

## STAADPro (Steel Building Design)

**COURSE CODE:** CE 0732-4108

**CREDIT: 01**  
**CIE MARKS: 30**

**SEE MARKS: 20**



CLO 01 **Identify** the key parameters/ factors for analysis and design of a Steel Building/ Frame

CLO 02 **Define** the applicable load cases and load combinations for the steel Buildings/Frame as per code.

CLO 03 **Analyze** Steel Building/Frame through computer software

CLO 04 **Design** various components of the Low-Rise Steel Building/Frame

<b>Sl.</b>	<b>Course Contents</b>	<b>Hours</b>	<b>CLOs</b>
1	Introduction to STAAD Pro, Geometrical Modeling of Steel Frame	20	CLO 1,
2	Modeling of loads and load combinations on Steel Frame , Analysis and Interpretation of Results of Analysis of Steel Frame	35	CLO 1
3	Design of Steel Frame using STAAD Pro, Interpretation of Results of Design of Steel Frame.	30	CLO 3

<b>Week</b>	<b>Topic</b>	<b>Teaching Learning Strategy</b>	<b>Assessment Strategy</b>	<b>CLOs</b>
1	Introduction to STAAD Pro	Lecture, Oral presentation	Lab Test, Quiz and Report	CLO 1
2	Geometrical Modeling of Steel Frame	Lecture, Discussion	Lab Test, Quiz and Report	CLO 2
3,4	Modeling of loads and load combinations on Steel Frame	Lecture, Discussion	Lab Test, Quiz and Report	CLO 2
5	Analysis and Interpretation of Results of Analysis of Steel Frame	Lecture, Discussion	Lab Test, Quiz and Report	CLO 3
6-8	Design of Steel Frame using STAAD Pro	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4
9	Interpretation of Results of Design of Steel Frame.	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4

**CIE- Continuous Internal Evaluation (60 Marks) (should be converted in actual marks)**

<b>Bloom's Category Marks (out of 60)</b>	<b>Lab Final (30)</b>	<b>Lab Report (10)</b>	<b>Continuous lab performance (10)</b>	<b>Presentation &amp; Viva (10)</b>	<b>External Participation in Curricular/Co-Curricular Activities (10)</b>
Remember	05			02	Attendance 10
Understand	05	05	02	03	
Apply	05		02		
Analyze	05		02		
Evaluate	05	05	02		
Create				05	

**SEE- Semester End Examination (60 Marks) (should be converted in actual marks)**

<b>Bloom's Category</b>	<b>Tests</b>
Remember	<b>05</b>
Understand	<b>07</b>
Apply	<b>08</b>
Analyze	<b>07</b>
Evaluate	<b>08</b>
Create	<b>05</b>

10-11	<b>Practice, Review/Reserved Day</b>	Lecture, Discussion	Lab Test, Quiz and Report	<b>CLO 4</b>
12-13	Lab Report Assessment, Self study	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4
14-17	Lab Test, Viva, Quiz, Overall Assessment, Skill Development Test (Competency)	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4

**Prepared By**

Somen Saha

Lecturer

Department of Civil Engineering

University of Global Village (UGV), Barishal

Ref– Chhatrapati shivaji institute of technology

# Introduction to Staad Pro

## Week 1

## **EXP NO 1 : INTRODUCTION TO STAAD PRO.**

### **Aim:**

To study the introduction of STAAD PRO.

### **Overview:**

STAAD or (STAAD Pro) is a structural analysis and design computer program originally developed by Research Engineers International in Yorba Linda, CA. In late 2005, Research Engineer International was bought by Bentley Systems. The commercial version STAAD Pro is one of the most widely used structural analysis and design software. It supports several steel, concrete and timber design codes. It can make use of various forms of analysis from the traditional 1st order static analysis, 2nd order p-delta analysis, geometric nonlinear analysis or a buckling analysis. It can also make use of various forms of dynamic analysis from modal extraction to time history and response spectrum analysis.

### **Advantages:**

1. Easy to use interface,
2. Conformation with the Indian Standard Codes,
3. Versatile nature of solving any type of problem,
4. Accuracy of the solution.

