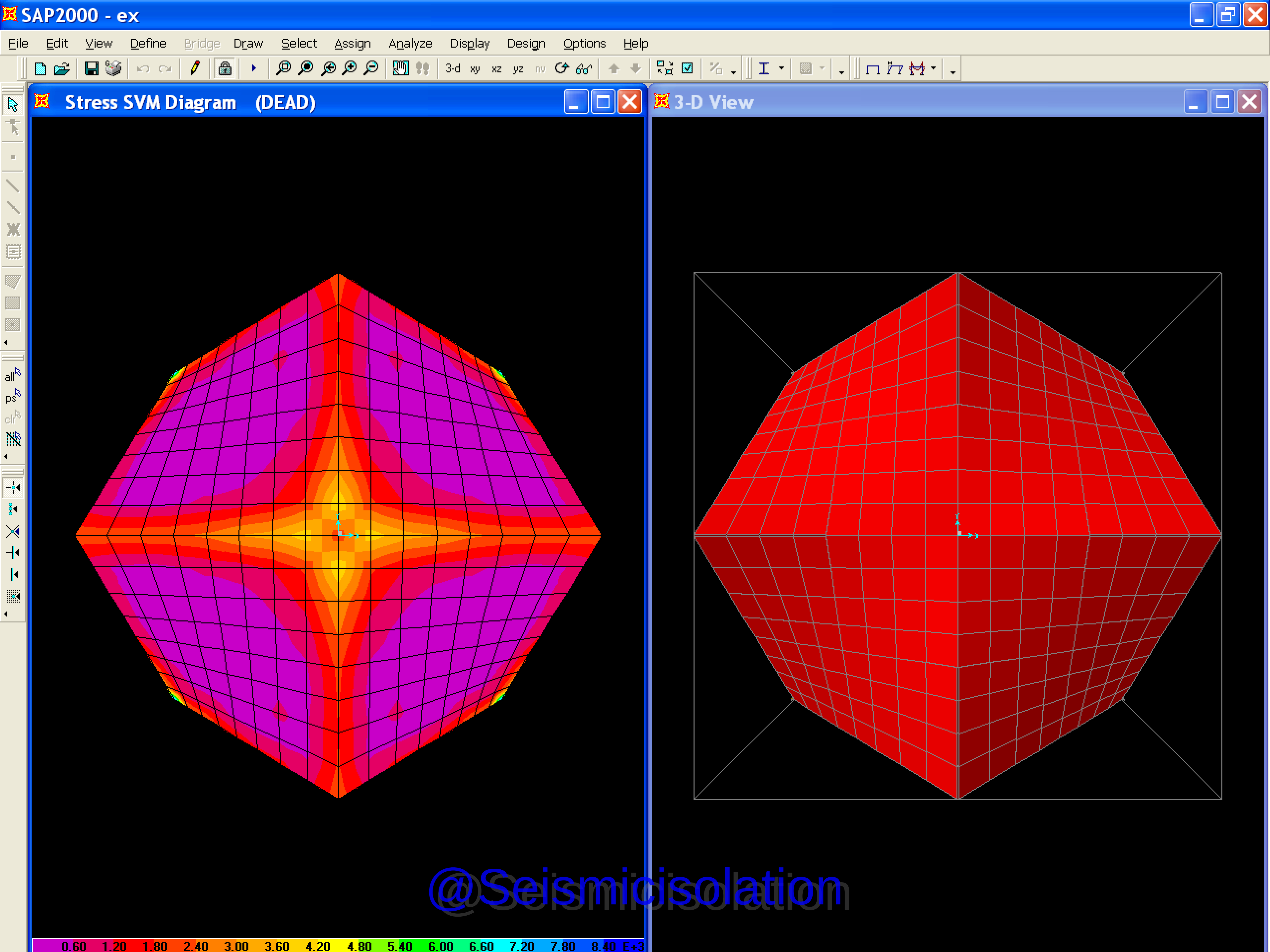

Edit View Define BrIM Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
3-D View
Object Model
Stress SVM Diagram - Top Face (DEAD)
Analysis Model
11.0 13.2 15.4 17.6 19.8 22.0 24.2 26.4 28.6
@Seismicisolation
eas, 1 Edges Selected
Microsoft PowerPoint - [Surface Structures]
GLOBAL
Kip, in, F


Edit View Define Draw Select Assign Analyze Display Design Options Help
Shell Uniform GLOBAL-Z (LL)
Stress S22 Diagram (COMB1)
-1.80 -1.35 -0.90 -0.45 0.00 0.45 0.90 1.35 1.80
Deformed Shape (COMB1)
Stress S11 Diagram (COMB1)
-400. -300. -200. -100. 0. 100. 200. 300. 400.
Click on any Shell Element for detailed diagram
Kip-in


SAP2000 - Ex.8.5 domeshell
Edit View Define Draw Select Assign Analyze Display Design Options Help
3-d rt rz tz
3-D View
Stress SMAX Diagram (COMB1)
-24.0 -12.0 0.0 12.0 24.0 36.0 48.0 60.0 72.0
Stress SMIN Diagram (COMB1)
-176. -154. -132. -110. -88. -66. -44. -22. 0.
Stress SMAX Diagram (COMB1)
-24.0 -12.0 0.0 12.0 24.0 36.0 48.0 60.0 72.0
@Seismicisolation




File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
3-D View
3-D View
3-d xy xz yz
3-D View
X0.00 Y0.00 Z0.00
Search The Web


SAP2000 - ex
File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Stress SVM Diagram (DEAD)
3-D View
0.60 1.20 1.80 2.40 3.00 3.60 4.20 4.80 5.40 6.00 6.60 7.20 7.80 8.40 E+3

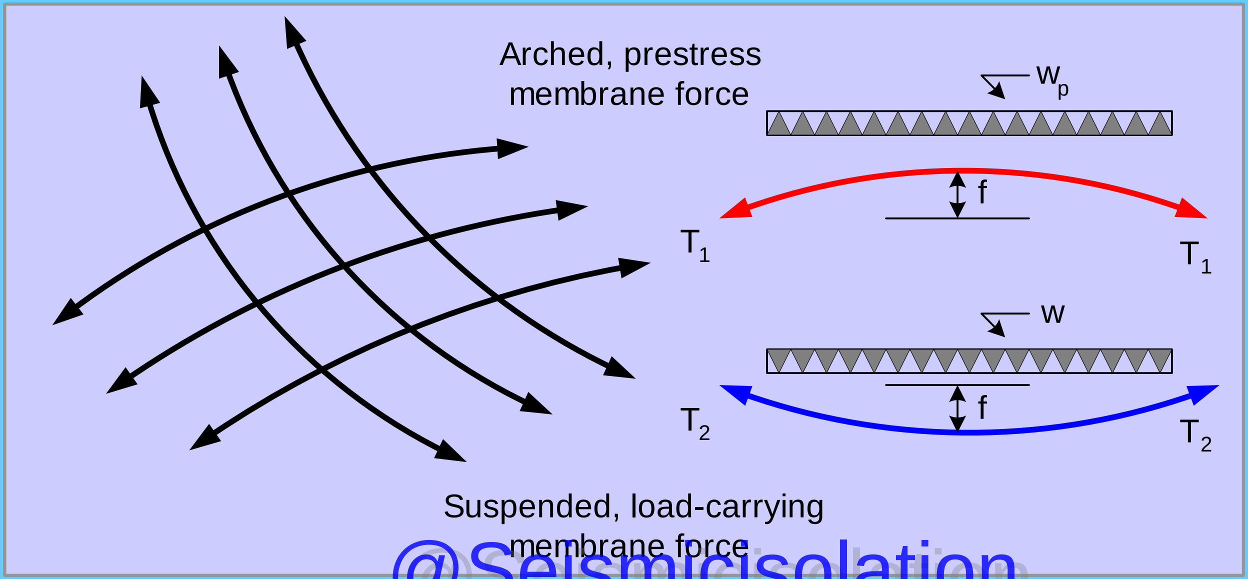
DG Bank, Berlin, Germany, 2001, Frank Gehry; Schlaich, Bergemann und Partners

