Cary H. Beatisula

UD69242SST78376

COURSE NAME:

Structural Analysis using Computer Software

Assignment Title:

Course Work Assignment for Structural Analysis using Computer Software

ATLANTIC INTERNATIONAL UNIVERSITY

January 2021

I. Introduction

Structural Analysis is the essential aspect of a given structure before its structural elements can be designed. At present structural analysis for complex systems can be performed using computer software.

STAAD Pro is a structural analysis and structural design-oriented program with an interactive user interface that allows the user to work exceptionally quickly. It can be used for modeling, structural analysis, and structural design of any structure's structural members.

STAAD Pro is a structural engineering software with features of performing structural analysis and structural design of any given complex structure. The following are the major item of work to be done using the above software: 1. We are modeling the structural geometry 2. types of loads to be applied to the structure 3. Input the constant parameter for the structure 4. Perform Analysis 5. Interpretation of Results 6. structural design of the structural members.

The following are the types of loads: Dead Loads (DL), Live Loads (LL), Wind Loads, Seismic loads (Earthquake Loads), and load combinations. The Structural Analysis output is available for the stresses in beams, stresses in columns, and support reactions.

STAAD Pro is commonly used for structural analysis of structures such as towers, buildings, bridges, transport infrastructure, industrial structures, and other complex structures. It is beneficial for residential buildings, commercial buildings, Hotel Buildings, Condominiums, and other complex facilities. It is also helpful for structural analysis of bridges, which includes the structural design of a foundation. It has a feature for structural analysis and structural design of Shear wall for a high rise building. It can display real images of the structural geometry of the structure both in 2D and 3D.

The Structural Analysis Application using computer software is applied to the Proposed 3 Storey Commercial Building with Roofdeck, constructed at Bansalan, Davao del Sur.

II. Modeling the Structural Geometry

The structure's geometry consists of joint numbers, coordinates, member numbers, details on the member connectivity, and numbers of plate elements.

a. Joint Coordinates

This command allows you to define the coordinates of the structure joint and to generate them. It initiates the specification of the coordinates with the Joint Coordinates. The **Repeat** and **Repeat All** command enable coordinates to be quickly produced using repetitive patterns.

Illustration for Joint Coordinates

1 0 0 0 ; 2 2 0 0 ; 3 2 0 2 ; 4 0 0 2

5 4 0 0 ; 6 4 0 2 ; 7 6 0 0 ; 8 6 0 2

9 2 0 4 ; 10 0 0 4 ; 11 4 0 4 ; 12 6 0 4

The joint number is given above, followed by the X, Y and Z coordinates. As line separators, semicolon signs (;) are used. That allows us on one line to provide multiple sets of data. Node 6, for instance, has (X, Y, Z) co-ordinates of (4, 0, 2).

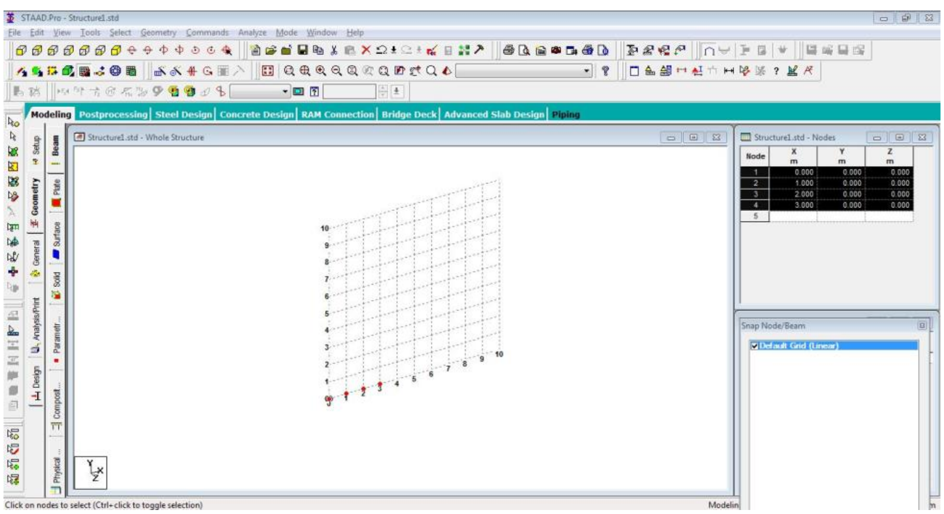


Figure 1. This figure is generated when the Joint Coordinates command is used.

b. **Member Incidences**

By defining the communication between joints, this set of commands is used to describe members. To promote the generation of repetitive patterns, **Repeat** and **Repeat All** commands are available.

The member/element incidence must be described so that the established model reflects only one structure, not two or more different structures.

Illustration for Member Incidences

1 1 2 3 4 ; 2 2 5 6 3 ; 3 5 7 8 6 ; 4 4 3 9 10 ;

5 3 6 11 9 ; 6 6 8 12 11

The incidences of elements are defined above. Element 3, for example, is defined as being linked between nodes 5, 7, 8 and 6.

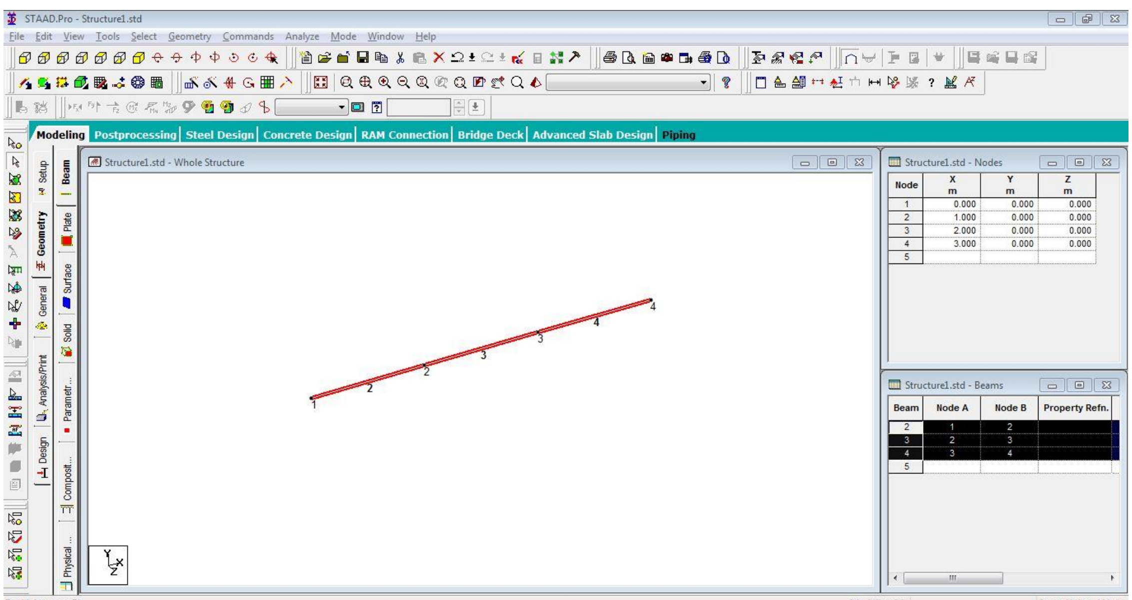


Figure 2. This figure is generated when the Member Incidences command is used.

c. **Member Property Specification**

This collection of commands can be used to define section properties for members of the frame.

The property assignment options fall under two broad categories:

1. Those that are listed, for example, for steel, aluminum, and timber, from the built-in property tables included with the software. (*see Figure 3.1,3.2)*

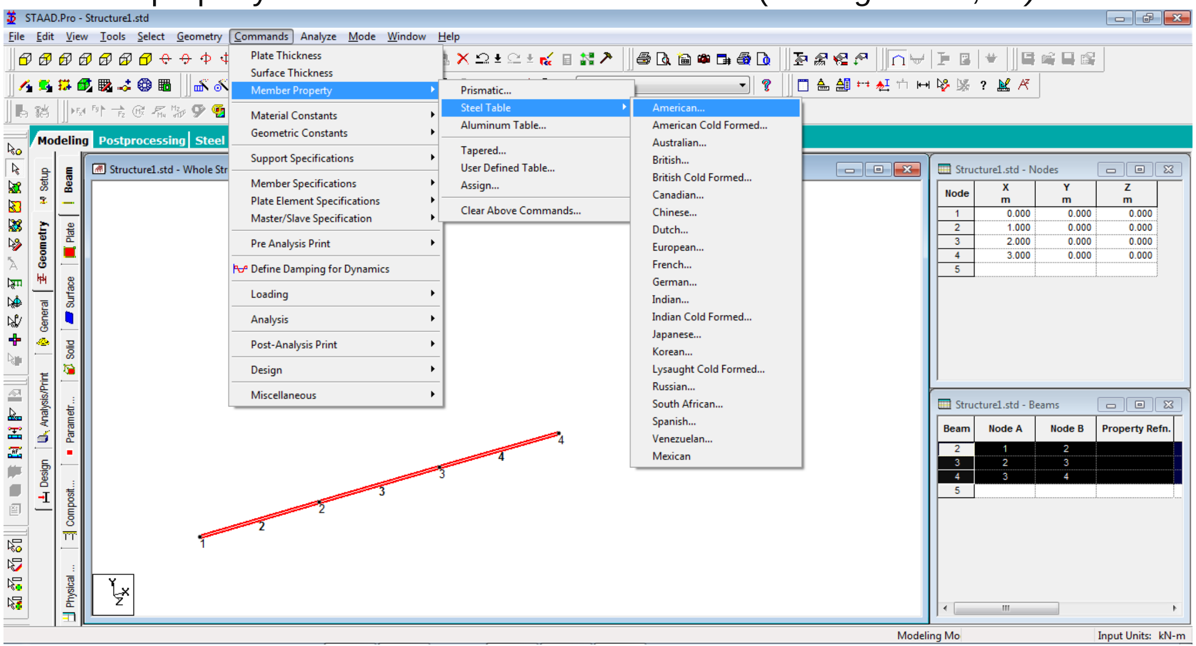


Figure 3.1

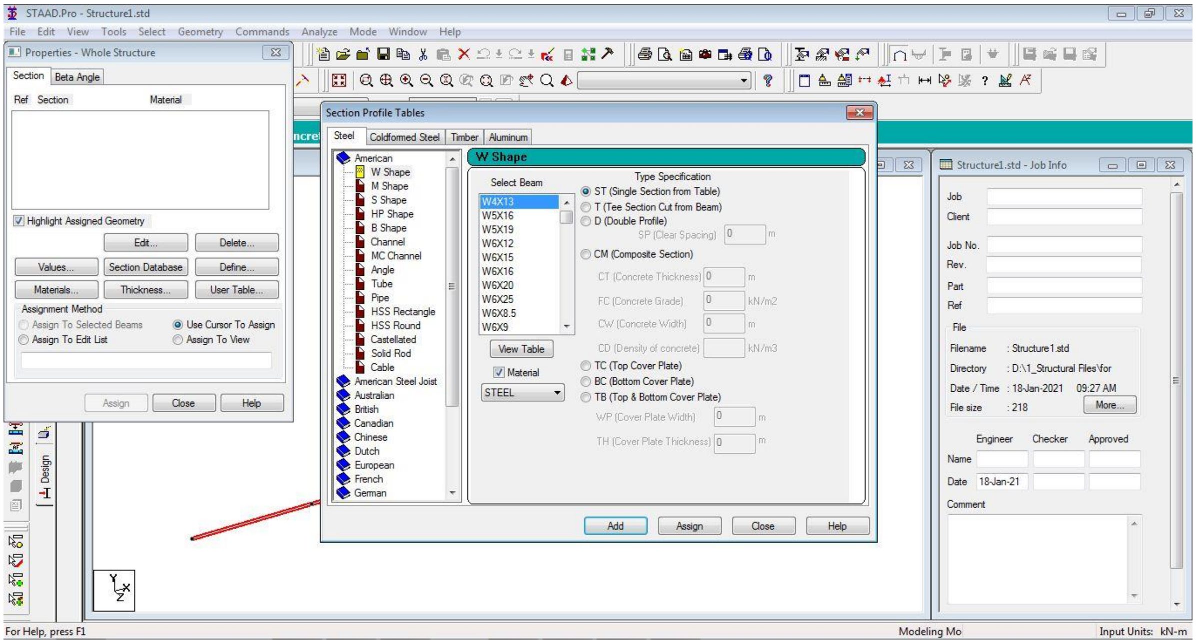


Figure 3.2 Properties which are specified from built-in property tables

2. Those that are not allocated from built-in tables are defined on a project-specific basis, such as concrete beams and columns, industrial structures, and custom-made parts. (*see Figure 3.3,3.4)*