STAAD Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD Pro is the professional's choice for steel, concrete, timber, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

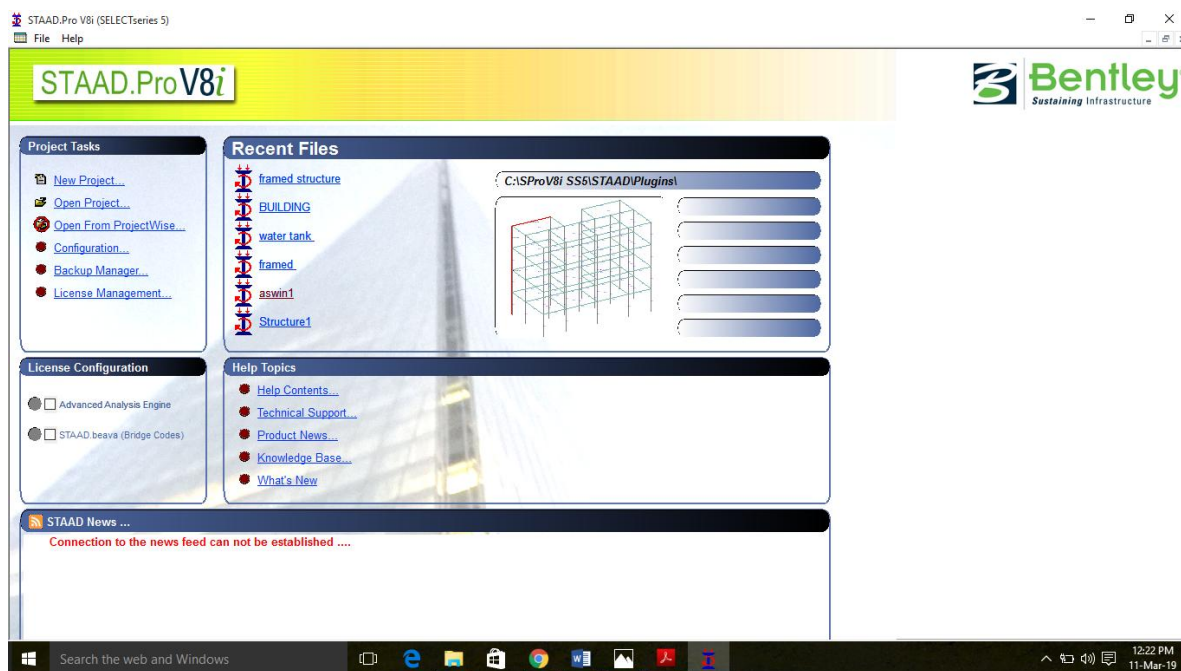
### **Starting STAAD Pro**

There are two possible ways to start STAAD Pro:

1. Go to Start/All Programs/STAAD.Pro.
2. Double-click the shortcut on the Windows Desktop.

## The Opening Screen

Either way you will see the following screen (this screen will pop-up each and every time you close a file.



Under Project Tasks you can:

1. Create New file
2. Open an existing file
3. Configure STAAD Pro for the next input file
4. Configure the Backup Manager.

## Creating new file

To create a new file use one of the following methods:

1. Under **Project Tasks**, click **New Project**
2. From menus select **File/New**, or click the **New Structure** button in the **File** toolbar. The following dialog box will be displayed.



The 'New' dialog box in STAAD Pro is shown. It has a title bar with 'New' and a close button. Inside, there's a list of structure types: 'Space' (selected), 'Plane', 'Floor', and 'Truss'. To the right, 'File Name' is 'Structure1' and 'Location' is 'C:\SProV8i SS5\STAAD\Plugins\' with a three-dot button. Below this is a text box explaining that a SPACE structure is a three-dimensional framed structure. At the bottom, there are two groups of radio buttons: 'Length Units' (Inch, Foot, Millimeter, Centimeter, Decimeter, Meter (selected), Kilometer) and 'Force Units' (Pound, KiloPound, Kilogram, Metric Ton, Newton, DecaNewton, KiloNewton (selected), MegaNewton). At the very bottom are four buttons: '< Back', 'Next >' (highlighted), 'Cancel', and 'Help'.

## File Name

Specify the name of the new file (no need to type .STD, STAAD will do that for you); file names in STAAD Pro can take long filenames.