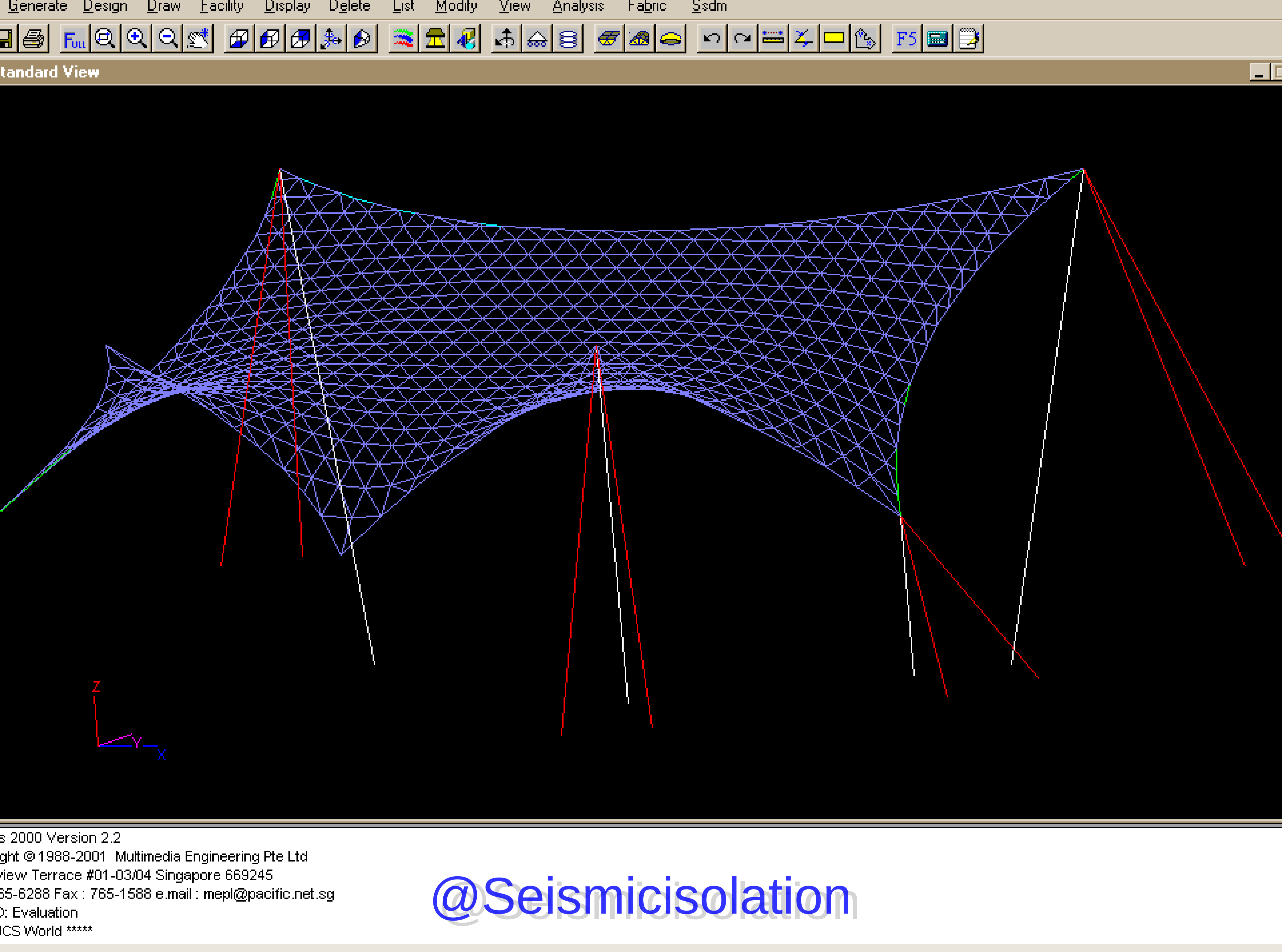

25 9 2001

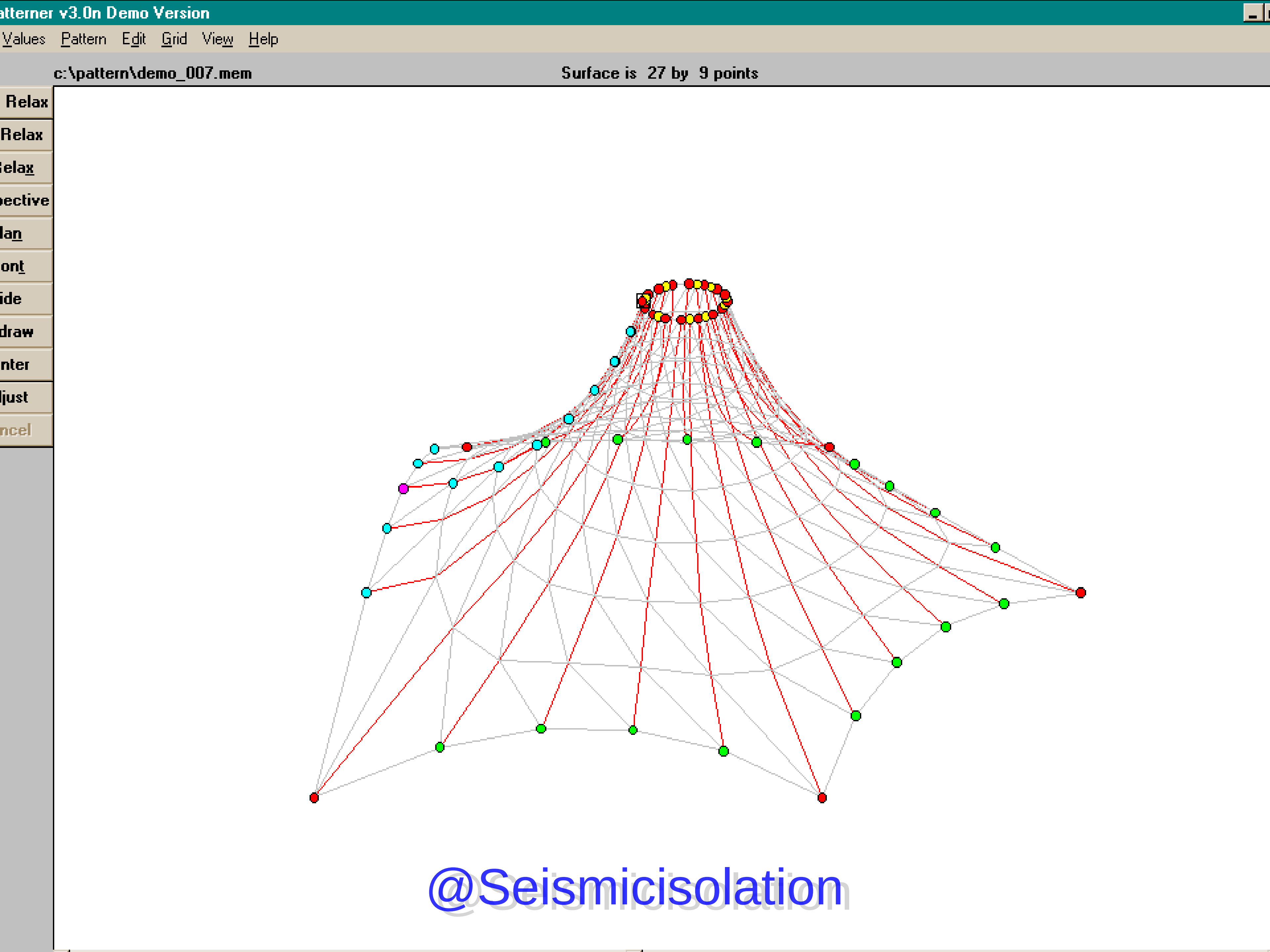
TENSILE SURFACES

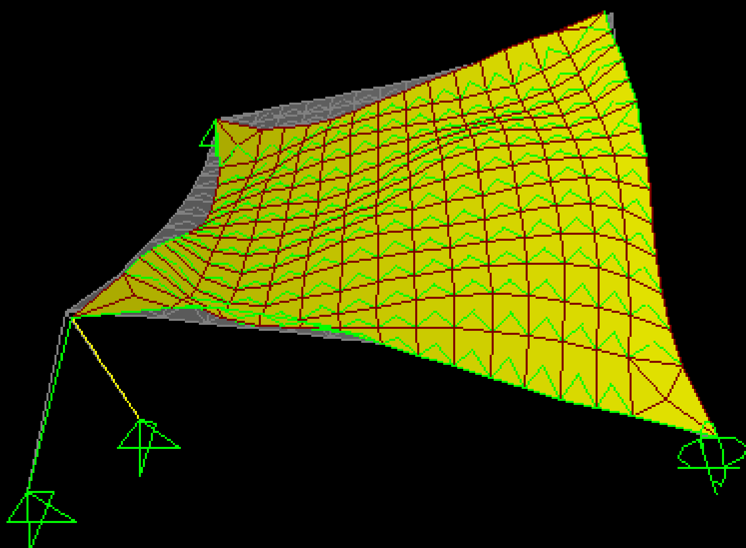
STRUCTURAL BEHAVIOR

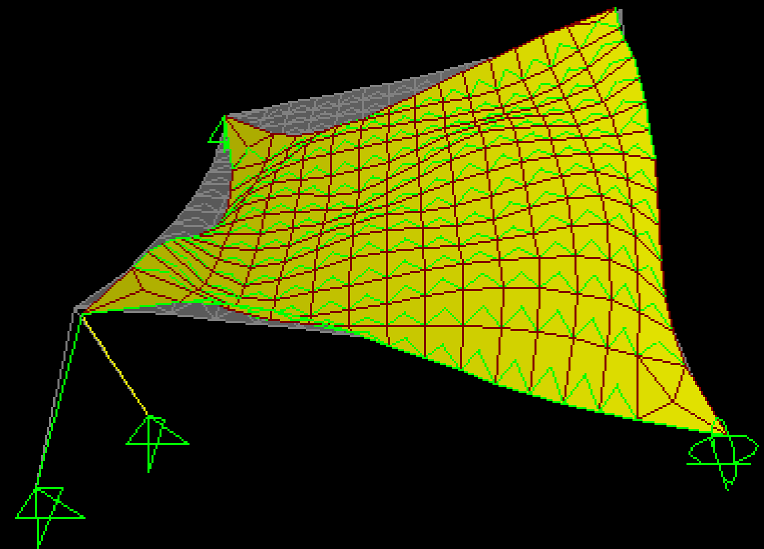
Soft membranes must adjust their shape (because they are flexible) to the loading so that they can respond in tension. The membrane surface must have double curvature of anticlastic geometry to be stable. The basic shape is defined mathematically as a hyperbolic paraboloid. In cable-nets under gravity loads, the main (convex, suspended, lower load bearing) cable is prevented from moving by the secondary (concave, arched, upper, bracing, etc.) cable, which is prestressed and pulls the suspended layer down, thus stabilizing it. Visualize the initial surface tension analogous to the one caused by internal air pressure in pneumatic structures.

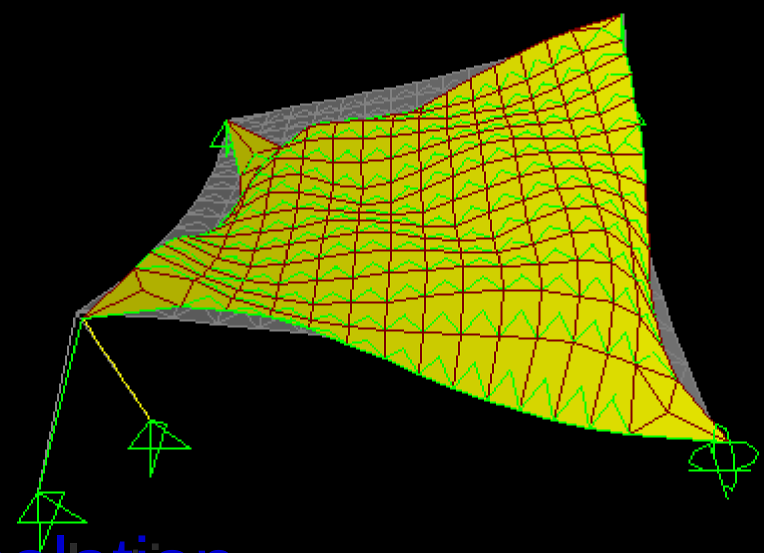



Generate Design Draw Facility Display Delete List Modify View Analysis Fabric Ssdm
Full
F5
tandard View
Z
Y
X
s 2000 Version 2.2
ght © 1988-2001 Multimedia Engineering Pte Ltd
view Terrace #01-03/04 Singapore 669245
65-6288 Fax : 765-1588 e.mail : mepl@pacific.net.sg
D: Evaluation
JCS World *****


atterner v3.0n Demo Version
Values Pattern Edit Grid View Help
c:\pattern\demo_007.mem
Surface is 27 by 9 points
Relax
Relax
elax
ective
lan
ont
ide
draw
nter
just
ncel