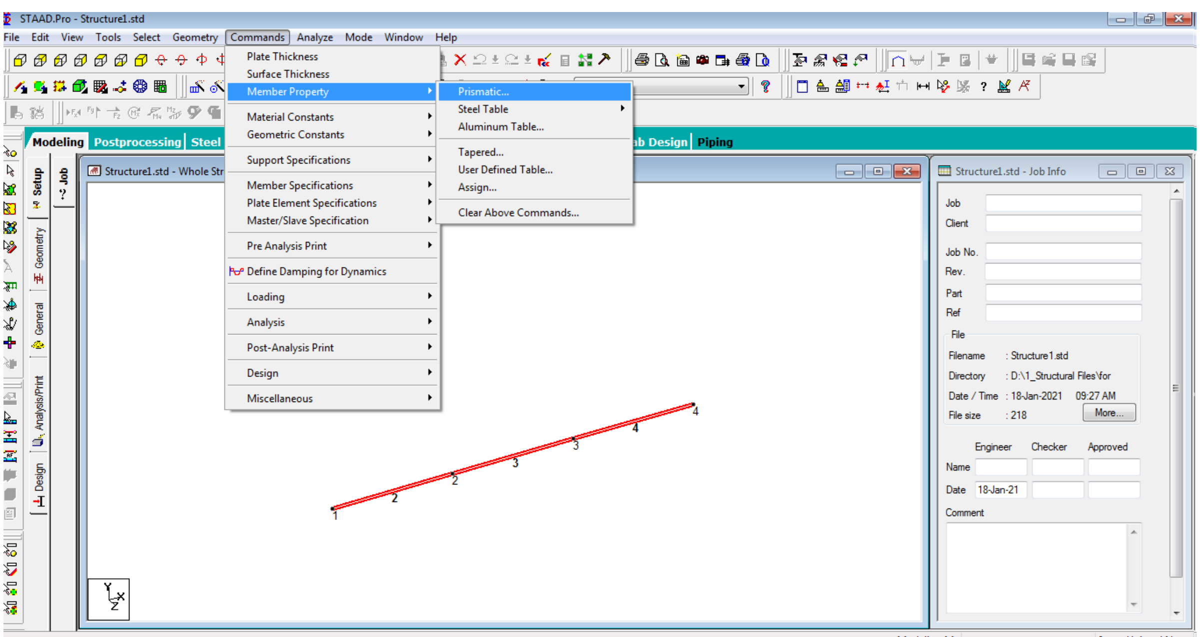


Figure 3.3

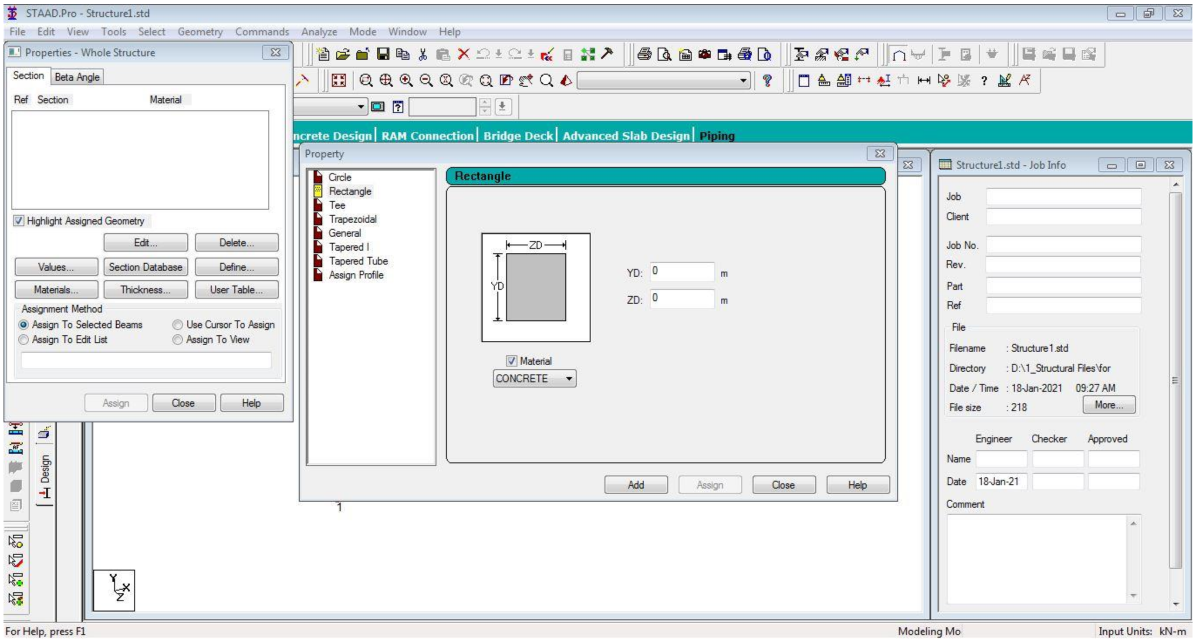


Figure 3.4 Properties that are not *specified from built-in property tables*

d. Types of Supports

STAAD.Pro provides facilities to assign support to both parallel and inclined orientation to global axes. The supports that can be specified are: **Pinned, Fixed, Fixed but, and Spring**.

A pinned support can resist translation movement from the definition but does not provide resistance to rotation. Fixed support can withstand all types of movement in all directions. Additionally, fixed supports have reactions for all forces and also for moments. In support, condition Fixed but is self-definitive. In every direction desired, but against forces and moments, i.e., Fx, Fy, Fz, Mx, My, Mz, and others, there are many facilities for release support.

Illustration:

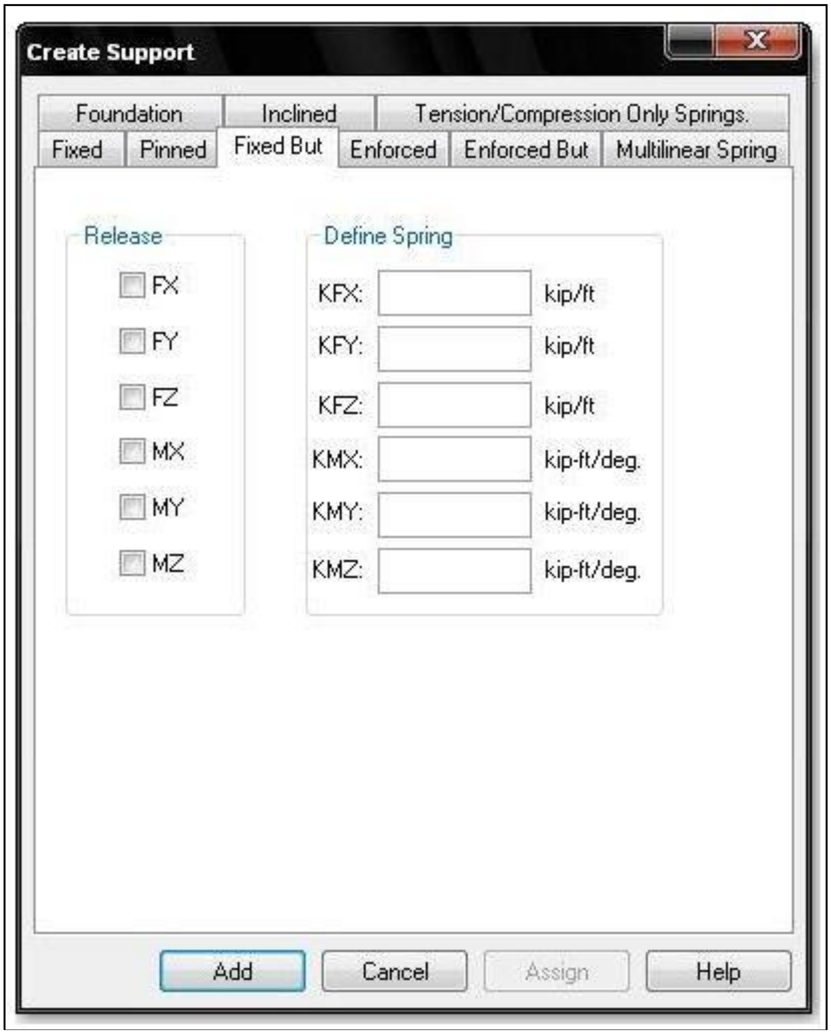


Figure 4. The figure shows the commands that can be used in specifying the support.

III. Types of Loads

Forces, deformations, or accelerations applied to structural components are structural loads or actions. Loads trigger structural stresses, deformations, and displacements. Structural analysis methods are used to analyze their effects. Categorized load into specific load group types like dead load, live load, wind load, seismic load, snow load, and user-defined.

a. **Dead loads (DL)** are static forces that are relatively constant for an extended time. They can be in tension or compression. In this load section, we have to define all the dead loads, which include the following: self-weight of all members, the weight of walls and other concentrated loads(member loads or point loads), floor loads such as the weight of slabs, loads of built-in furniture, utilities, and others.

b. **Live loads (DL)** are usually variable or moving loads. These can have a significant dynamic element and may involve considerations such as impact, momentum, vibration, and slosh dynamics of fluids. Live loads are those which we consider as moving (loads such as loads due to occupancy of living things and loads due to movable furniture).

c. **Wind loads**, typically, buildings are designed to resist a strong wind with a very long return period, such as 50 years or more. From historical records, the design wind speed is determined using extreme value theory to predict future excessive wind speeds. Wind speeds are generally calculated based on some regional design standards or standards.

d. **Seismic loads** are basic earthquake engineering concepts that imply applying a seismic oscillation to a structure. It usually happens either with the earth or with adjacent structures on the contact surfaces of a structure. Seismic loading depends primarily on earthquake vulnerability, site geotechnical parameters, and the structure's natural frequency.

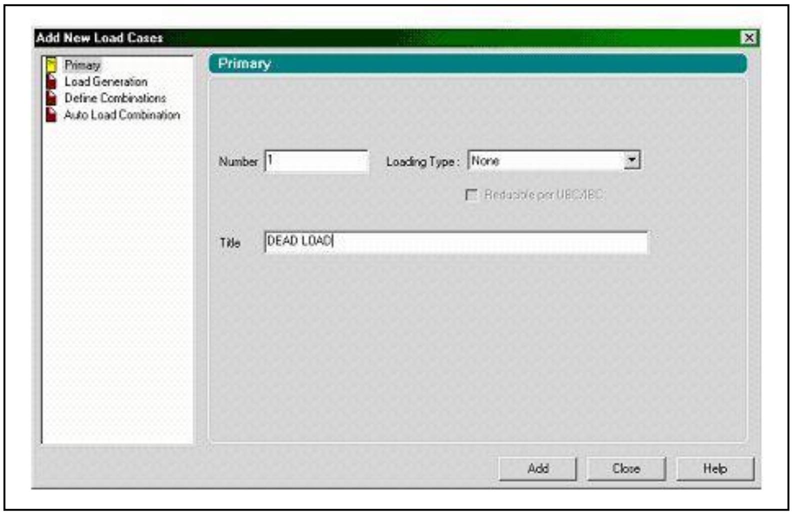


Figure 5. This window shows when the Load command is used.

IV. **Load Combination**

To build safe, serviceable structures as a structural engineer, magnitudes of different loads applied to the structure need to be estimated. The various forms of load used simultaneously should also be taken into account.

When more than one load type acts on the structure, a load combination results. To maintain the structure's safety under various maximum expected loading scenarios, building codes usually require some load combinations together with load factors (weightings) for each load type.

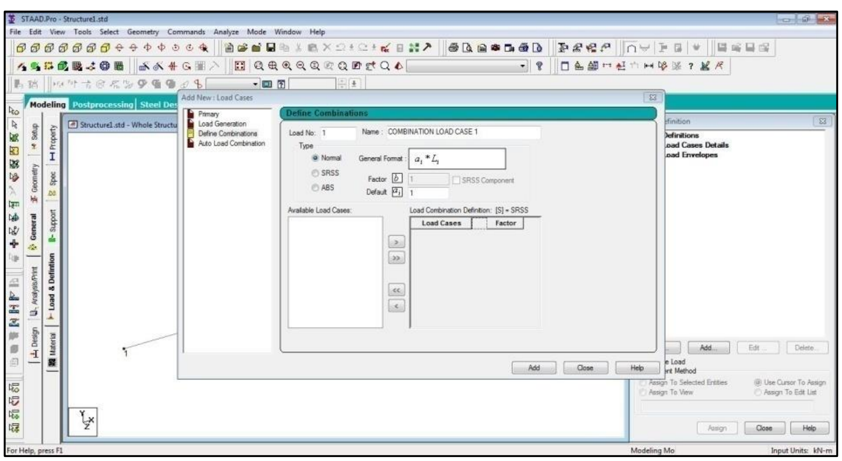
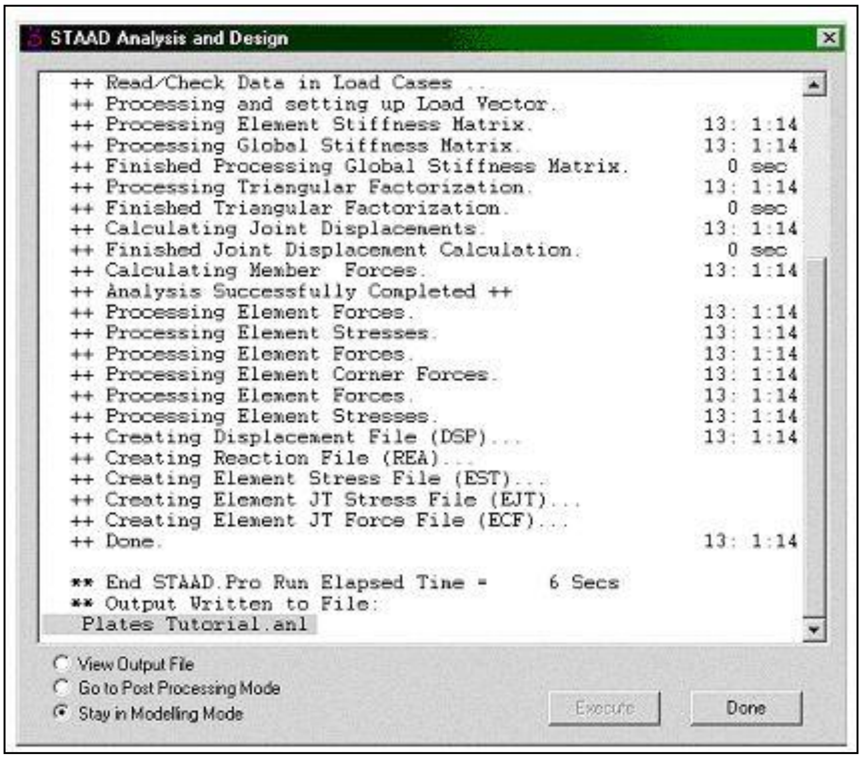


Figure 6. This window shows when the Load Combination command is used.