## Location

Specify where you will save this file in your local hard drives, or any network hard drive, and then specify the folder name (subdirectory) (example C:\SProV8i SS5\ STAAD\Examp), To change these settings, simply click the three dots button, and the following dialog will appear:

## Type of Structure

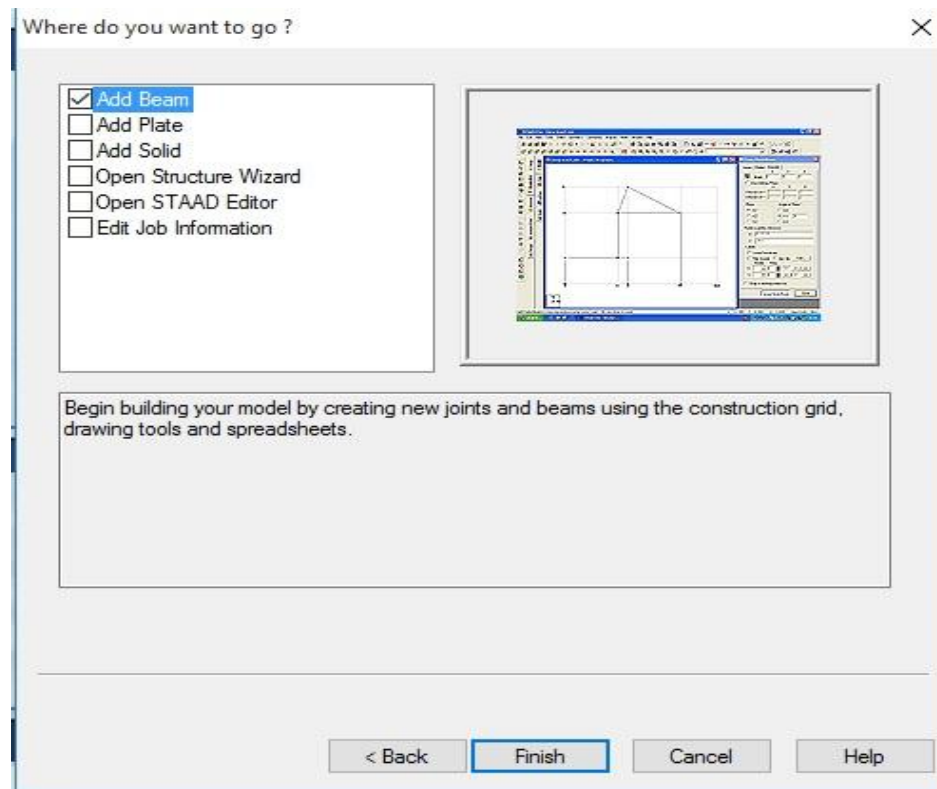
STAAD Pro provides 4 different structure types.

1. **Space:** Three-dimensional framed structure with loads applied in any plane (The most general).
2. **Plane:** Two-dimensional structure framed in the X-Y plane with loads in the same plane.
3. **Floor:** Two, or three-dimensional structure having no horizontal (global X or Z) movement of the structure (FX, FZ & MY, are restrained at every joint).
4. **Truss:** Any structure consists of truss members only, which can have only axial member forces and no bending in the members.