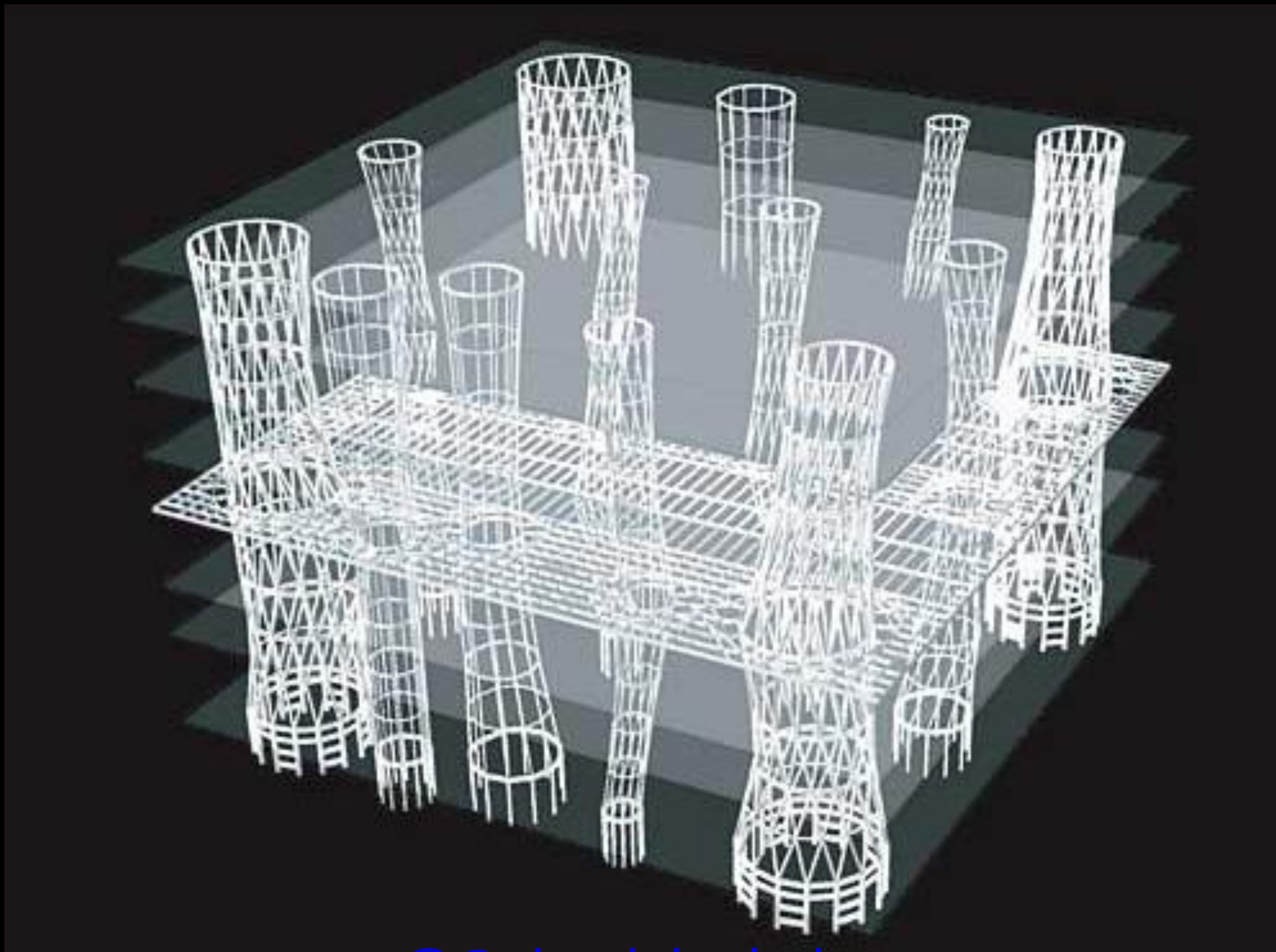


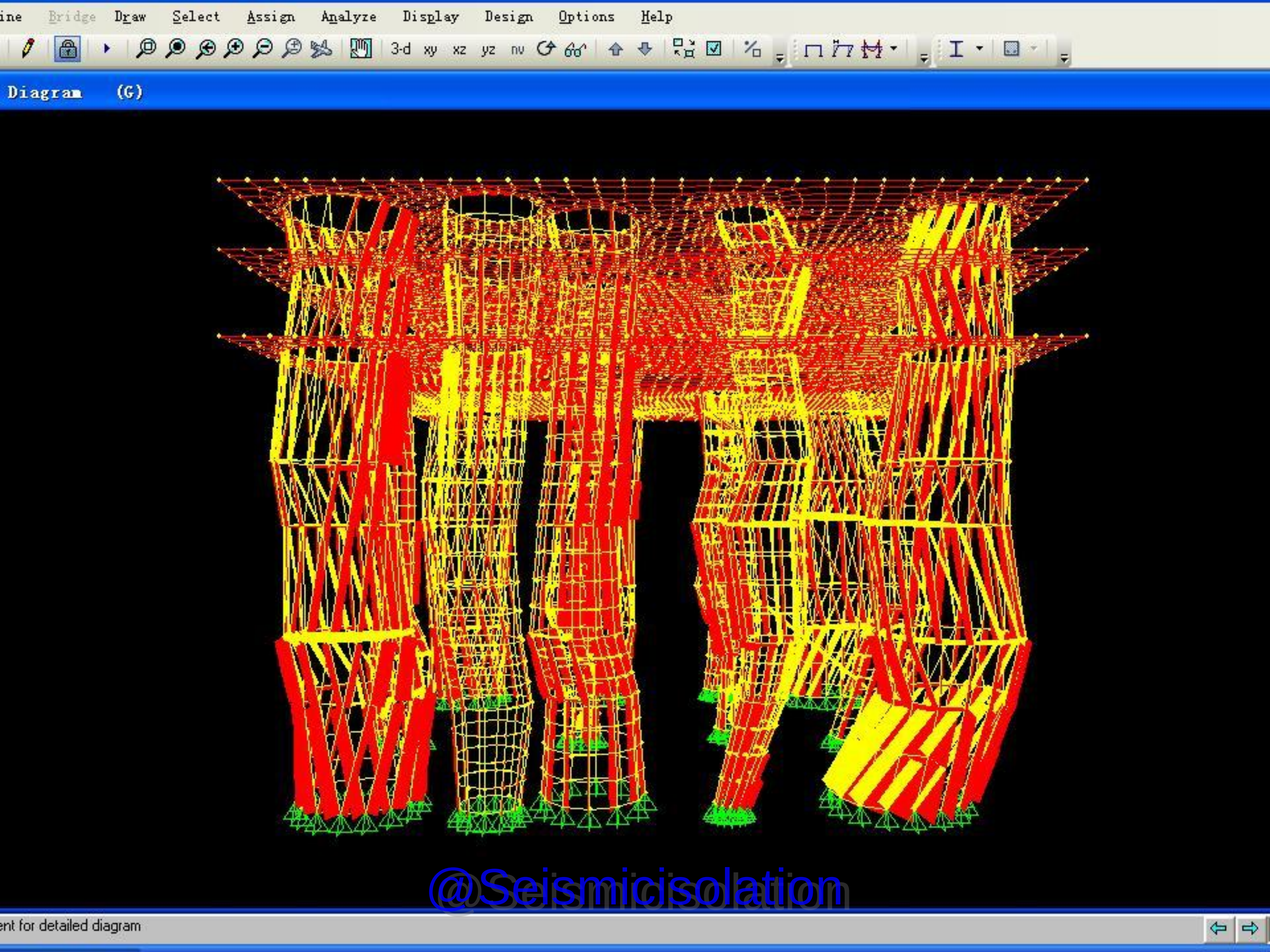


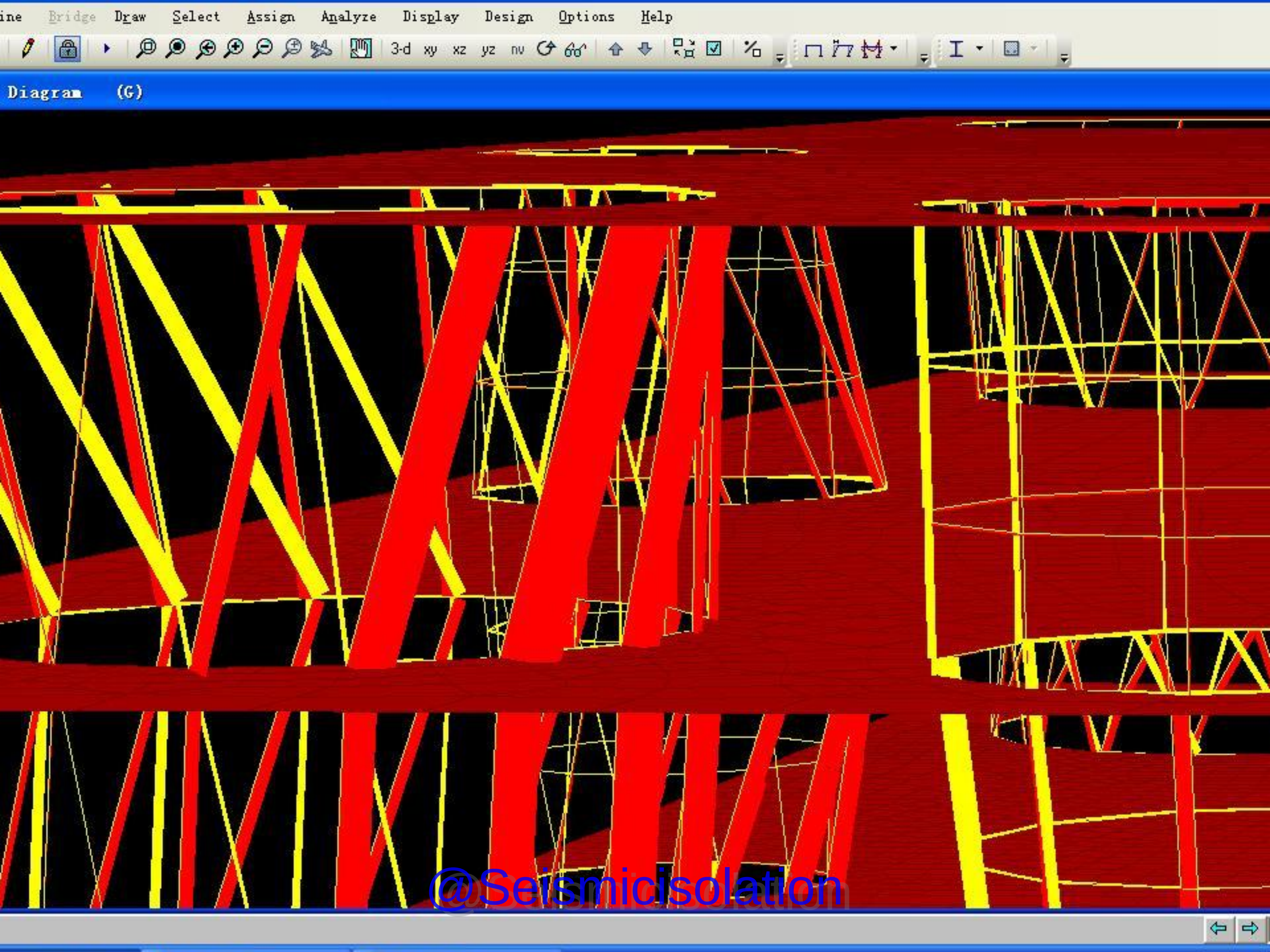


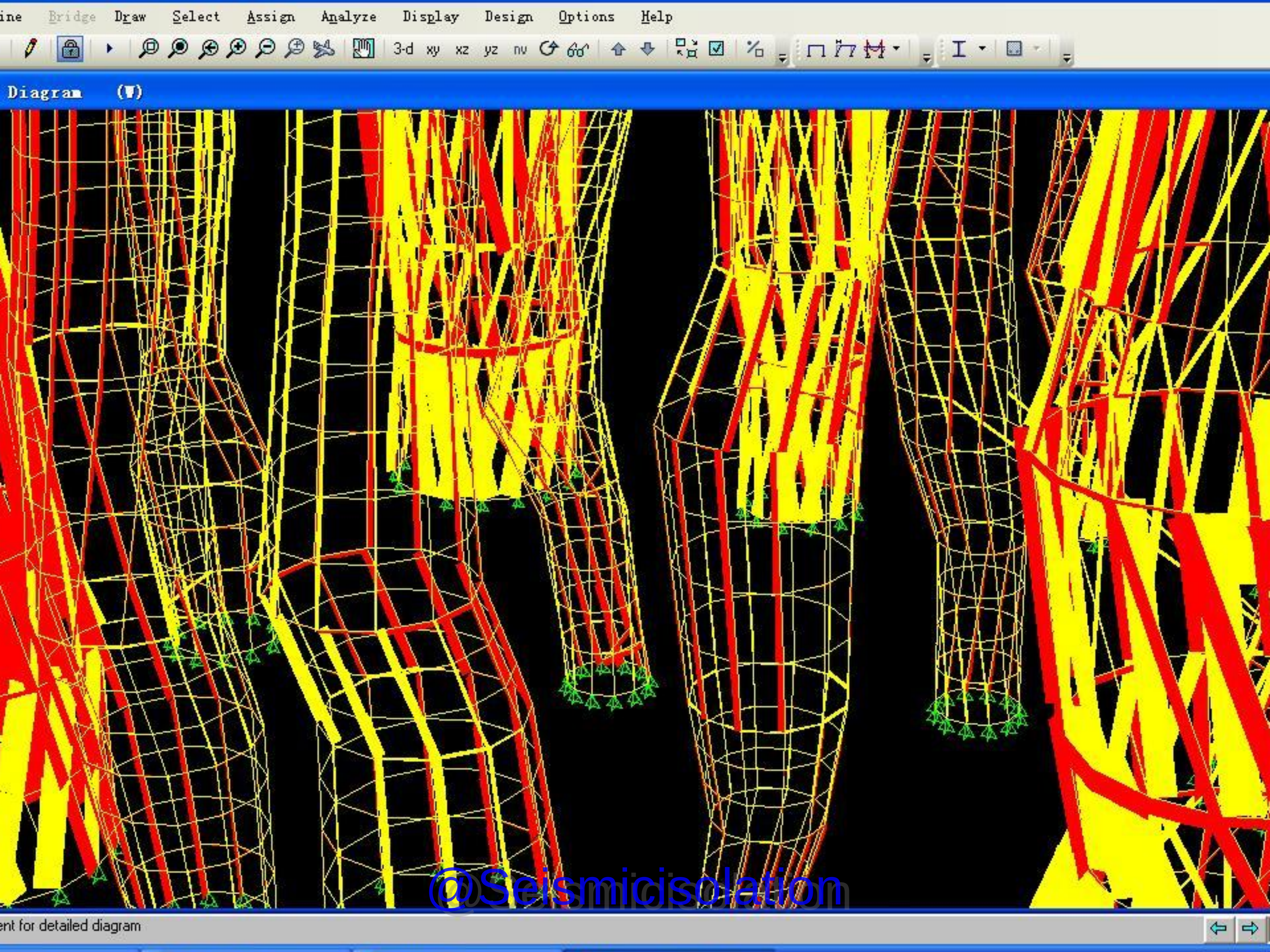