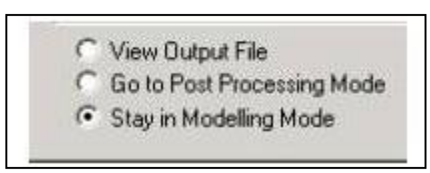
V. Run Structural Analysis

The model has to be analyzed to obtain the displacements, forces, stresses, and reactions in the structure due to the applied loads. If the pass-fail status of the members and elements per the requirements of steel and concrete codes is to be determined, it involves a design process. Both these processes are launched using the **Run Analysis** option from the Analyze menu.

As the analysis progresses, several messages are shown in the next figure.



At the end of these calculations, two activities take place. First, a Done button becomes active, and second, three options become available at the bottom left corner of this information window.



These options are indicative of what will happen after clicking the Done button.

The **View Output File** option allows us to view the STAAD generated output file. The output file contains the numerical results produced in response to the various input commands defined during model generation. It also tells whether any errors were encountered, and if so, whether the analysis and design were completed or not.

The **Go-To Post Processing Mode** option allows going to the graphical part of the post-processor program. This is where one can extensively verify the results, view the results graphically, plot result diagrams, and produce reports.

The **Stay in Modeling Mode** lets us continue to be in the Program generation mode of the program (the one we currently are in) if we wish to make further changes to our model.

a. Beam Stresses

Structural stresses are defined as fatigue-relevant quantities determined from the resulting internal forces and moments in the joint face of the weld spot, both in the structural portion and in the specimen (shear forces, cross-tension force, and bending moments, the torsional moment being neglected).

When internal particles react to each other, stress results from internal forces or forces that result, the measure of the amount of energy applied to an object is force. When a load is applied to an object, these internal forces are caused. Longitudinally-loaded objects, axially-loaded objects, and torsional-loaded, or twisted objects are the most common loading types.

Six major types of stress cause failures. Each of these different stresses is caused by a unique situation; the way the object is loaded dictates what kind of stress the object experiences. The two main ways forces can be applied to an object are axially or longitudinally. When an object is axially loaded, the forces are used in line with the object's central axis. With longitudinally-loaded structures, forces are applied, so they are perpendicular to the major axis.

Compression, tension, shear, bending, torsion, and fatigue are the six significant types of beam stresses.

1. **Compression stress** is the result of axially-loaded forces pointing towards the center of an object. There are two significant issues with compression stress: Compression stress has two big problems: compression forces can make an item shorten, or they can cause an object to buckling. If an object buckles, it bends so that it can no longer bear the load, although the object can hold more tension than is applied to it, structurally speaking.

2. **Tension stress** is caused when axially-loaded forces are pulling away from an object's center and perpendicular to the object's surface. Tension stress can

cause the lengthening of an object. There are some materials, such as concrete, where, when the object is in compression, the object can only withstand a fraction of the stress.

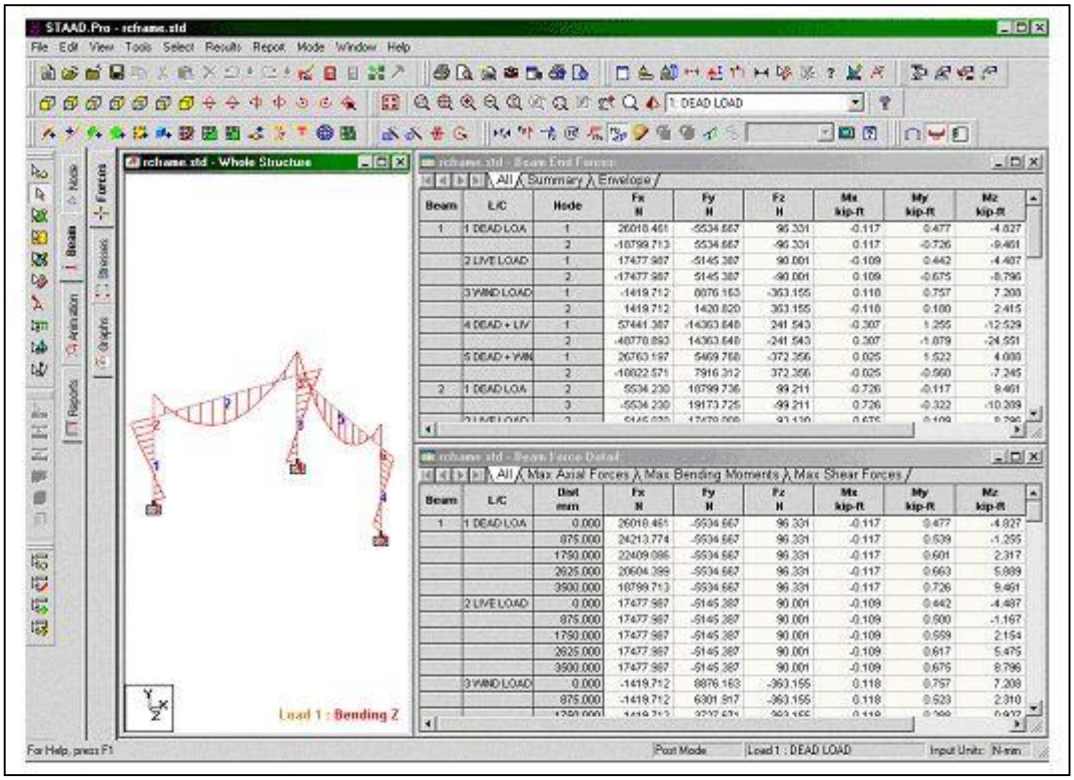
3. **Shear stress** is caused when the forces applied to an object are parallel to the object's cross-section. This stress can cause the object to deform and to fall apart in some cases. It shifts as the object deforms. The shape of the object will shift, which can impact how the object resists other forces.

4. **Bending stress** is seen in longitudinally-loaded objects. The forces cause the object to bend in a downward direction usually. The farther away from the object's fixed supports, the greater the bending stress.

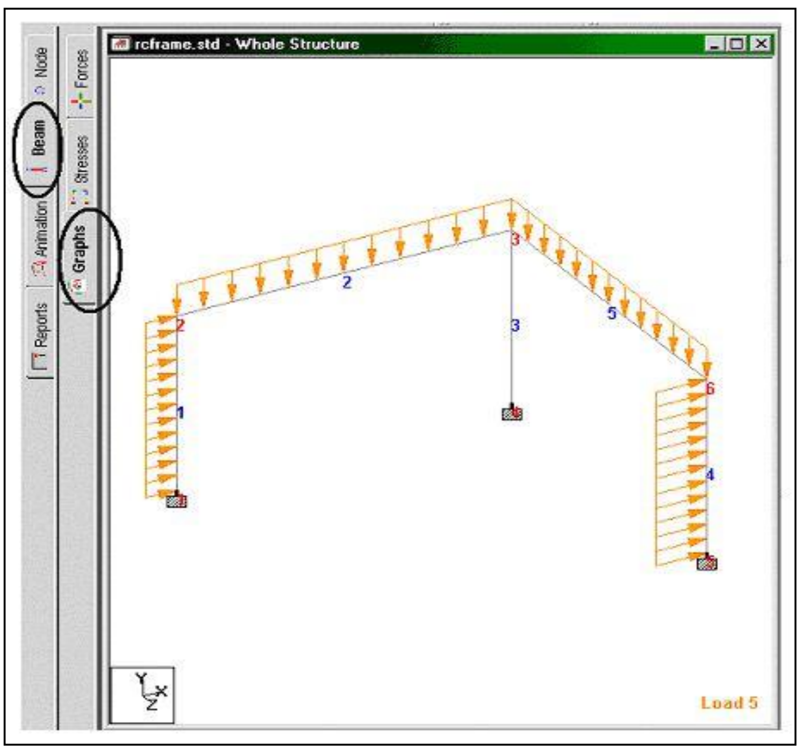
5. **Torsional shear stress or Torsional stress** is the shear stress produced in the shaft due to the twisting. The couple acting on it triggers this twisting in the pipe. A pair is two equal and opposite parallel forces acting as a pair on a body with a separate acting line.

6. **Fatigue** weakens a material caused by cyclic loading that results in structural damage, progressive and localized, and cracks formation. Once a fatigue crack has been initiated, it will grow a small amount with each loading cycle, typically producing striations on some parts of the fracture surface. The crack will continue to develop until it reaches a critical size, which happens when the crack's stress intensity factor exceeds the material's fracture power, producing rapid propagation and the structure's usually complete fracture.

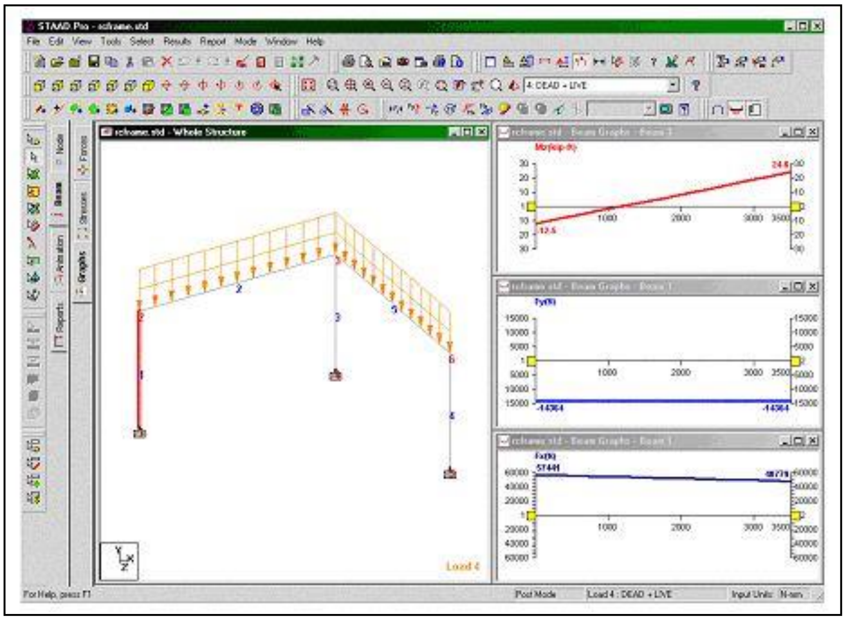
The **Beam Forces** Table pops up the Beam **|** Forces is selected from the page control area on the left side, shown in the figure below.



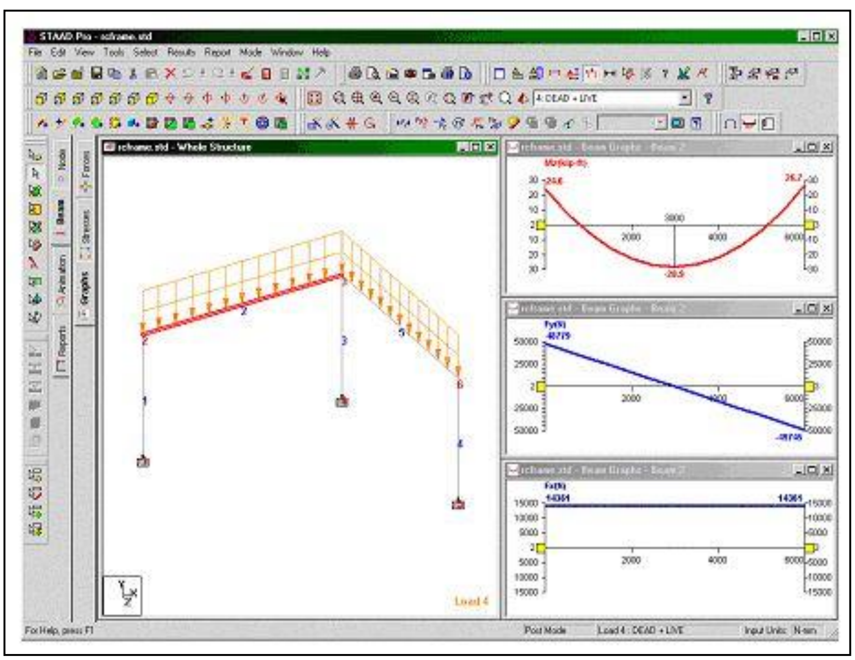
The **Graphs** page in the Post Processing Mode allows us to graphically view moments and forces such as Axial, Bending zz, Shear yy, and Combined Stresses for individual members. Select the Graphs sub-page nested under the Beam page from the screen's left side, as shown in the figure below.



The Main Window area shows the loading on the structure. The force/moment diagrams appear on the right side of the screen (see figure below). When a member is highlighted in the main window by clicking on it, the graphs are plotted for that member in the data area.



The following figure shows the graphs plotted for member 2 for the same load case.



b. Column Stresses

Columns are commonly known as vertical structural elements but may be placed in any direction, such as in a truss, diagonal and horizontal compression elements. In trusses, building frames, and sub-structure supports for bridges; columns are used as main components.

The most significant parts of a structure are columns. Columns protect compression loads from roofs, floors or bridge decks. They transfer loads of the structure to the surrounding soil through the foundation. So, buildings should have strong columns. Otherwise, failure will occur.

Two building materials, concrete, and steel are used to create columns. Before designing the columns, civil engineers should calculate total stress due to the building's live load and dead load. The structure will collapse when the applied pressure reaches the allowable (calculated) stress. Different types of column failures are Compression and Buckling Failure.

Compression failure happens when columns are axially loaded, the concrete and steel will experience some stresses. When the loads are more significant than the cross-section of all areas of the column, the concrete and steel will reach the yield stress, and failure will start without any deformation.

In extended columns, buckling failure occurs typically because they are very slender and have the least lateral dimension. The load-carrying capacity of the column decreases very drastically in such a situation.

Also, under small loads, the columns appear to become unstable and begin to buckle sideways. That implies that the concrete and steel reached their yield stress for even small loads and start failing due to lateral buckling. By not constructing long columns of slenderness ratio, this form of failure can be avoided.