## Length, and Force Units

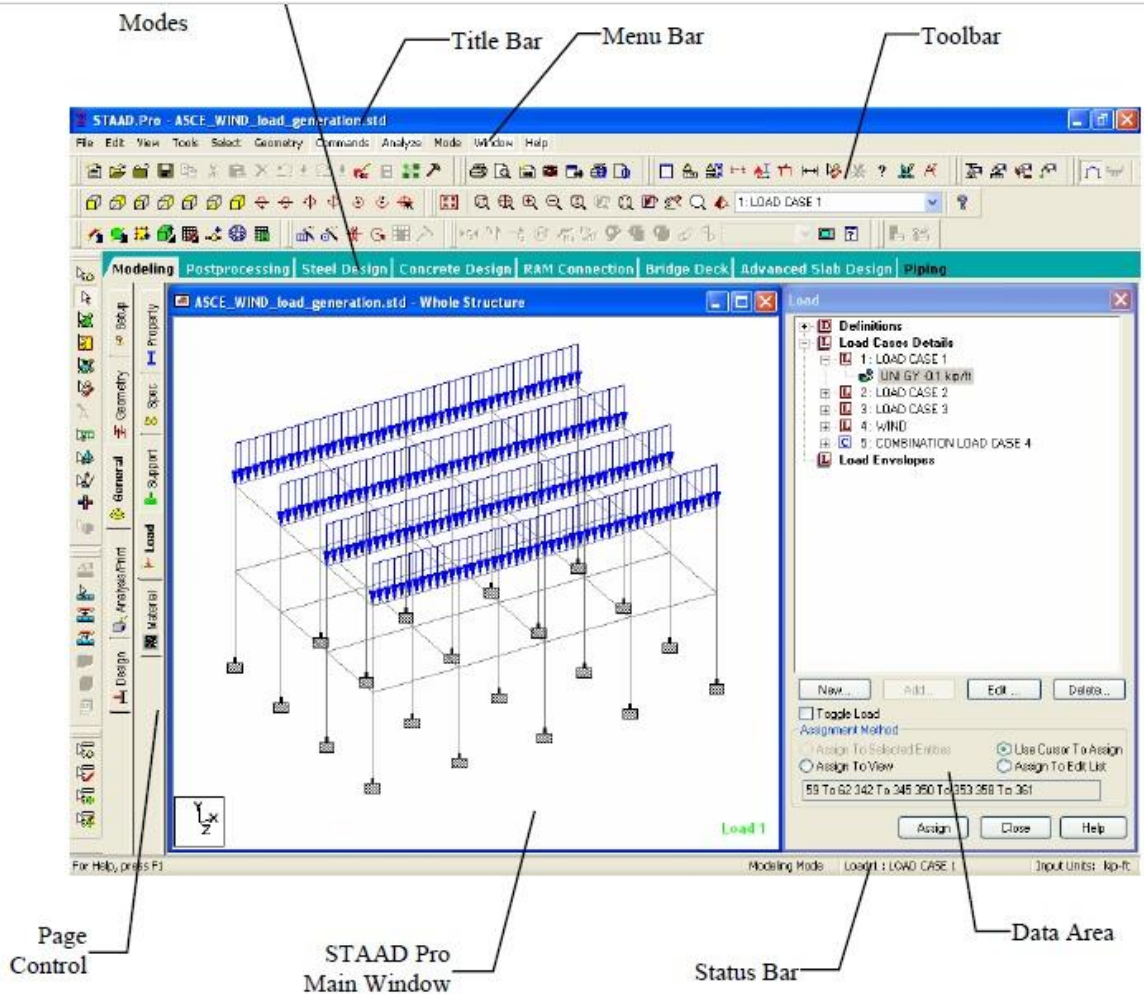
When you install the software at your hard drive, the installation software will ask you to specify what is your default unit system, English (ft, inch, kips) or Metric (m, mm, KN). For this courseware we chose Metric, hence the default Length and Force Units are Meter, and Kilo Newton respectively.

5. This will be to-start-with units, and not the only units you can use while you are creating the input file. As a user you have the ability to change the units at any point to whatever desired units (STAAD internally will make the necessary conversion).
6. When you are done click **Next** in order to proceed. The following dialog box will be displayed:



7. The only purpose of this dialog box is to ask the user what is the first step to be done in creating the input file.
8. To finish creating a file in STAAD Pro, click **Finish**.

## STAAD Pro Screen



### Notes on Page Control & Data Area

9. Page Control is another way (after menus, and toolbars) to execute commands in STAAD Pro.
10. Page Controls are:
  11. The tabs that appear at the left of the main window.
  12. Each Page Control has its own sub-pages.
  13. Each Page Control has its own function,

### Exiting STAAD Pro

To exit STAAD Pro select **File/Exit** and STAAD Pro will close the current file, and exit the software. The only difference between closing a file and exiting STAAD Pro is the closing of the software, and the rest is the same.

### Saving and Saving As

1. To save the current file, you can select **File/Save**, or click the **Save** button in the File toolbar
2. To save the current file under a new name, simply select **File/Save As**, a dialog box will be displayed.
3. First select the desired drive, and folder. Then, type in the file name, leave the file type to be STAAD Space File (\*.std), click **Save**.

**Result:**

Thus the introduction about STAAD Pro is studied successfully.