T H E NEW GEOMETRY
Daniel Libeskind . BERLIN CITY EDGE

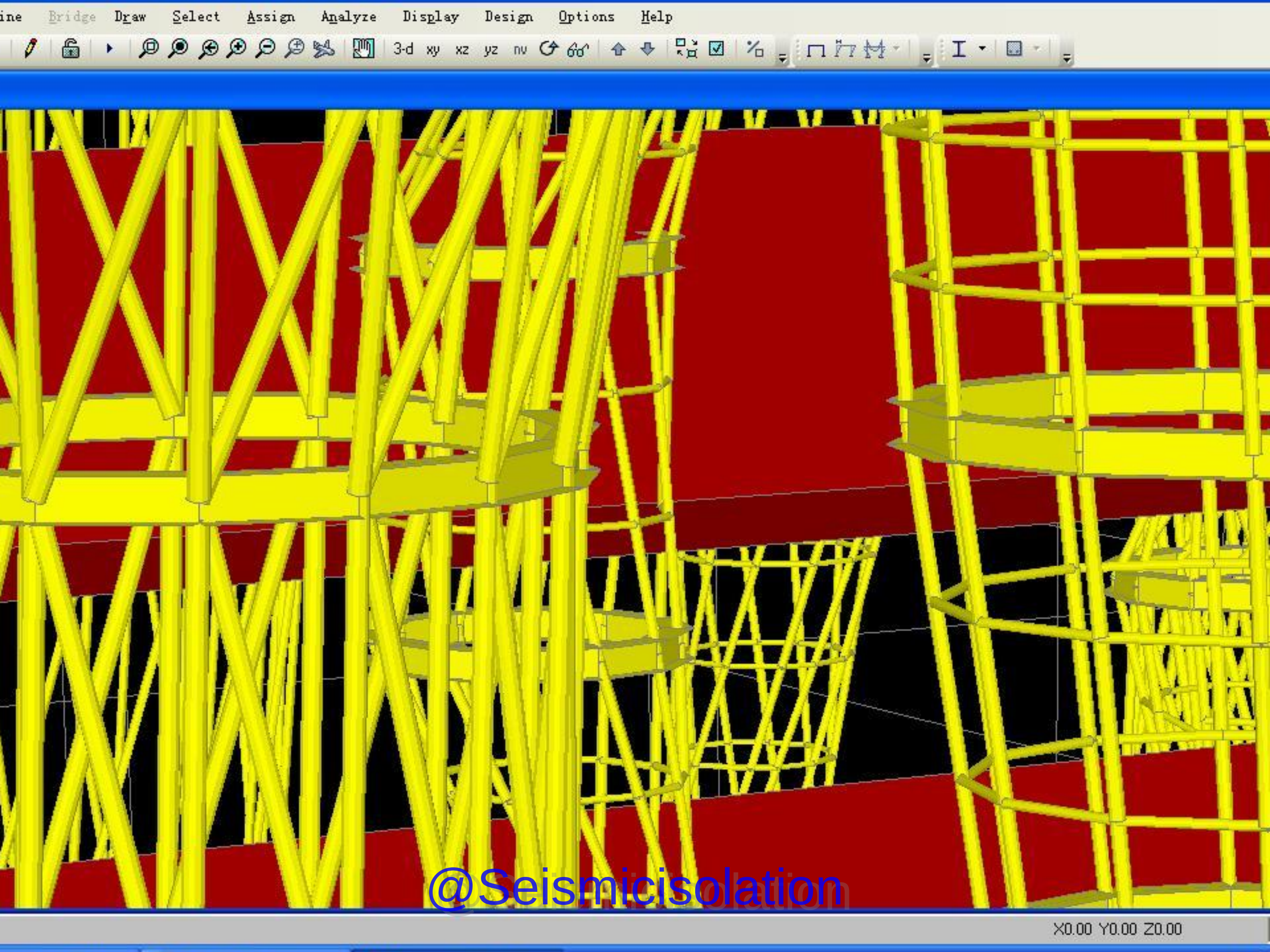
Sendai Mediatheque, Sendai-shi, Japan, 2001, Toyo Ito + Mutsuro Sasaki