The **Beam End Forces** table window has three tabs: All, Summary, and Envelope.

rcframe.std - Beam End Forces:

All | Summary | Envelope

Beam	L/C	Node	Fx N	Fy N	Fz N
1	1 DEAD LOA	1	26018.465	-5534.667	96.331
		2	-18799.715	5534.667	-96.331
	2 LIVE LOAD	1	17477.987	-5145.387	90.001
		2	-17477.987	5145.387	-90.001
	3 WIND LOAD	1	-1419.712	8876.163	-363.155
		2	1419.712	1420.820	363.155

All - This tab lists all forces and moments corresponding to all 6 degrees of freedom for all selected load cases at each elected member's start and end.

rcframe.std - Beam End Forces:

All | Summary | Envelope

Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx MTon-m	My MTon-m	Mz MTon-m
1	1 DEAD LOA	1	26.018	-5.535	0.096	-0.016	0.066	-0.667
		2	-18.800	5.535	-0.096	0.016	-0.100	-1.308
	2 LIVE LOAD	1	17.478	-5.145	0.090	-0.015	0.061	-0.620
		2	-17.478	5.145	-0.090	0.015	-0.093	-1.216
	3 WIND LOAD	1	-1.420	8.876	-0.363	0.016	0.105	0.996
		2	1.420	1.421	0.363	-0.016	0.025	0.334
	4 DEAD + LIV	1	57.441	-14.364	0.242	-0.042	0.174	-1.732
		2	-48.779	14.364	-0.242	0.042	-0.260	-3.394
	5 DEAD + WIN	1	26.763	5.470	-0.372	0.003	0.210	0.565
		2	-18.823	7.916	0.372	-0.003	-0.077	-1.002
2	1 DEAD LOA	2	5.534	18.800	0.099	-0.100	-0.016	1.308
		3	-5.534	19.174	-0.099	0.100	-0.044	-1.422
	2 LIVE LOAD	2	5.145	17.478	0.092	-0.093	-0.015	1.216
		3	-5.145	17.826	-0.092	0.093	-0.041	-1.322
	3 WIND LOAD	2	1.420	-1.420	-0.363	0.025	0.016	-0.334
		3	-1.420	1.420	0.363	-0.025	0.206	-0.535
	4 DEAD + LIV	2	14.361	48.779	0.259	-0.260	-0.042	3.394
		3	-14.361	49.745	-0.259	0.260	-0.116	-3.690
	5 DEAD + WIN	2	7.925	18.823	-0.366	-0.077	0.003	1.002
		3	-7.925	22.946	0.366	0.077	0.220	-2.264
3	1 DEAD LOA	3	38.348	5.434	-5.434	-0.000	1.322	1.322
		4	-46.766	-5.434	5.434	0.000	0.617	0.617
	2 LIVE LOAD	3	35.652	5.052	-5.052	0.000	1.229	1.229
		4	-35.652	-5.052	5.052	-0.000	0.574	0.574
	3 WIND LOAD	3	1.474	2.295	-0.227	-0.143	0.039	0.251
		4	-1.474	-2.295	0.227	0.143	0.042	0.568
	4 DEAD + LIV	3	99.490	14.097	-14.097	-0.000	3.430	3.430

Summary - The maximum and minimum values (forces and moments) for each degree of freedom are shown in this tab, shown in the figure below. It takes into consideration all beams and all load cases defined during the Results Setup. With the corresponding occurrence node and load case number (L/C), maximum values for all freedom degrees are presented.

rcframe.std - Beam End Forces:

All | Summary | Envelope

	Beam	L/C	Node	Fx kN	Fy kN	Fz kN	Mx MTon-m	My MTon-m	Mz MTon-m
Max Fx	3	4 DEAD + LIV	4	**109.593**	14.097	-14.097	-0.000	-1.601	-1.601
Min Fx	1	3 WIND LOAD	1	**-1.420**	8.876	-0.363	0.016	0.105	0.996
Max Fy	2	4 DEAD + LIV	2	14.361	**48.779**	0.259	-0.260	-0.042	3.394
Min Fy	2	4 DEAD + LIV	3	14.361	**-49.745**	0.259	-0.260	0.116	3.690
Max Fz	5	5 DEAD + WIN	6	5.909	20.609	**1.035**	0.482	-0.225	1.413
Min Fz	4	5 DEAD + WIN	5	28.549	-5.909	**-21.328**	-0.225	3.148	-0.696
Max Mx	5	5 DEAD + WIN	6	5.909	20.609	1.035	**0.482**	-0.225	1.413
Min Mx	2	4 DEAD + LIV	2	14.361	48.779	0.259	**-0.260**	-0.042	3.394
Max My	3	4 DEAD + LIV	3	99.490	14.097	-14.097	-0.000	**3.430**	3.430
Min My	3	4 DEAD + LIV	4	109.593	14.097	-14.097	-0.000	**-1.601**	-1.601
Max Mz	2	4 DEAD + LIV	3	14.361	-49.745	0.259	-0.260	0.116	**3.690**
Min Mz	1	4 DEAD + LIV	1	57.441	-14.364	0.242	-0.042	0.174	**-1.732**

Envelope - This tab displays a table consisting, for each member, of the maximum and minimum for each degree of freedom and the load case responsible for each of those values.

rcframe.std - Beam End Forces:

All | Summary | Envelope

Beam	Node	Env	Fx kN	Fy kN	Fz kN	Mx MTon-m	My MTon-m	Mz MTon-m
1	1	+ve	57.441	8.876	0.242	0.016	0.210	0.996
			4 DEAD + LIV	3 WIND LOAD	4 DEAD + LIV	3 WIND LOAD	5 DEAD + WI	3 WIND LOAD
		-ve	-1.420	-14.364	-0.372	-0.042	0.000	-1.732
			3 WIND LOAD	4 DEAD + LIV	5 DEAD + WI	4 DEAD + LIV	-	4 DEAD + LIV
1	2	+ve	48.779	0.000	0.242	0.016	0.260	3.394
			4 DEAD + LIV	-	4 DEAD + LIV	3 WIND LOAD	4 DEAD + LIV	4 DEAD + LIV
		-ve	-1.420	-14.364	-0.372	-0.042	-0.025	-0.334
			3 WIND LOAD	4 DEAD + LIV	5 DEAD + WI	4 DEAD + LIV	3 WIND LOAD	3 WIND LOAD
2	2	+ve	14.361	48.779	0.259	0.025	0.016	3.394
			4 DEAD + LIV	4 DEAD + LIV	4 DEAD + LIV	3 WIND LOAD	3 WIND LOAD	4 DEAD + LIV
		-ve	0.000	-1.420	-0.366	-0.260	-0.042	-0.334
			-	3 WIND LOAD	5 DEAD + WI	4 DEAD + LIV	4 DEAD + LIV	3 WIND LOAD
2	3	+ve	14.361	0.000	0.259	0.025	0.116	3.690
			4 DEAD + LIV	-	4 DEAD + LIV	3 WIND LOAD	4 DEAD + LIV	4 DEAD + LIV
		-ve	0.000	-49.745	-0.366	-0.260	-0.220	0.000
			-	4 DEAD + LIV	5 DEAD + WI	4 DEAD + LIV	5 DEAD + WI	-
3	3	+ve	99.490	14.097	0.000	0.000	3.430	3.430
			4 DEAD + LIV	4 DEAD + LIV	-	2 LIVE LOAD	4 DEAD + LIV	4 DEAD + LIV
		-ve	0.000	0.000	-14.097	-0.187	0.000	0.000
			-	-	4 DEAD + LIV	5 DEAD + WI	-	-
3	4	+ve	109.593	14.097	0.000	0.000	0.000	0.000
			4 DEAD + LIV	4 DEAD + LIV	-	2 LIVE LOAD	-	-
		-ve	0.000	0.000	-14.097	-0.187	-1.601	-1.601
			-	-	4 DEAD + LIV	5 DEAD + WI	4 DEAD + LIV	4 DEAD + LIV
4	5	+ve	57.441	0.136	0.000	0.042	3.148	0.029
			4 DEAD + LIV	3 WIND LOAD	-	4 DEAD + LIV	5 DEAD + WI	3 WIND LOAD
		-ve	-0.054	-14.364	-21.328	-0.225	-0.174	-1.732

c. Support Reaction

A member that allows other members to withstand loads is supported in a structure. Different types of supports, their reactions and structural applications, and their details are discussed. Supports the load to the ground in a structure and provides the structure supported on it with stability.

External supports are supports that are generally provided externally without disrupting the structural members. Different types of external supports are Fixed Support, Pinned Support or Hinged Support, Roller Support, Rocker Support, and Simple Support.

1. **Fixed Supports** are also called rigid supports. Both rotation and translation are restricted against fixed supports to resist any force or moment. In structural analysis, there are three unknowns to find for fixed support that can satisfy all three equilibrium equations. To provide good stability to the structure, at least one rigid support should be provided. Beam fixed in a wall is an excellent example of improved support. The figure below is an example of a Fixed Support Structure.

Fixed Supports Structure

2. **Pinned Supports or Hinged Supports** can resist both vertical and horizontal forces, but they cannot resist. This means hinged support is restrained against translation. Using equations of equilibrium, one can find out the components of horizontal and vertical forces. The most common example for hinged support is the door leaf, which only rotates about its vertical axis without any horizontal or vertical movement.

   The rotation of pinned support or hinged support is allowed in only one direction and is resisted in another order. Hinged supports are also used in three hinged arched bridges with two supports at ends, and a third hinge is provided at the center of the arch, which is called an internal hinge.

HingedSupport of Sydney Harbor Bridge

3. **Roller Supports** only resists perpendicular forces, and they cannot resist parallel or horizontal forces and moment. It means that without resisting horizontal force, the roller support can move freely along the surface. This type of support is provided at one end of bridge spans.

   Providing roller support at one end is to allow contraction or expansion of the bridge deck concerning temperature differences in the atmosphere. If roller support is not provided, it will cause severe damage to the bridge banks. However, this horizontal force should be resisted by at least one support to provide stability, so roller support should be provided at one end only, not at both ends.

Hinged Support in a Structure

4. **Rocker Support** is similar to supporting the roller. It also resists vertical force and enables rotation and horizontal translation. But in this situation, as shown in the figure below, horizontal movement is due to the curved surface provided at the bottom. So, in this case, the amount of horizontal direction is limited.

Rocker Support in a Structure

5. **Simple support** is just support on which structural member rests. They cannot resist lateral movement and moment like roller supports. With the help of gravity, they only resist vertical movement of support.

   The horizontal or lateral motion allowed is limited to a limited extent, and the structure loses its support after that. It’s like a brick resting longitudinally on two bricks. This type of support is not commonly used for structural purposes. However, simple support structures can be used in areas with frequent seismic activity.

Simple Support in a Structure

VI. Application of Structural Analysis Using Computer Software

The Structural Analysis Application using Computer Software is applied to the Proposed 3 Storey Reinforced Concrete Building constructed at Bansalan, Davao del Sur.

A. Structural Analysis Using Computer Software STAAD

STAAD Pro programming is commonly used as part of the structural analysis and design of structures such as towers, facilities, bridges, transport infrastructure, service, and industrial structures