**Viva Questions**

1. How to start this software for truss designing ?
2. How to start this software for Frame designing ?
3. Tell me the difference between truss and frame ?
4. What is a hybrid truss structure ?
5. Tell me the full procedure of creating a new file .

# Geometric Modeling Of Steel Frame

## Week 2

## **EXPERIMENT NO. 2 : Geometrical Modeling of Steel Frame**

**AIM:** Geometrical Modeling of Steel Frame .

### **SOFTWARE USED:**

STAAD Pro

### **PROCEDURE:**

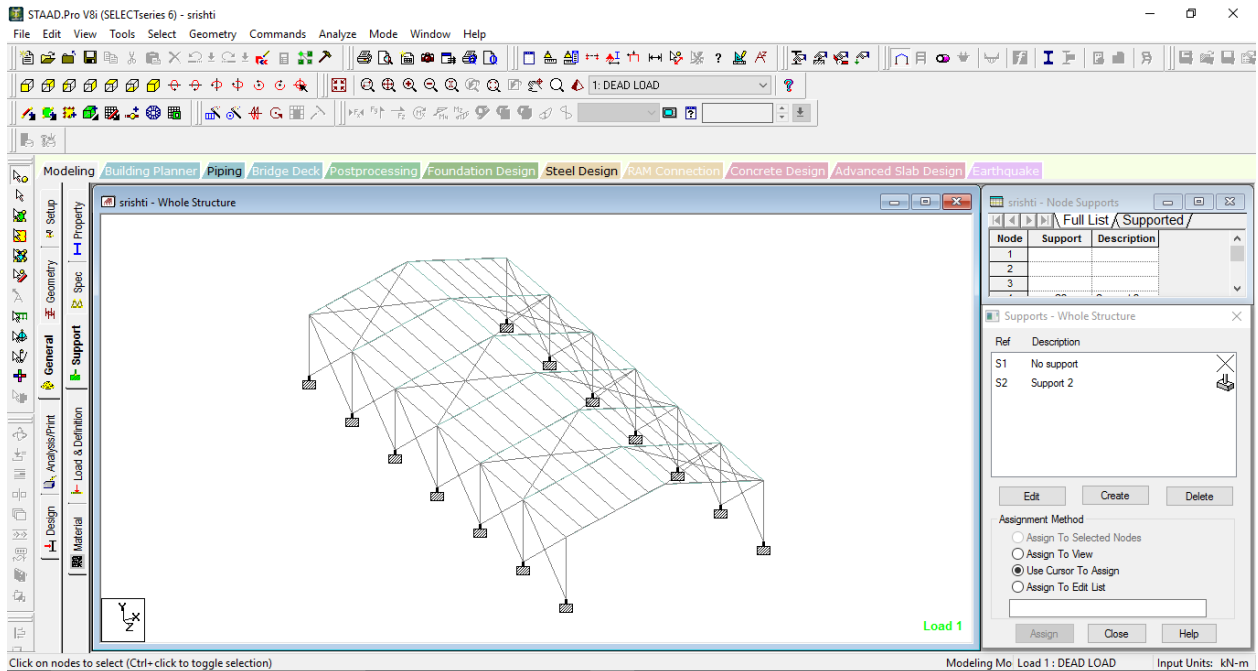
- Under Project Tasks, click New Project.
- Specify the name of the new file.
- Specify where you will save this file or location of the file.
- Select Space as the appropriate structure.
- Click on next to proceed.
- Take the default length in metre and force unit in Kilo Newton.
- Click on add beam and then click finish option.
- Select the front view of the snap node beam.
- click on edit on the snap node grid option and take right X Co-ordinate as 20.
- Now join the coordinate (0,0), (0,5), (8,7), (16,5), (16,0).
- select any of the beam to split the beam and click on the geometry option given on the top.
- click on insert node.
- take the value of 'n' as 5 and then click on 'add n points' and then click on ok. Similarly split the another beam.
- select the structure and click on transitional repeat option given on the top.

Take no. of steps -6

step spacing-6

select the global direction as Z and click on the link steps and open base and then click on ok.

- click on isometric view to view the structure.
- select add beam option and create the bracings.