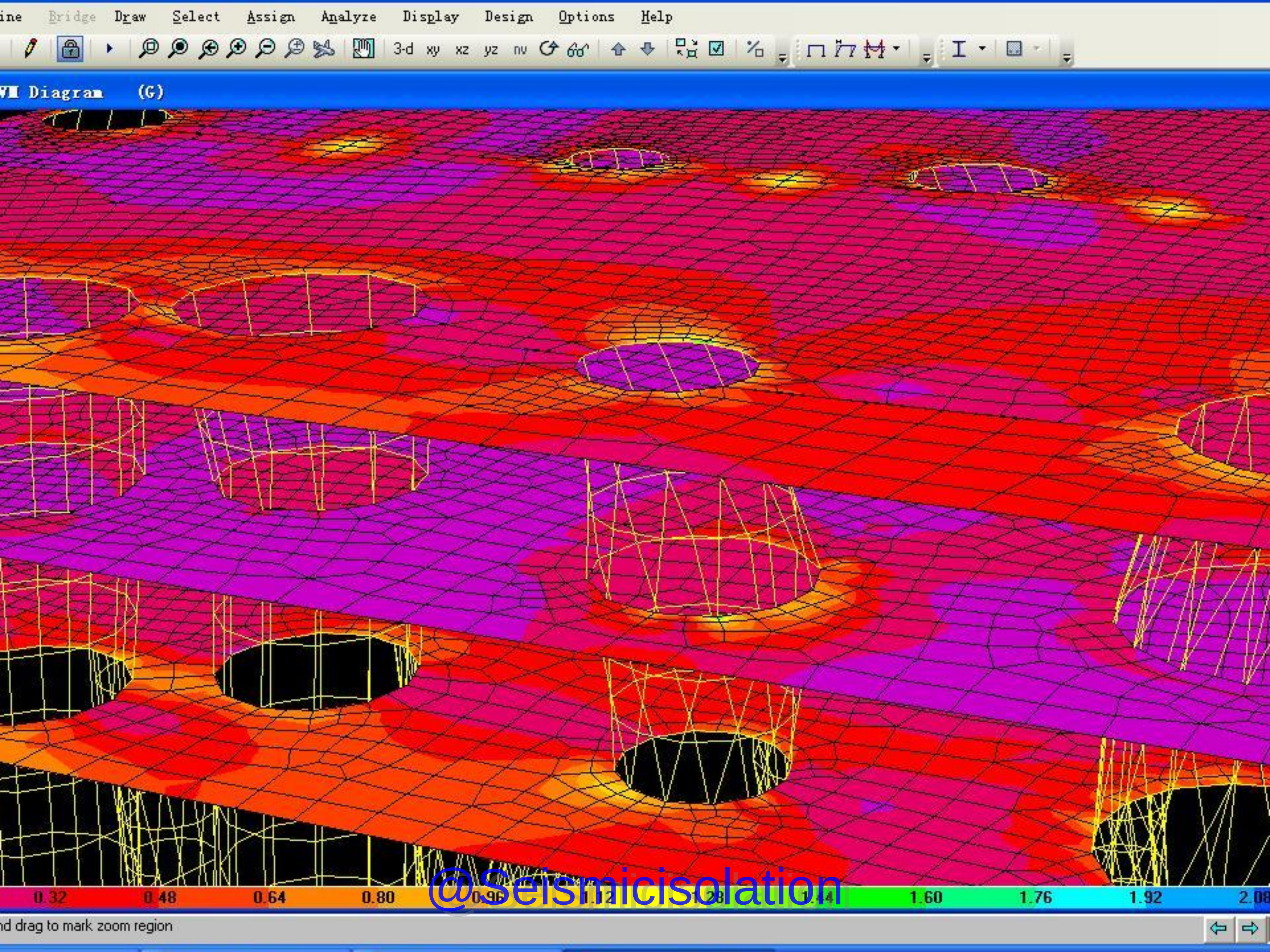


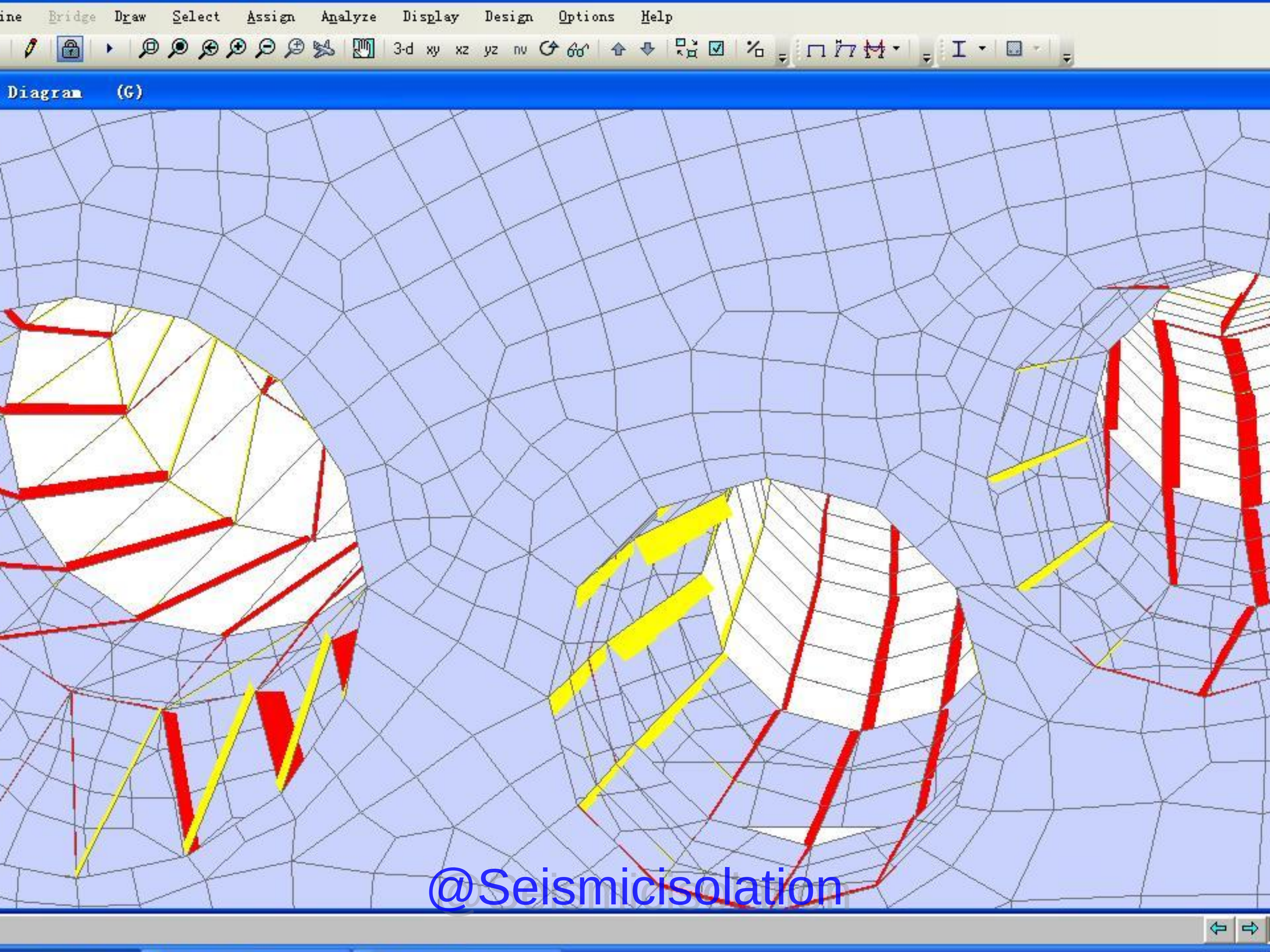

ine
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
nv
Diagram
(G)
ent for detailed diagram

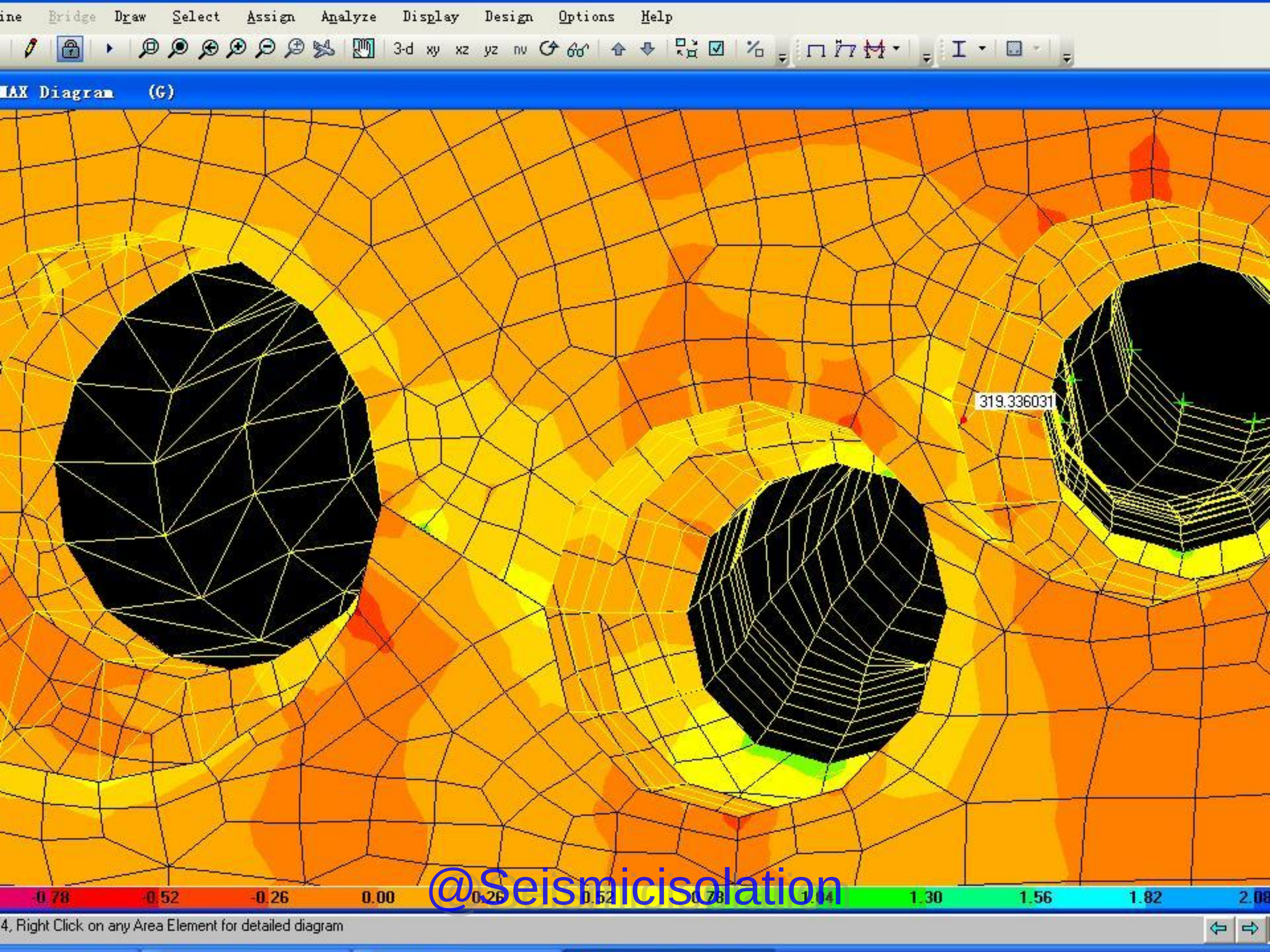

ine Bridge Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
Diagram (G)
@Seismicisolation


ine
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
nv
Diagram
ent for detailed diagram