B. Modeling the Structural Geometry

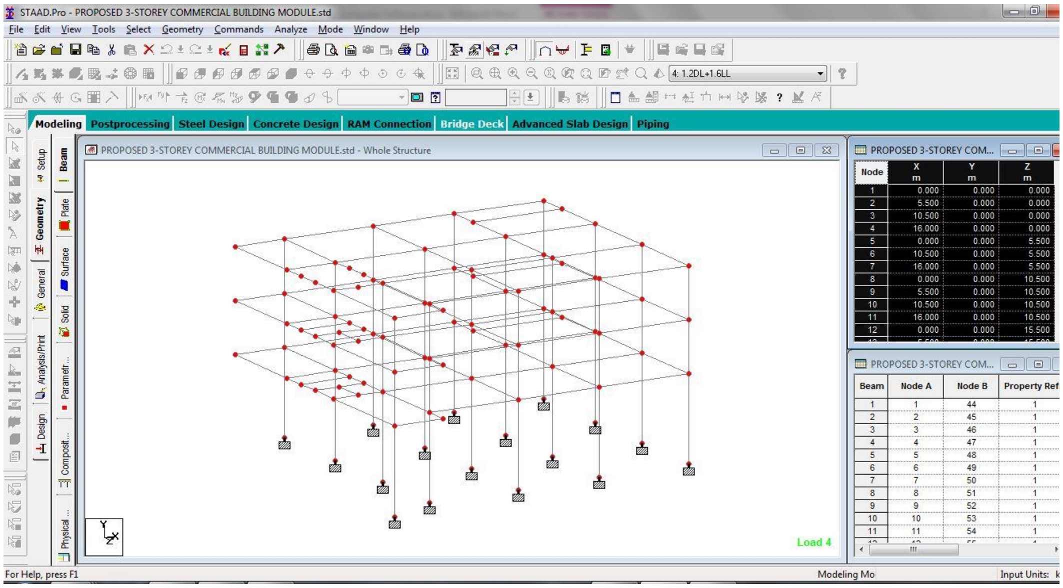
a. **Joint Coordinates**

Input Editor for Joint Coordinates

```
JOINT COORDINATES
1 0 0 0; 2 5.5 0 0; 3 10.5 0 0; 4 16 0 0; 5 0 0 5.5; 6 10.5 0 5.5; 7 16 0 5.5;
8 0 0 10.5; 9 5.5 0 10.5; 10 10.5 0 10.5; 11 16 0 10.5; 12 0 0 15.5;
13 5.5 0 15.5; 14 10.5 0 15.5; 15 16 0 15.5; 16 -3 0 17; 43 5.5 0 5.5;
44 0 5 0; 45 5.5 5 0; 46 10.5 5 0; 47 16 5 0; 48 0 5 5.5; 49 10.5 5 5.5;
50 16 5 5.5; 51 0 5 10.5; 52 5.5 5 10.5; 53 10.5 5 10.5; 54 16 5 10.5;
55 0 5 15.5; 56 5.5 5 15.5; 57 10.5 5 15.5; 58 16 5 15.5; 59 -3 5 17;
60 5.5 5 5.5; 61 0 5 17; 62 -3 5 0; 63 -3 5 5.5; 64 -3 5 10.5; 65 0 5 8.5;
66 -3 5 8.5; 67 0 5 7; 68 -3 5 7; 69 -1.5 5 8.5; 70 -1.5 5 10.5; 71 0 8 0;
72 5.5 8 0; 73 10.5 8 0; 74 16 8 0; 75 0 8 5.5; 76 10.5 8 5.5; 77 16 8 5.5;
78 0 8 10.5; 79 5.5 8 10.5; 80 10.5 8 10.5; 81 16 8 10.5; 82 0 8 15.5;
83 5.5 8 15.5; 84 10.5 8 15.5; 85 16 8 15.5; 86 -3 8 17; 87 5.5 8 5.5;
88 0 8 17; 89 -3 8 0; 90 -3 8 5.5; 91 -3 8 10.5; 92 0 8 8.5; 93 -3 8 8.5;
94 0 8 7; 95 -3 8 7; 96 -1.5 8 8.5; 97 -1.5 8 10.5; 98 0 11 0; 99 5.5 11 0;
100 10.5 11 0; 101 16 11 0; 102 0 11 5.5; 103 10.5 11 5.5; 104 16 11 5.5;
105 0 11 10.5; 106 5.5 11 10.5; 107 10.5 11 10.5; 108 16 11 10.5;
109 0 11 15.5; 110 5.5 11 15.5; 111 10.5 11 15.5; 112 16 11 15.5; 113 -3 11 17;
114 5.5 11 5.5; 115 0 11 17; 116 -3 11 0; 117 -3 11 5.5; 118 -3 11 10.5;
119 0 11 8.5; 120 -3 11 8.5; 121 0 11 7; 122 -3 11 7; 124 -1.5 11 10.5;
125 0 13 0; 126 5.5 13 0; 127 10.5 13 0; 128 16 13 0; 131 16 13 5.5;
135 16 13 10.5; 136 0 13 15.5; 137 5.5 13 15.5; 138 10.5 13 15.5;
139 16 13 15.5; 140 0 13 5.5; 141 0 13 10.5; 142 10.5 5 2; 143 16 5 2;
144 10.5 8 2; 145 16 8 2; 146 10.5 11 2; 147 16 11 2;
```

The figure below illustrates the Joint Coordinate of the structural model that selects all nodes in the model.

Illustration for Joint Coordinates:



b. **Member Incidences**

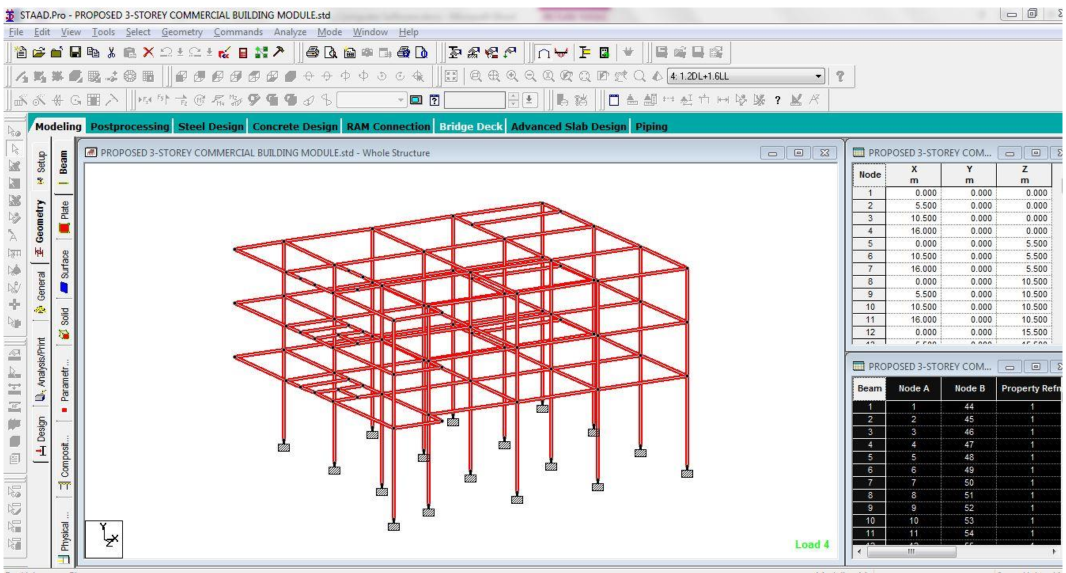
Input Editor for Member Incidences

MEMBER INCIDENCES
1 1 44; 2 2 45; 3 3 46; 4 4 47; 5 5 48; 6 6 49; 7 7 50; 8 8 51; 9 9 52;
10 10 53; 11 11 54; 12 12 55; 13 13 56; 14 14 57; 15 15 58; 16 16 59; 17 43 60;
18 62 44; 19 44 45; 20 45 46; 21 46 47; 22 63 48; 23 48 60; 24 60 49; 25 49 50;
26 68 67; 27 66 69; 28 69 65; 29 64 70; 30 51 52; 31 52 53; 32 53 54; 33 55 56;
34 56 57; 35 57 58; 36 59 61; 37 62 63; 38 44 48; 39 45 60; 40 46 142;
41 47 143; 42 63 68; 43 48 67; 44 68 66; 45 67 65; 46 66 64; 47 65 51;
48 60 52; 49 49 53; 50 50 54; 51 64 59; 52 51 55; 53 52 56; 54 53 57; 55 54 58;
56 55 61; 57 70 51; 58 44 71; 59 45 72; 60 46 73; 61 47 74; 62 48 75; 63 49 76;
64 50 77; 65 51 78; 66 52 79; 67 53 80; 68 54 81; 69 55 82; 70 56 83; 71 57 84;
72 58 85; 73 59 86; 74 60 87; 85 89 71; 86 71 72; 87 72 73; 88 73 74; 89 90 75;
90 75 87; 91 87 76; 92 76 77; 93 95 94; 94 93 96; 95 96 92; 96 91 97; 97 78 79;
98 79 80; 99 80 81; 100 82 83; 101 83 84; 102 84 85; 103 86 88; 104 89 90;
105 71 75; 106 72 87; 107 73 144; 108 74 145; 109 90 95; 110 75 94; 111 95 93;
112 94 92; 113 93 91; 114 92 78; 115 87 79; 116 76 80; 117 77 81; 118 91 86;
119 78 82; 120 79 83; 121 80 84; 122 81 85; 123 82 88; 124 97 78; 125 69 70;
126 96 97; 127 71 98; 128 72 99; 129 73 100; 130 74 101; 131 75 102;
132 76 103; 133 77 104; 134 78 105; 135 79 106; 136 80 107; 137 81 108;
138 82 109; 139 83 110; 140 84 111; 141 85 112; 142 86 113; 143 87 114;

154 116 98; 155 98 99; 156 99 100; 157 100 101; 158 117 102; 159 102 114; 160 114 103; 161 103 104; 165 118 124; 166 105 106; 167 106 107; 168 107 108; 169 109 110; 170 110 111; 171 111 112; 172 113 115; 173 116 117; 174 98 102; 175 99 114; 176 100 146; 177 101 147; 178 117 122; 179 102 121; 180 122 120; 181 121 119; 182 120 118; 183 119 105; 184 114 106; 185 103 107; 186 104 108; 187 118 113; 188 105 109; 189 106 110; 190 107 111; 191 108 112; 192 109 115; 193 124 105; 195 98 125; 196 99 126; 197 100 127; 198 101 128; 201 104 131; 205 108 135; 206 109 136; 207 110 137; 208 111 138; 209 112 139; 210 102 140; 211 105 141; 212 142 49; 213 143 50; 214 142 143; 215 144 76; 216 145 77; 217 144 145; 218 146 103; 219 147 104; 220 146 147;

The diagram below illustrates the Member Incidences of the structural model that selects all beams in the model.

Illustration for Member Incidences:



c. **Member Property Specification**

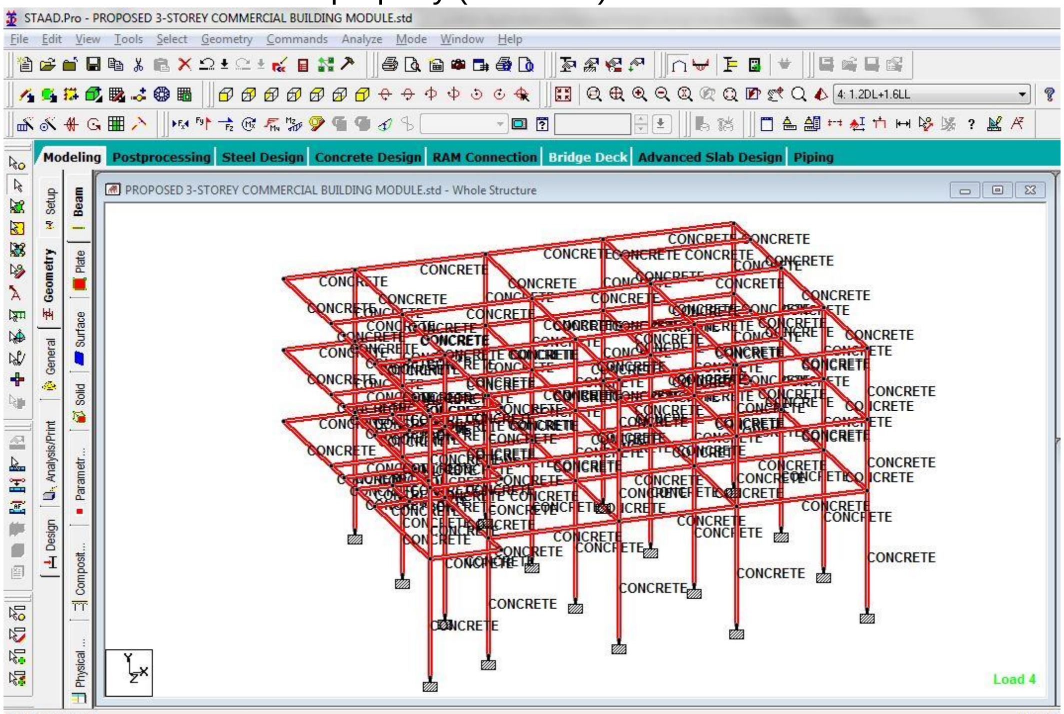
Member Property operation defines the section if the member is concrete, steel, or aluminum.

Input Editor for Member Property

```
MEMBER PROPERTY AMERICAN
1 TO 15 17 58 TO 72 74 127 TO 141 143 195 TO 198 201 205 TO 210 -
211 PRIS YD 0.4 ZD 0.4
19 TO 21 23 TO 25 30 TO 35 38 TO 41 43 45 47 TO 50 52 TO 55 212 TO 214
217 -220 PRIS YD 0.5 ZD 0.3
18 22 26 TO 29 36 37 42 44 46 51 56 57 125 PRIS YD 0.4 ZD 0.3
86 TO 88 90 TO 92 97 TO 102 105 TO 108 110 112 114 TO 117 119 TO 122
215 -216 PRIS YD 0.45 ZD 0.3
85 89 93 TO 96 103 104 109 111 113 118 123 124 126 PRIS YD 0.35 ZD 0.3
155 TO 157 159 TO 161 166 TO 171 174 TO 177 179 181 183 TO 186 188 TO
191 -218 219 PRIS YD 0.4 ZD 0.3
154 158 165 172 173 178 180 182 187 192 193 PRIS YD 0.3 ZD 0.3
16 73 142 PRIS YD 0.6 ZD 0.43
CONSTANTS
MATERIAL CONCRETE ALL
```

The illustration below shows that we choose concrete for member properties in the structure.

Illustration for Member property (Concrete)



d. Types of Supports

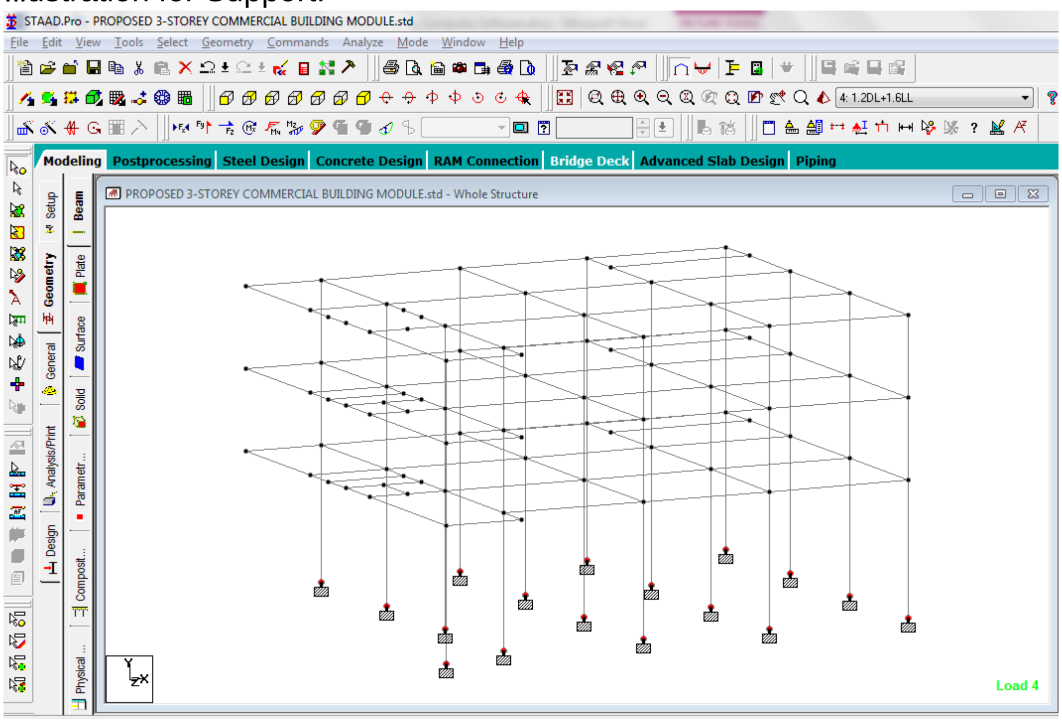
For Support, that can be specified: **Pinned, Fixed, Fixed but, and Spring.** Depending on your design, you can choose what support you select in the structure.