- Go to the property by clicking on general, and then click on the section database option.
- click on Indian and select I Shape and click on add.
- Select Pipe and then click on add.
- Click on S- Shape and select ISHB 400, and then click on add. Likewise select the angle section.
- Click on SUPPORT and then click on create to select pinned support.

**RESULT :** Geometrical modelling of steel frame is studied successfully.

### VivaQuestion

1. Tell about your experiment in short .
2. Explain the full procedure of steel design in brief .
3. Which code is used for live load designing .
4. Which codebook is used to design wind load .
5. Written any five I.S. codes .
6. What are shear forces
7. Explain bending moment diagrams .
8. Why there is need to draw BMD and SFD .
9. How bending works to affect a building .

# Modeling of loads and load combinations on Steel Frame .

Week 3-4



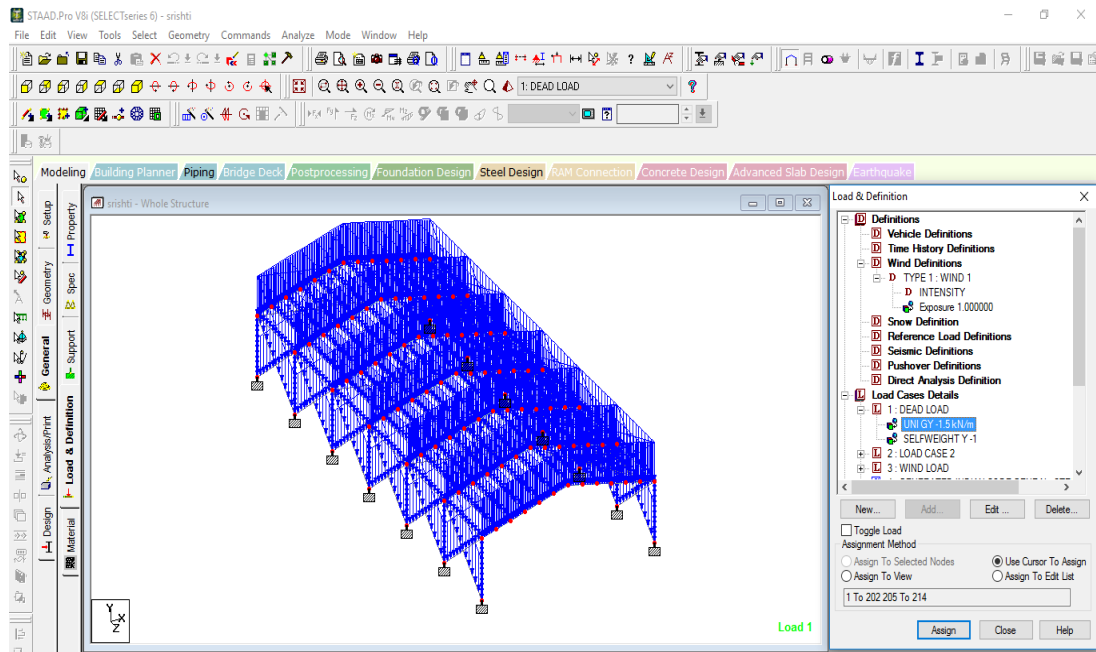
## EXPERIMENT NO.-3 Modeling of loads and load combinations on Steel Frame .

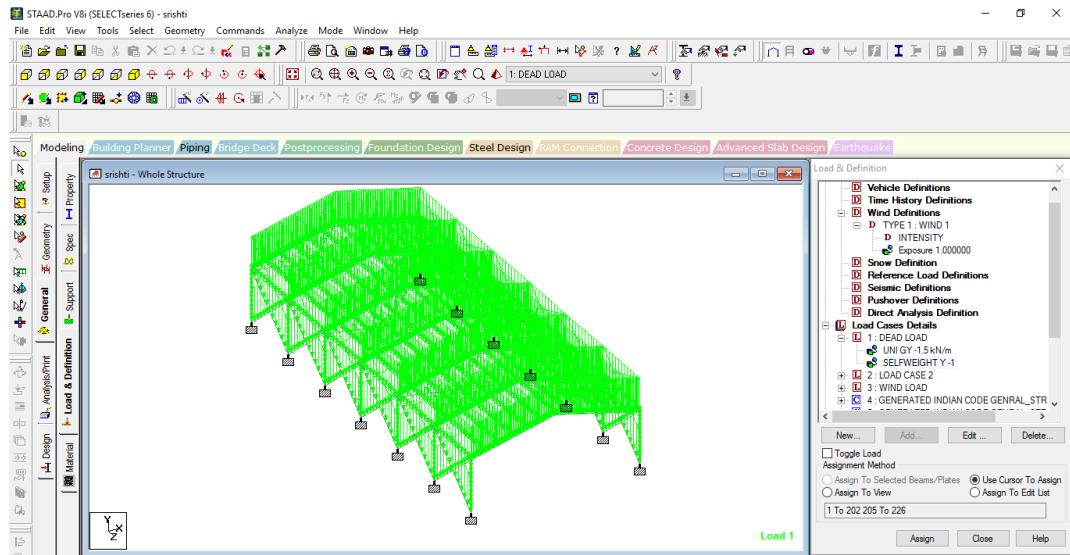
**AIM:** Modeling of loads and load combinations on Steel Frame .