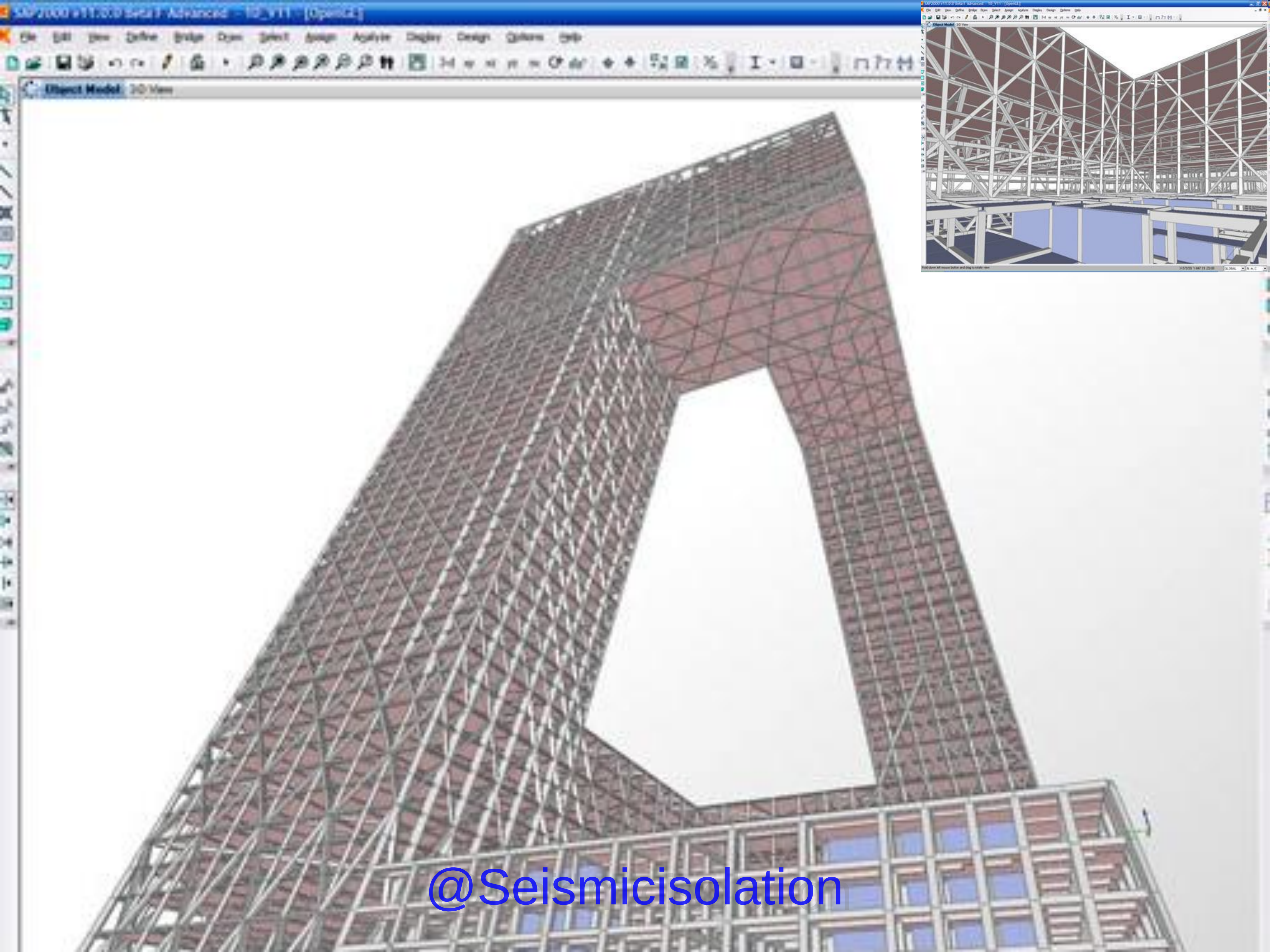

ine
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
nv
X0.00 Y0.00 Z0.00


ine
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
nv
Diagram (G)
0.32
0.48
0.64
0.80
1.60
1.76
1.92
@Seismicisolation
nd drag to mark zoom region

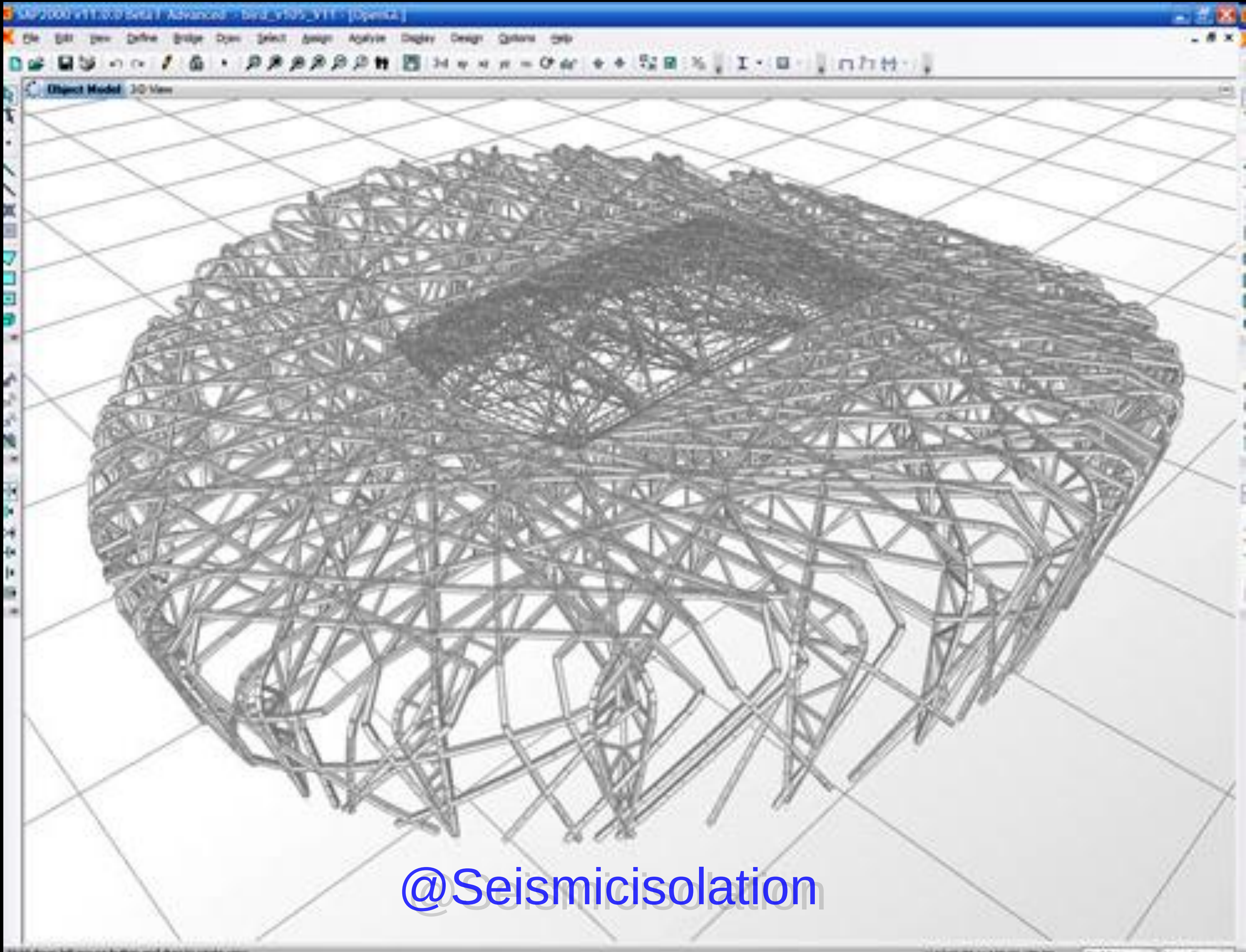

ine
Bridge
Draw
Select
Assign
Analyze
Display
Design
Options
Help
3-d
xy
xz
yz
nv
Diagram (G)

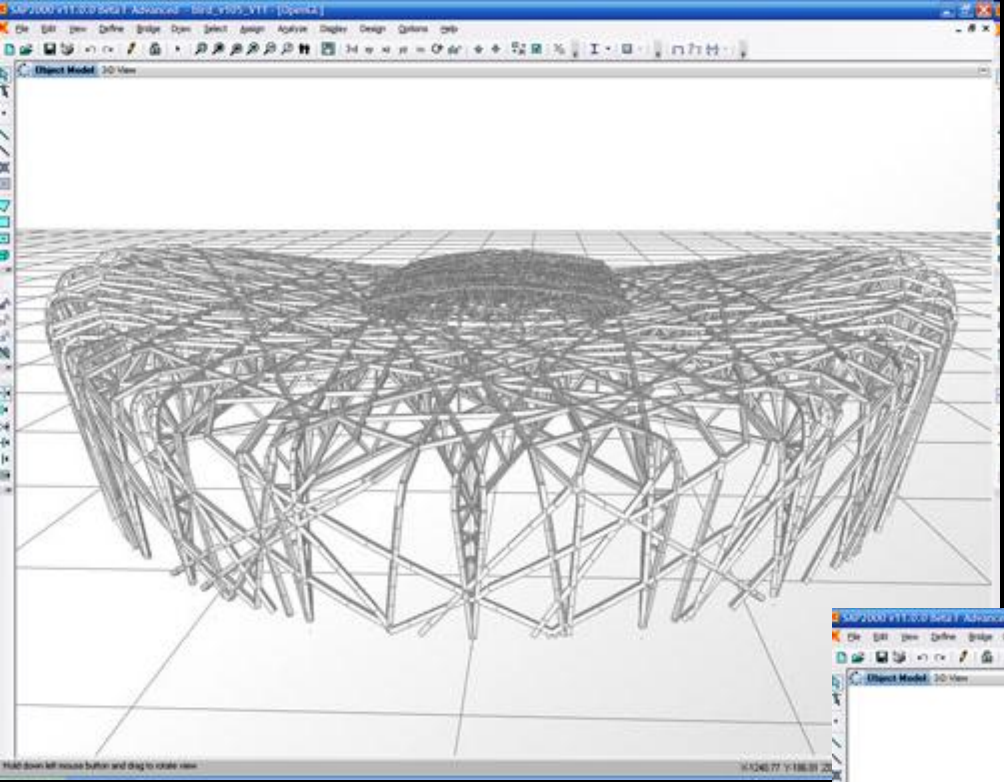

ine Bridge Draw Select Assign Analyze Display Design Options Help
3-d xy xz yz nv
MAX Diagram (G)
319.336031
-0.78 -0.52 -0.26 0.00 0.26 0.52 0.78 1.04 1.30 1.56 1.82 2.08
4, Right Click on any Area Element for detailed diagram