Input Editor for Support

SUPPORTS
1 TO 16 43 FIXED

From the illustration shown in the drawing, we use fixed support in the design model.

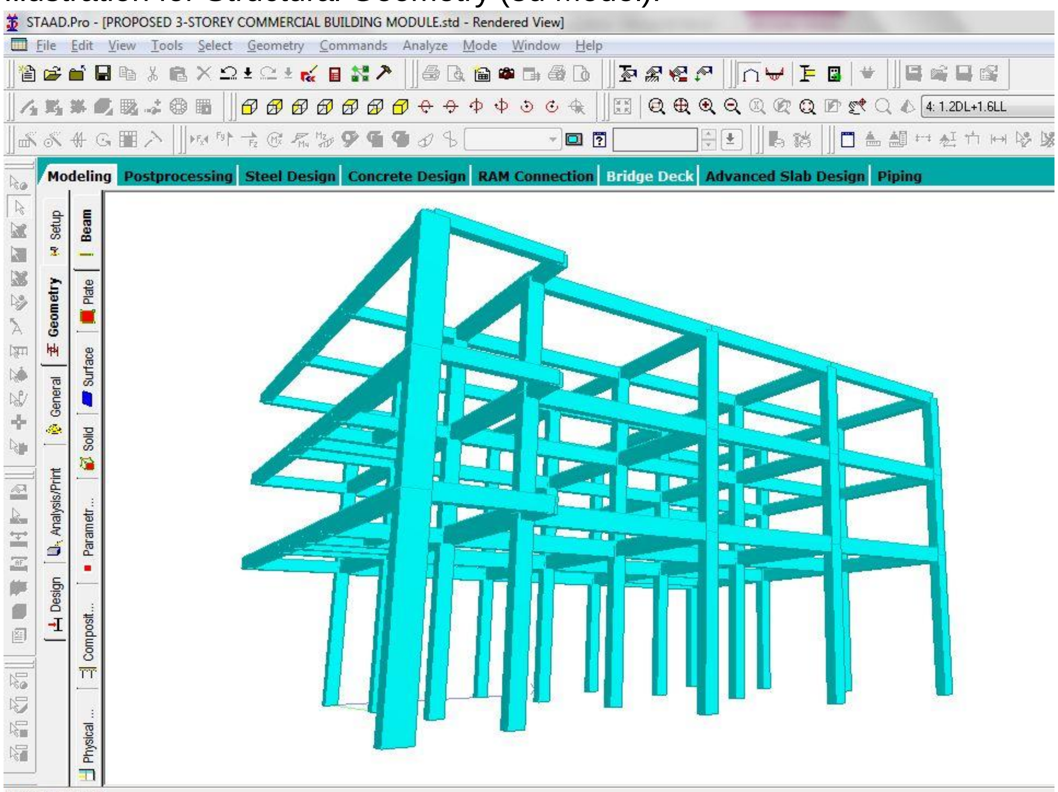
Illustration for Support:



Structural Geometry

The structure geometry consists of **Joint Coordinates**, **Member Incidences**, and **Member Property**. With this command now, we can see the structural geometry in the model.

Illustration for Structural Geometry (3d Model):



C. Types of Loads

Input Editor for Dead Load

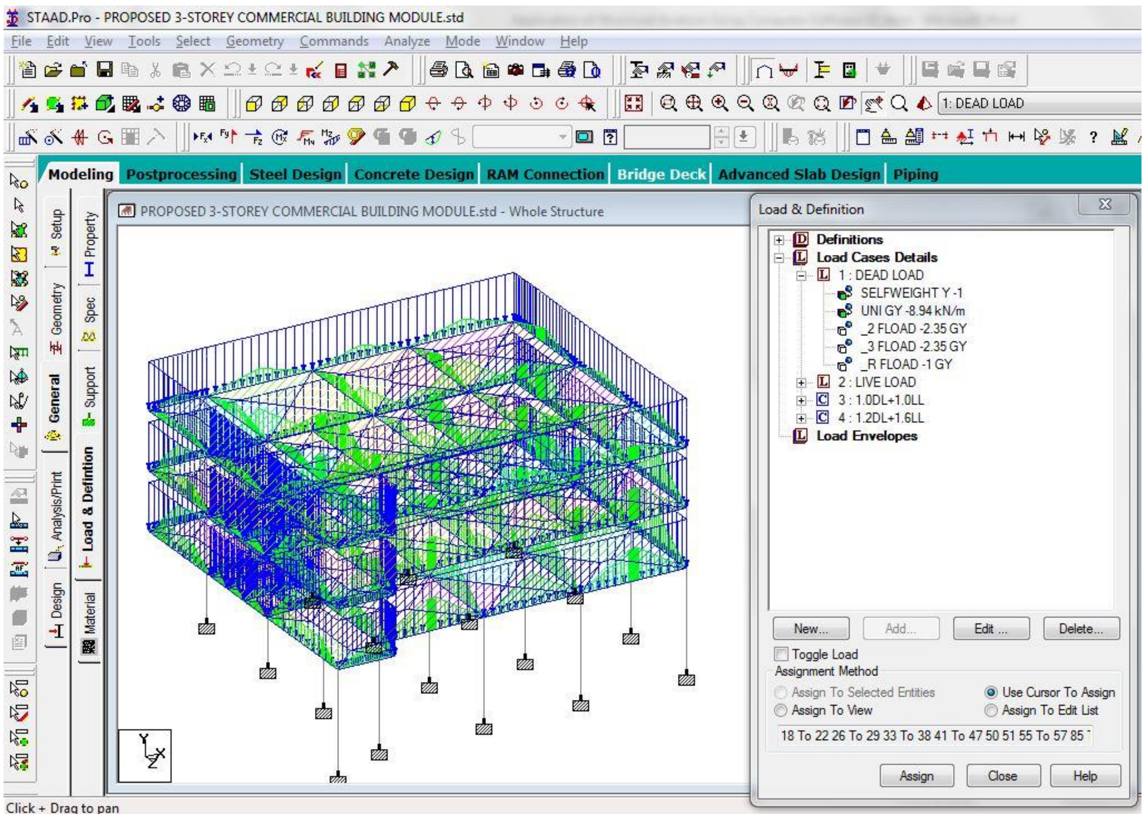
LOAD 1 LOAD TYPE Dead TITLE DEAD LOAD
SELFWEIGHT Y -1 LIST 1 TO 74 85 TO 143 154 TO 161 165 TO 193 195 TO 198 201 - 205 TO 211
MEMBER LOAD
18 TO 22 26 TO 29 33 TO 38 41 TO 47 50 51 55 TO 57 85 TO 89 93 TO 96 - 100 TO 105 108 TO 114 117 118 122 TO 126 154 TO 157 169 TO 173 177 178 180 -182 186 187 191 192 213 216 219 UNI GY -8.94

FLOOR LOAD
_2 FLOAD -2.35 GY
_3 FLOAD -2.35 GY
_R FLOAD -1 GY

a. **Dead Load**

Dead Loads are load static and self-weight of the structure all loading base on **NSCP 2015** minimum Design load Dead Loads.

Illustration for Dead Load:



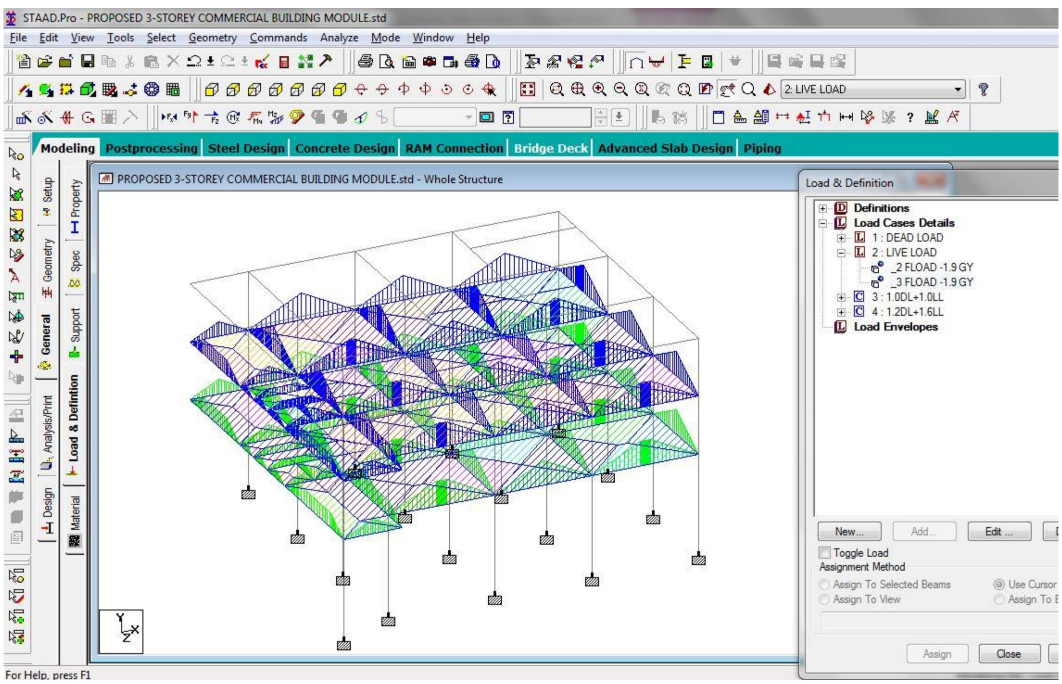
b. **Live Load**

Input Editor for Live Load

LOAD 2 LOAD TYPE Live TITLE LIVE LOAD
FLOOR LOAD
_2 FLOAD -1.9 GY
_3 FLOAD -1.9 GY

Live loads are usually variable or moving loads. These can have a significant dynamic element and may involve considerations such as impact, momentum, vibration, and slosh dynamics of fluids.

Illustration for Live Load:



D. **Load Combination**

Load Combination Dead Load and Live Load Combination to get max result load for the structure to ensure the structure's safety.

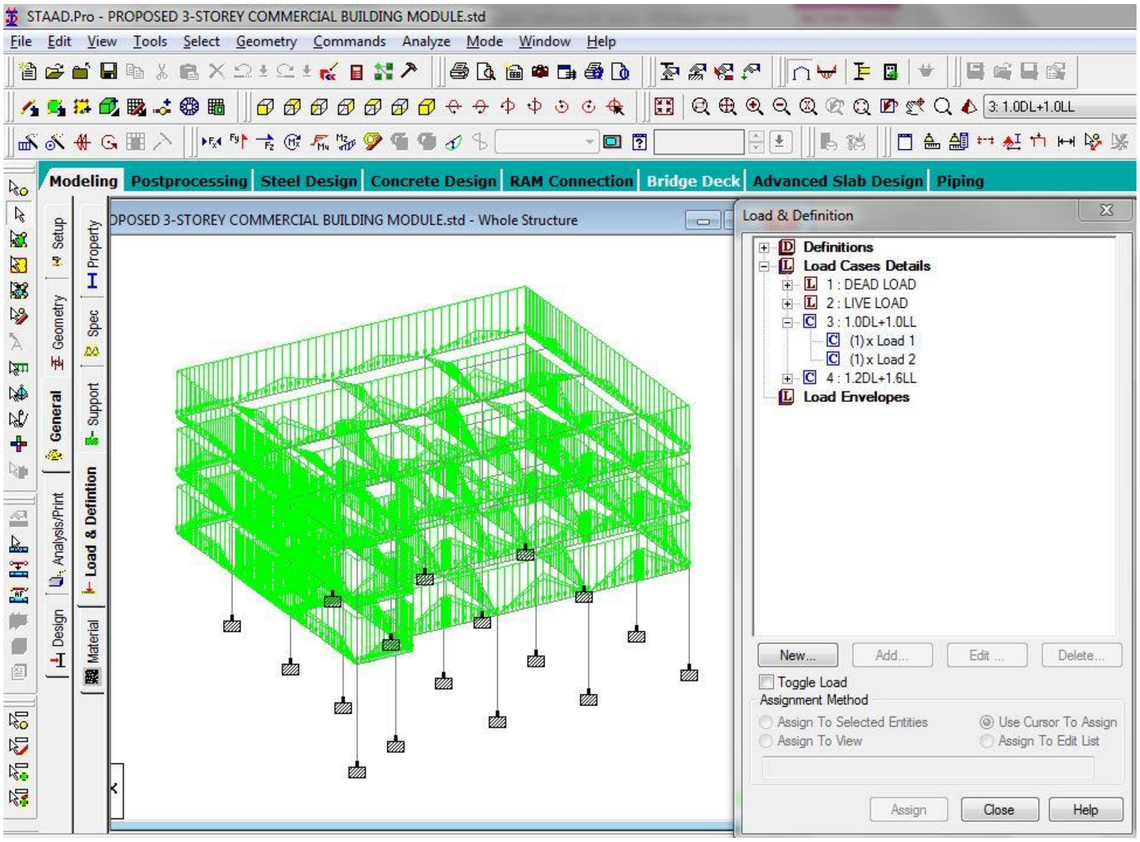
Load Combination

No.	Primary load case name	Symbol
1	DEAD LOAD	DL
2	LIVE LOAD	LL
3	1.0 DEAD LOAD + 1.0 LIVE LOAD	1.0DL+1.0LL
4	1.2 DEAD LAOD + 1.6 LIVE LOAD	1.2DL+1.6LL

Input Editor for Load Combination

```
LOAD COMB 3 1.0DL+1.0LL
3 1.0 4 1.0
LOAD COMB 4 1.2DL+1.6LL
3 1.2 4 1.6
```

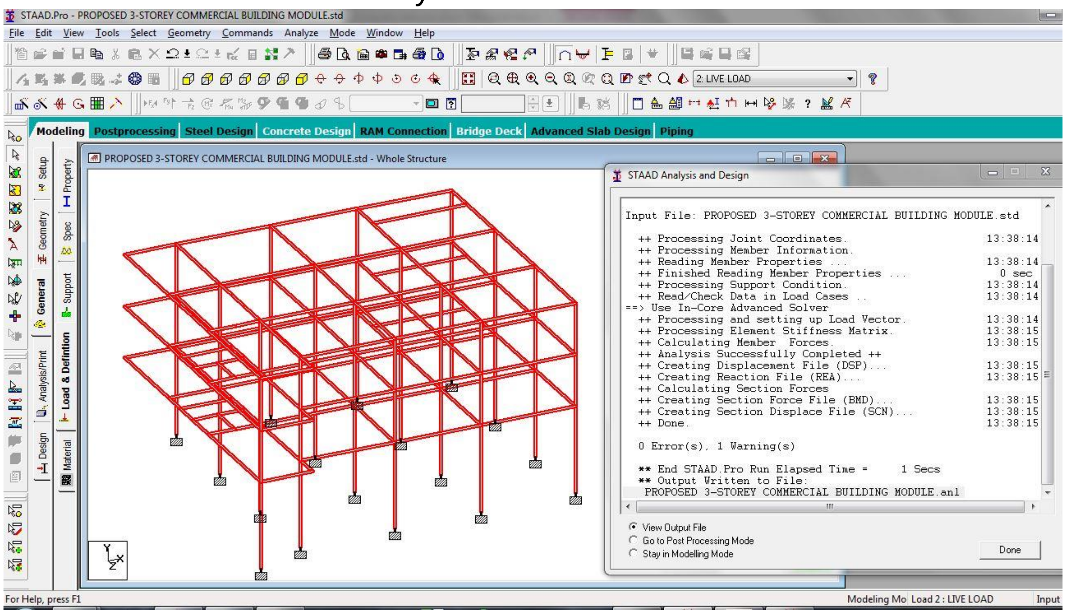
Illustration for Load Combination:


STAAD.Pro - PROPOSED 3-STOREY COMMERCIAL BUILDING MODULE.std
File Edit View Tools Select Geometry Commands Analyze Mode Window Help
3: 1.0DL+1.0LL
Modeling Postprocessing Steel Design Concrete Design RAM Connection Bridge Deck Advanced Slab Design Piping
Setup
Geometry
General
Analysis/Print
Design
Property
Spec
Support
Load & Defintion
Material
OPOSED 3-STOREY COMMERCIAL BUILDING MODULE.std - Whole Structure
Load & Definition
Definitions
Load Cases Details
1 : DEAD LOAD
2 : LIVE LOAD
3 : 1.0DL+1.0LL
(1) x Load 1
(1) x Load 2
4 : 1.2DL+1.6LL
Load Envelopes
New...
Add...
Edit ...
Delete...
Toggle Load
Assignment Method
Assign To Selected Entities
Use Cursor To Assign
Assign To View
Assign To Edit List
Assign
Close
Help

E. Run Structural Analysis

Running structural analysis, we can now see the result of stress, moment, and force of the structure.

Illustration for Structural Analysis:



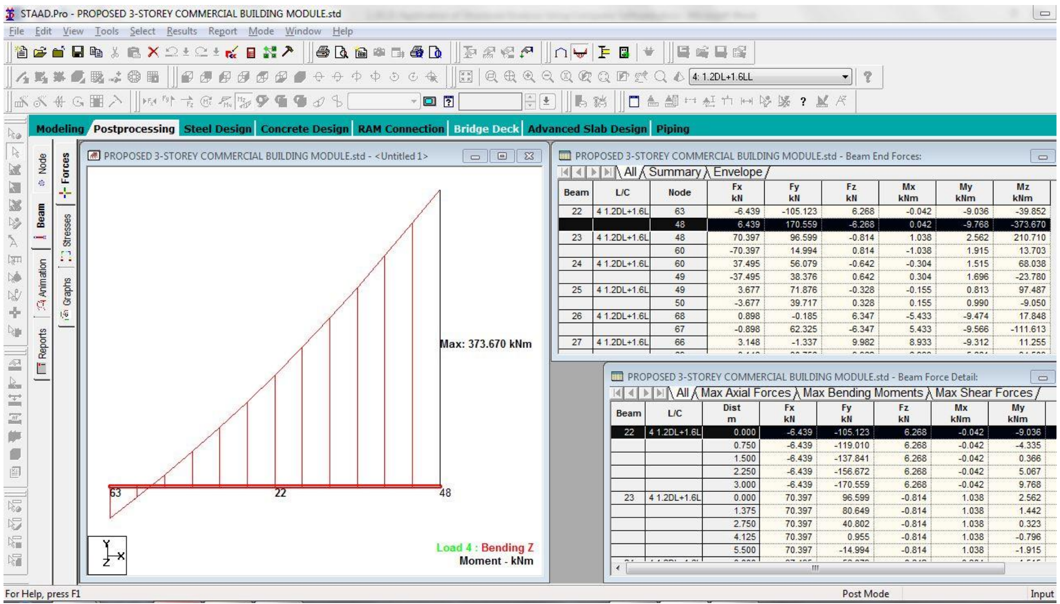
F. Output

This output contains the result of stress, moments, and forces of the structure due to loading.

a. Beam Stress

Beam stress we can see the stress of beams and internal moment due to loading

The second-floor results maximum moment.
Illustration Second Floor maximum moment



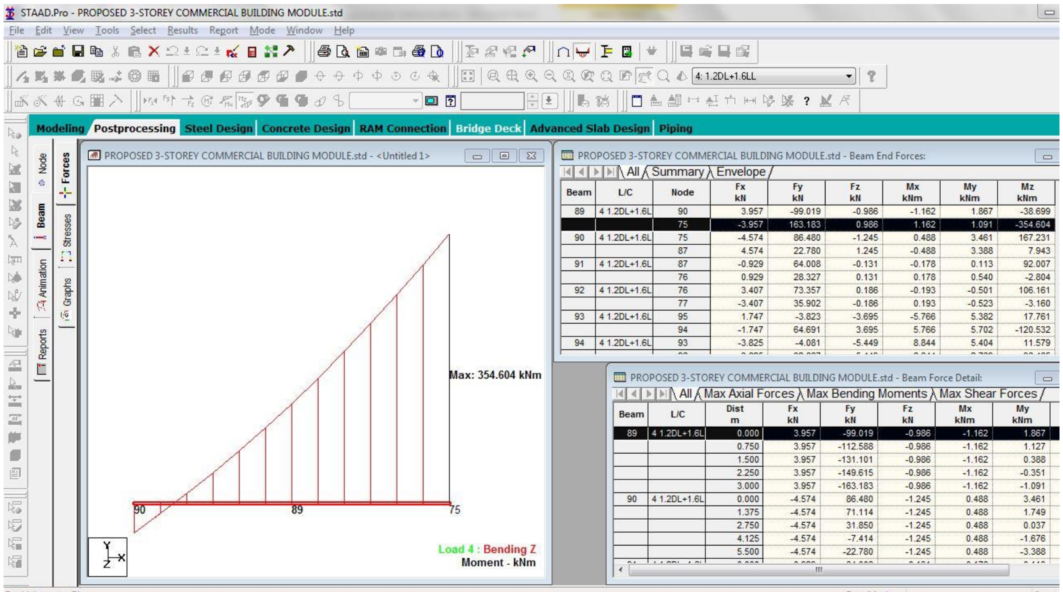
Select> Postprocessing >Beam>find maximum beam **Mz kNm**

Beam	L/C	Node	Mz kN-m
22	4 1.2DL+1.6LL	48	-373.67

The beam 22 node 48 is the selected result of maximum Moment in the structure's second-floor beam.

The third floor results in the maximum moment

Illustration Third Floor maximum moment



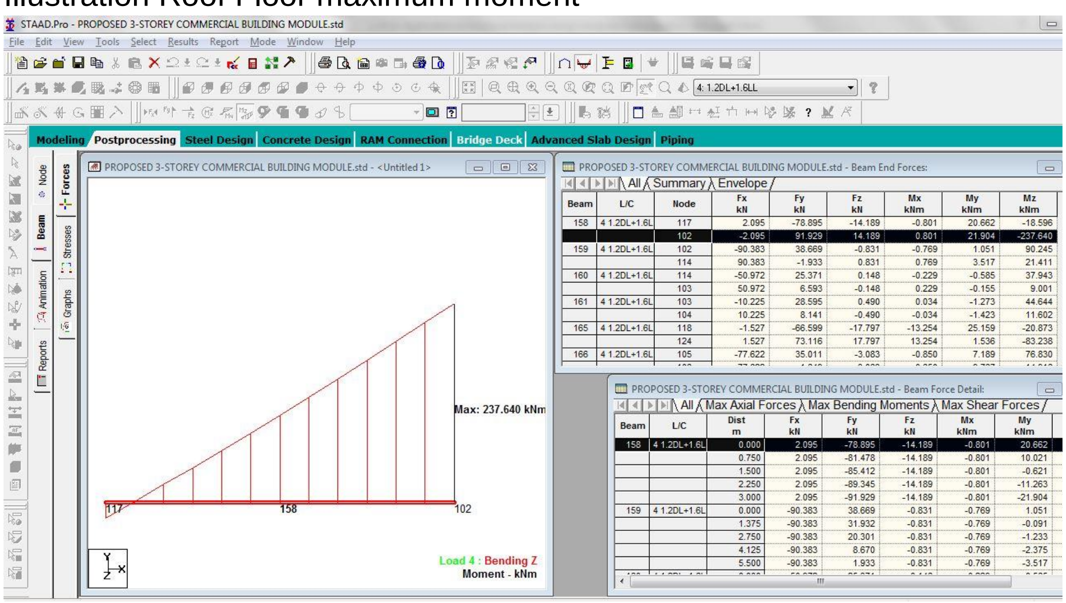
Select> Postprocessing >Beam>find maximum beam **Mz kNm**

Beam	L/C	Node	Mz kN-m
89	4 1.2DL+1.6LL	75	-354.60

The beam 89 node 75 is the selected result of maximum Moment on the structure's third floor.

Roof floor results in the maximum moment

Illustration Roof Floor maximum moment



Select>Postprocessing>Beam>find maximum beam **Mz kNm**

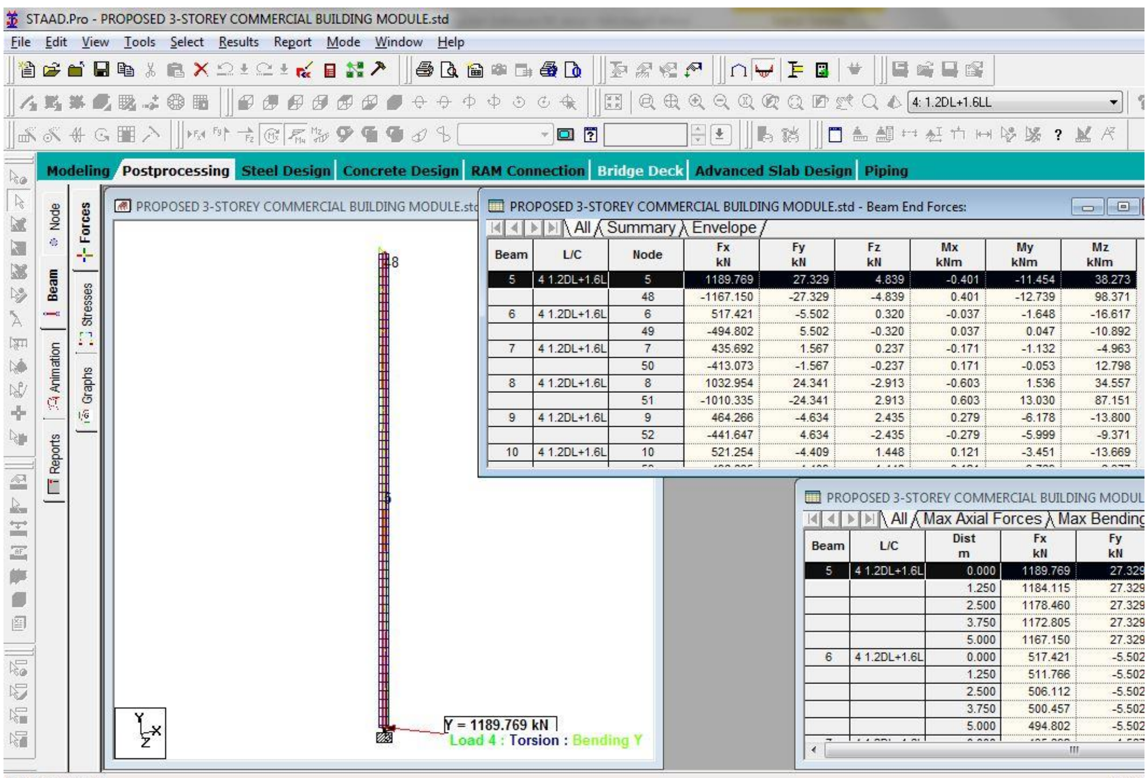
Beam	L/C	Node	Mz kN-m
158	4 1.2DL+1.6LL	102	-237.637

The beam 89 node 75 is the selected result of maximum Moment in the structure's Roof floor.

b. Column Stress

Column stress we can see the column stress due to loading.