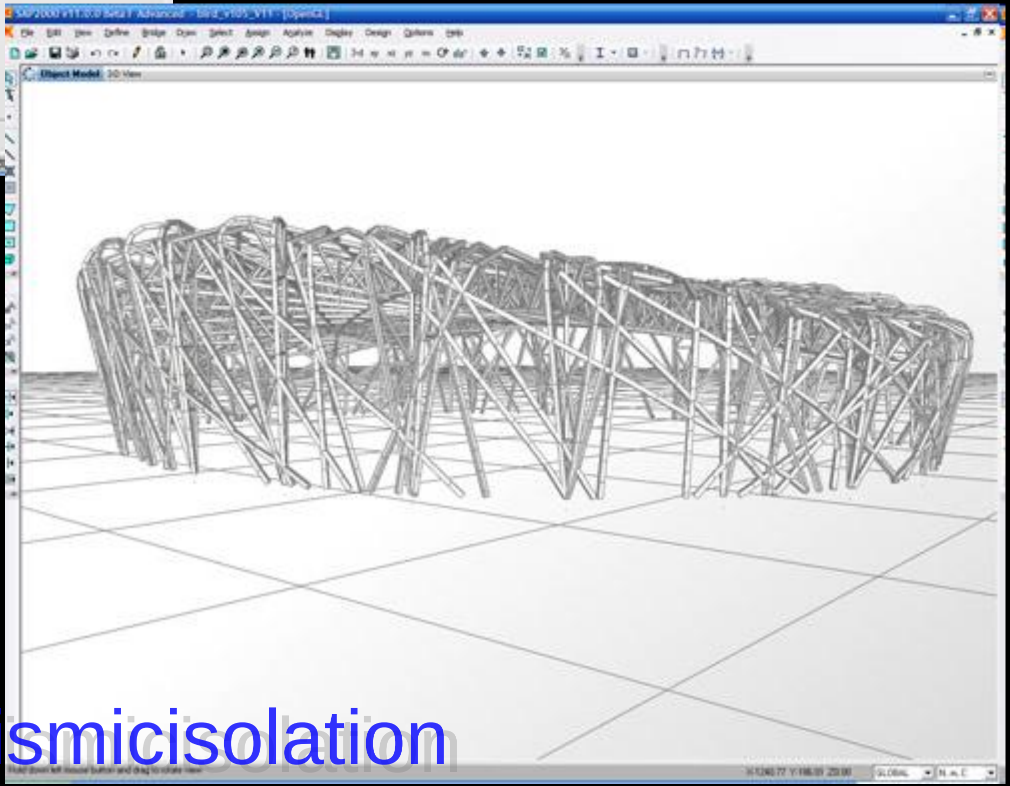
CCTV Headquarters and TVCC Building, Beijing, 2008, Rem Koolhaas and Ole Scheeren, Arup Eng.

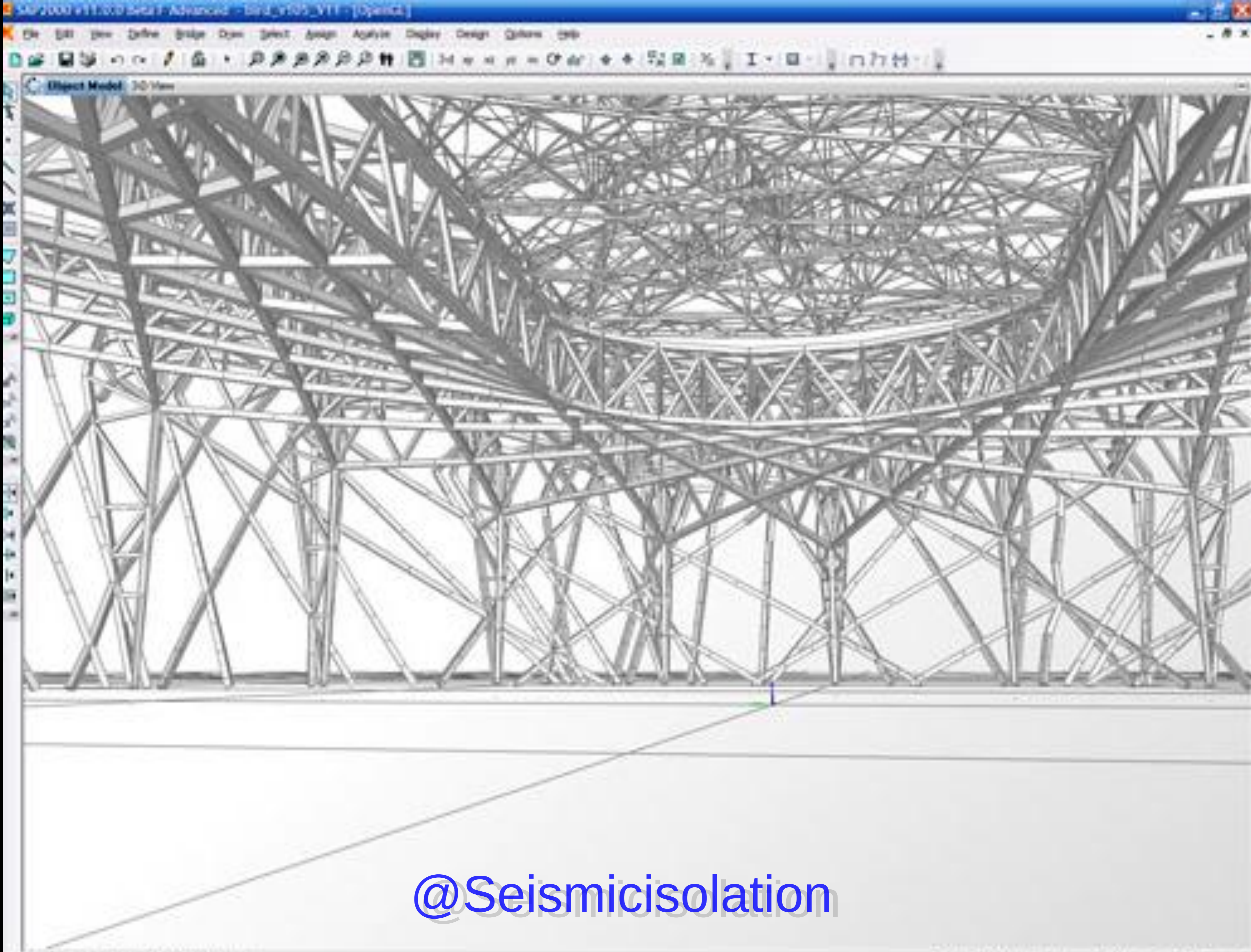

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help
Object Model

Beijing National Stadium, 2008, Herzog and De Meuron Arch, Arup Eng.

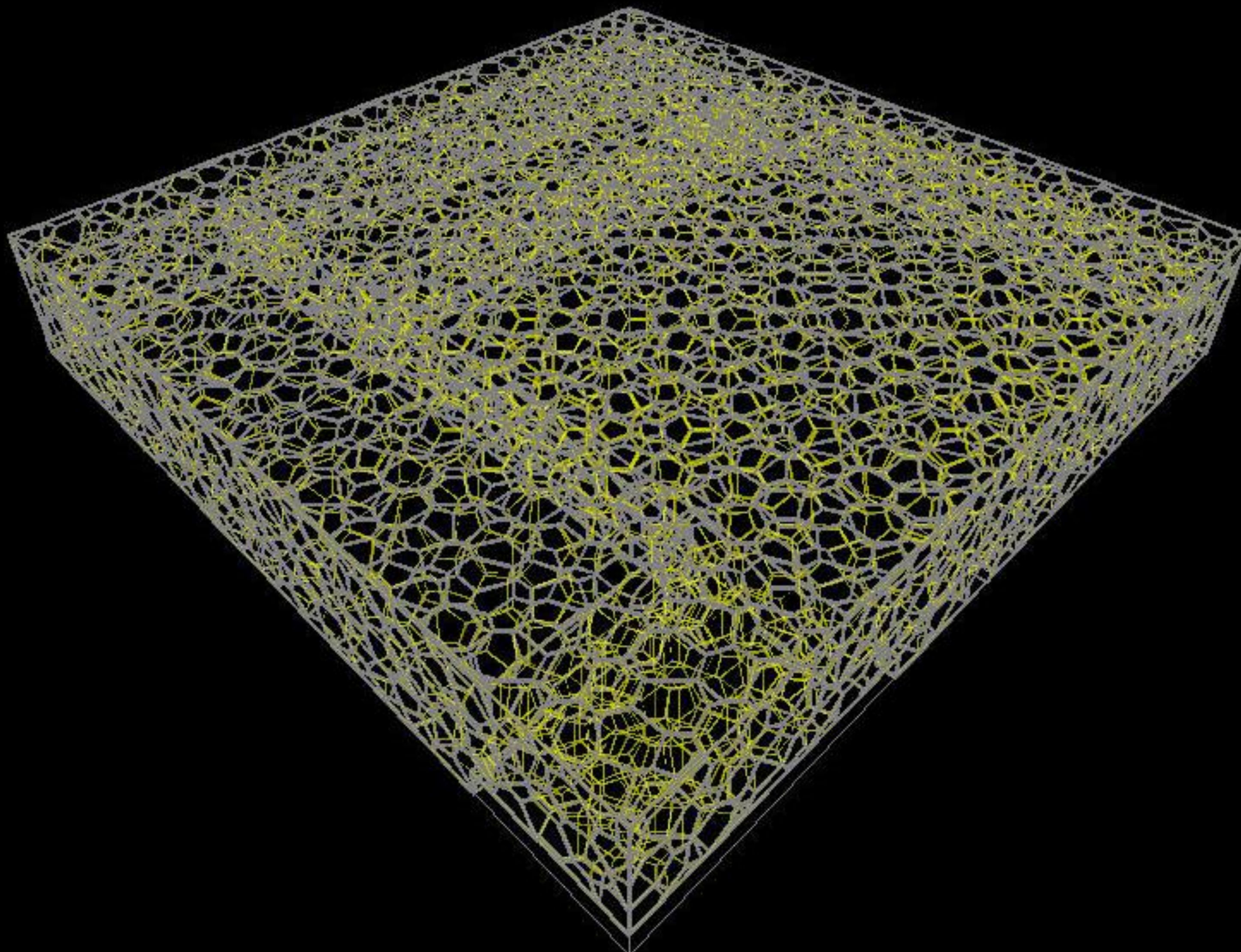


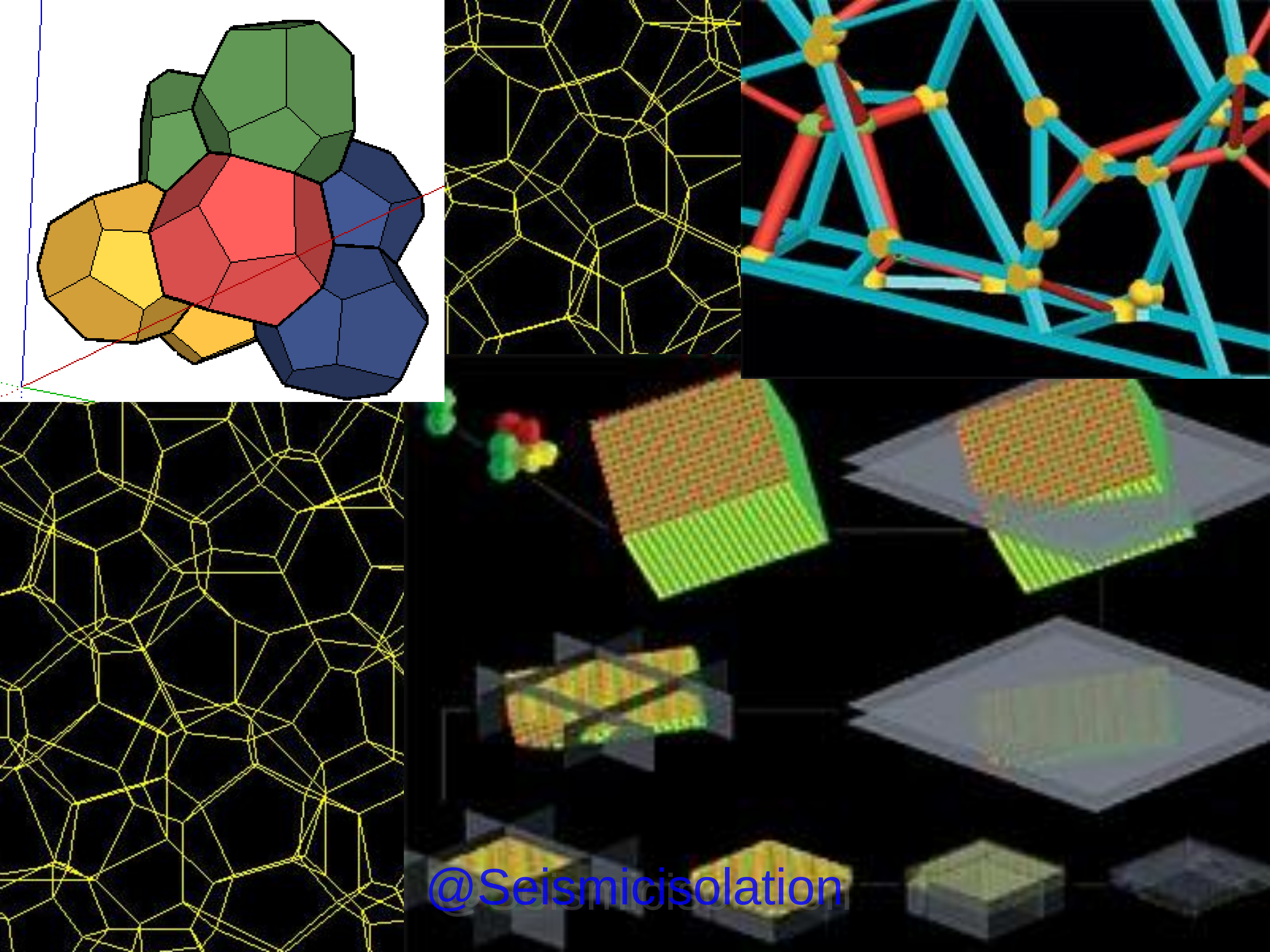




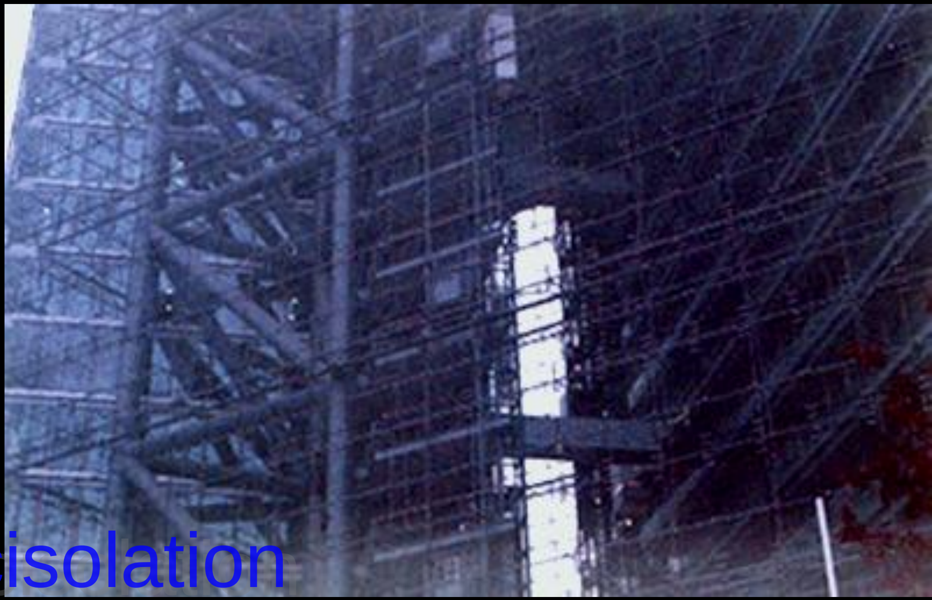

File Edit View Define Bridge Draw Select Assign Analyze Display Design Options Help

National Swimming Center, Beijing, 2008, Arup Arch and Eng

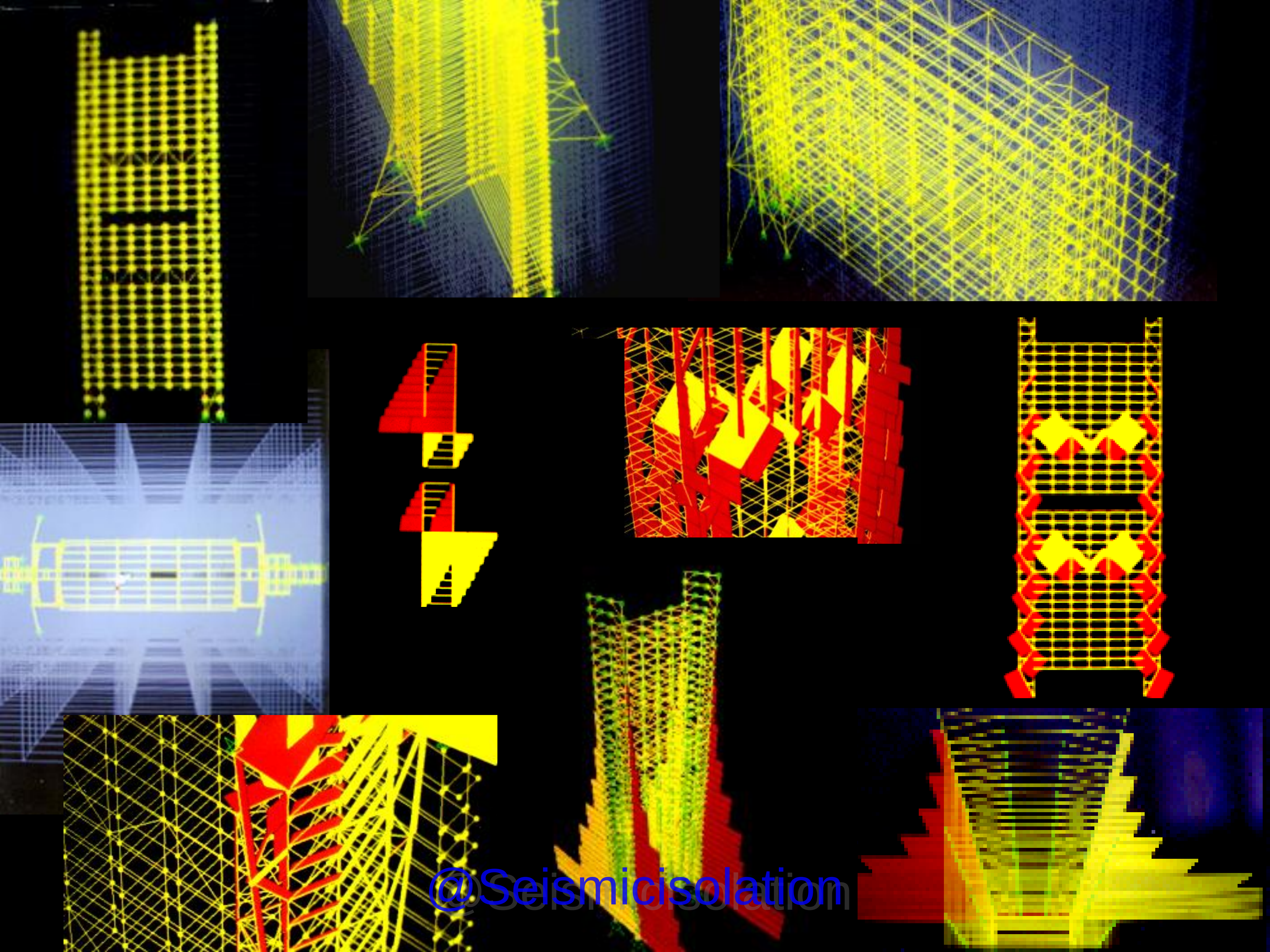




Duesseldorf City Gate,
Duesseldorf, Germany, 1997,
H. Petzinka + Fink Arch



Seoul Broadcasting Center, Seoul, 2003, Richard Rogers Arch. And Buro Happold Struct. Eng



Samsung Samsung Jongro Tower, Seoul, 1999, Rafael Vinoly