Ground Floor column results in maximum force
Illustration for Ground floor column stress:



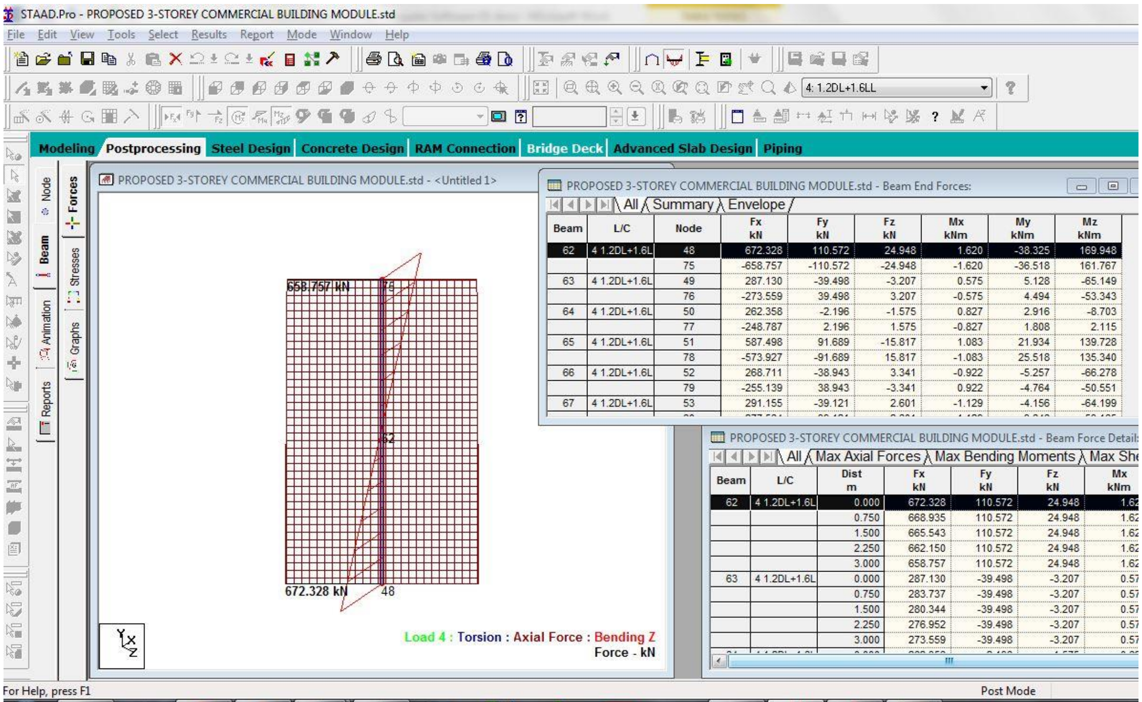
Select>Postprocessing>Beam>find maximum column **Fx kN**

Beam	L/C	Node	Fx kN	My kN-m	Mz kN-m
5	4 1.2DL+1.6LL	5	1189.769	-11.45	38.273

The beam five node 5 is the selected result of the ground column maximum force of the structure.

The second column results in maximum force

Illustration for Second-floor column stress:



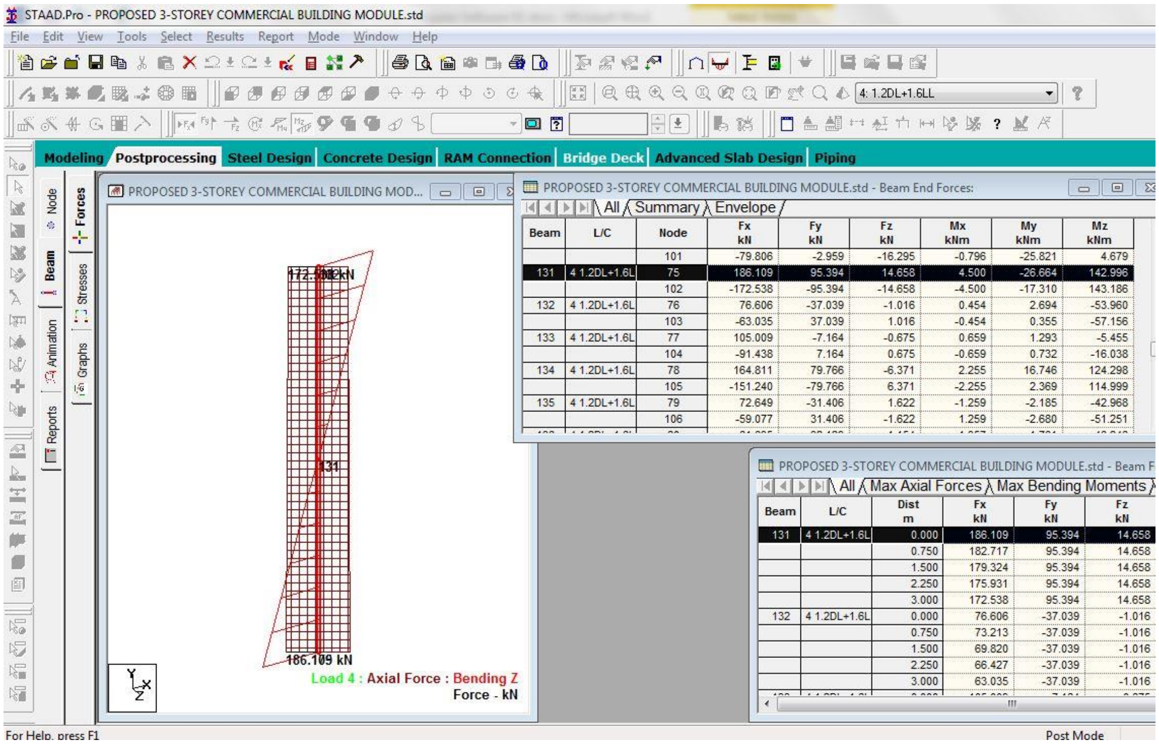
Select>Postprocessing>Beam>find maximum column **Fx kN**

Beam	L/C	Node	Fx kN	My kN-m	Mz kN-m
62	4 1.2DL+1.6LL	48	672.32	-38.32	169.94

The beam 62 node 48 is the selected result of the second column maximum force of the structure.

The third column results in maximum force

Illustration for third-floor column stress:



Select>Postprocessing>Beam>find maximum column **Fx kN**

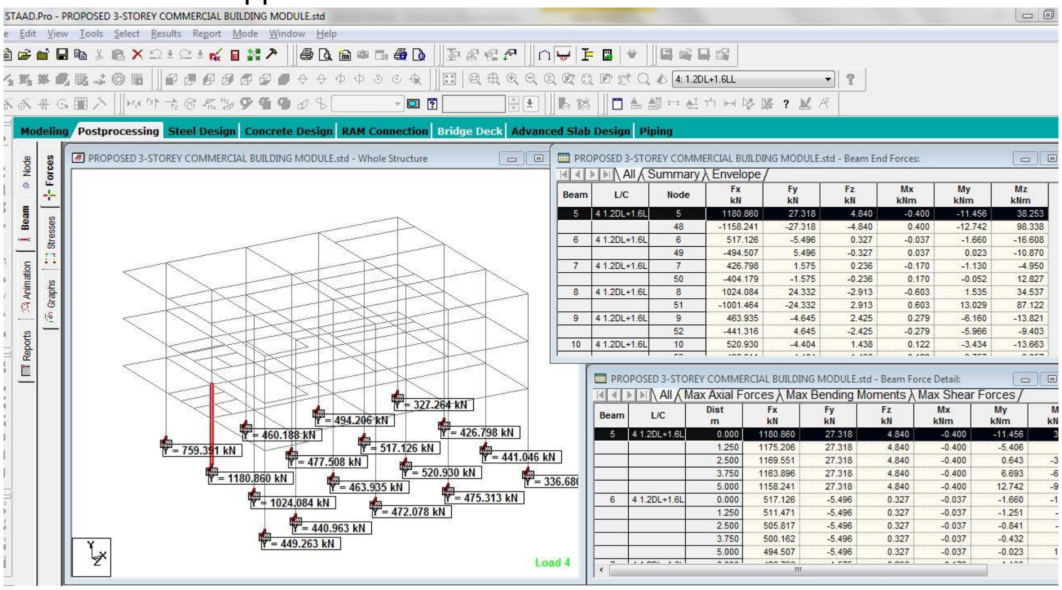
Beam	**L/C**	**Node**	**Fx kN**	**My kN-m**	**Mz kN-m**
131	4 1.2DL+1.6LL	75	186.109	-26.664	142.996

The beam 131 node 75 is the selected result of the third column maximum force of the structure.

c. Support Reaction

This illustration below shows the support reaction on the structure and selecting the maximum Support Reaction to use for design.

Illustration for Support Reaction:



Select>Prostprcessing>Beam>find maximum Reaction **Fx kN**

Beam/ Element	L/C	Node/ Support	Fx kN	Mx kN-m	My kN-m
5	4 1.2DL+1.6LL	5	1180.86	-0.400	-11.45

The beam five node 5 is the selected result of the maximum Support Reaction of the structure.

VII. Conclusions

The Structural Analysis using structural engineering software like the STAAD Pro is very user-friendly software. It allows the structural designer to view the structural geometry of the structure directly. The structure can be evaluated quickly for overlapping of joints and members.

The Loads applied to the structure, such as Dead Loads and Live Loads, can be evaluated correctly if applied to the members.

The structural analysis output can be evaluated for each of the members, such as stresses in beam, stresses in columns, and the support reactions. Structural Analysis for the 3 Storey Commercial Building with Roofdeck has the following summary of stresses:

A. Summary of maximum stresses in beams

Stresses in Beams				
Floor	**Beam**	**L/C**	**Node**	**Mz kN-m**
2nd Floor	22	4 1.2DL+1.6LL	48	-373.67
3rd Floor	89	4 1.2DL+1.6LL	75	-354.60
Roof Deck	158	4 1.2DL+1.6LL	102	-237.637

B. Summary of maximum stresses in columns

Stresses in Columns						
Floor	**Beam**	**L/C**	**Node**	**Fx kN**	**My kN-m**	**Mz kN-m**
Ground Floor	5	4 1.2DL+1.6LL	5	1189.769	-11.45	38.273
2nd Floor	62	4 1.2DL+1.6LL	48	672.32	-38.32	169.94
3rd Floor	131	4 1.2DL+1.6LL	75	186.109	-26.664	142.996

C. Summary of the support reactions

Forces in the Support					
Beam/ Element	**L/C**	**Node/ Support**	**Fx kN**	**Mx kN-m**	**My kN-m**
5	4 1.2DL+1.6LL	5	1180.86	-0.400	-11.45
8	4 1.2DL+1.6LL	8	1024.08	-0.603	1.535
1	4 1.2DL+1.6LL	1	759.391	-0.562	-18.625

In conclusion, structural engineering software in the structural analysis allows the structural designer to quickly determine the maximum stresses for each of the structural elements. The stresses for each of the beams and columns can be printed from the output and be evaluated for optimization of the design. Similarly, each of the support forces can be obtained for the structural design of a foundation.

References

American Institute of Steel Construction. (2016). *Steel Design Guide.* Retrieved from American Institute of Steel Construction Publications: https://www.aisc.org/products/publication/design-guide/design-guide---set-1/

Bentley Systems. (1997). STAAD Pro 3D Structural Analysis and Design software.

Building Design and Management . (n.d.). Retrieved from https://www.ombudsman.gov.ph/UNDP4/wp-content/uploads/2012/12/Chap-09.-Building-Design-Manangement-30Nov06-UPF.pdf

Construction, A. I. (1991). *Low- And Medium-Rise Steel Buildings.* United States of America: American Institute of Steel Construction.

Das et.al. (2011). *Structural Analysis.* PHI Mediamatics Publication.

Das, S. (2016). *STAAD.Pro CONNECT Edtion - Seismic Analysis using IS 1893 (Part-1)-2016.* Bentley Institute Press.

Gupta, R. S. (2014). *Principles of Structural Design: Wood, Steel, and Concrete.* CRC Press.

Handbook on Good Building Design and Construction in the Philippines. (2008). Retrieved from https://www.unisdr.org/files/10329_GoodBuildingHandbookPhilippines.pdf

Hibbeler, R. C. (2018). *Structural Analysis Tenth Edition.* New York City: Pearson.

International Handbook of Earthquake & Engineering. (2002). *Earthquake & Engineering Seismology.* Academic Press.

International Journal of Innovative Research in Science, E. a. (2016, January). *A Review on the Comparative Study of Steel, RCC, and Compostie Building.* Retrieved from http://www.ijirset.com/upload/2016/january/45_A_Review.pdf

Kassimali, A. (2015). *Structural Analysis Fifth Edition.* Stanford: Cengage Learning Inc.

Kumar, V. (2015). *Structural Modeling, Analysis & Design Using Staad Pro Software.* LAP LAMBERT Academic Publishing.

Lindeburg, M. R., & McMullin , K. M. (2014). *Seismic Design of Building Structures.* Proffesional Publications Inc.

McCormac, J. C., & Csernak, S. F. (2018). *Structural Steel Design Sixth Edition.* New York City: Pearson.

McCormac, J., & Brown, R. (2016). *Design of Reinforced Concrete.* United States of America: John Wiley & Sons, Inc. .

National Structural Code of the Philippines . (2015). Quezon City : Association of Structural Code of the Philippines, Inc.

Paulay, T., & Priestley, M. (2013). *Seismic Design of Reinforced Concrete and Masonry Buildings.* India: Wiley India.

Sarma, T. (2017, 16 December). *Design of R.C.C. Buildings using Staad pro v8i with Indian examples: Static and Dynamic Methods.* Retrieved from https://play.google.com/store/books/details/T_S_Sarma_Design_of_R_C_C_Buildings_using_Staad_Pr?id=VShBDwAAQBAJ&hl=en&gl=US

Segui, W. T. (2018). *Steel Design Sixth Edition.* Boston: Cengage Learning.

Ubejd Mujagic, J., Dolan, J., Ekwueme, C., Fanella, D., & LaBoube, R. (2012). *Structural Design of Low-Rise Buildings in Cold-Formed Steel, Reinforced Masonry, and Structural Timber.* New York: McGraw-Hill Education.