### Procedure:

#### LOADS AND DEFINITIONS-

- Click on definition and then click on the add option.
- Select the wind option and then click on add and take the factor as 1 and then assign it.
- Now click on load case details and then add.
- Then add the dead load, selfweight taking the factor as -1, and the member load taking the factor as -1 and the click on add and then assign it.



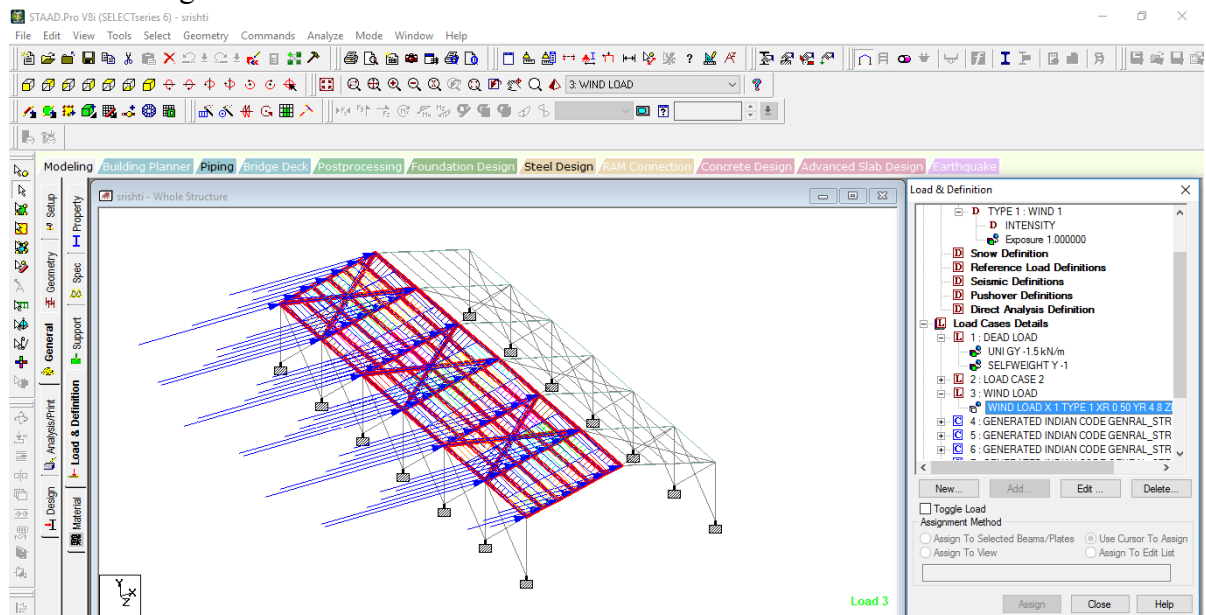


- Likewise click on the live load and then click on add then again click on the plate load option where the value of  $w = -0.05$  in GY –direction and then add and assign it.
- Likewise click on wind load and then add

Minimum- 4

Maximum- 8

And then assign it.



- Now go to the 3D Rendering view to view the structure.

**RESULT :** Modelling of loads and loads combination of steel frame is studied successfully.

# ANALYSIS AND INTERPRETATION OF RESULTS OF STEEL FRAME.

Week 5

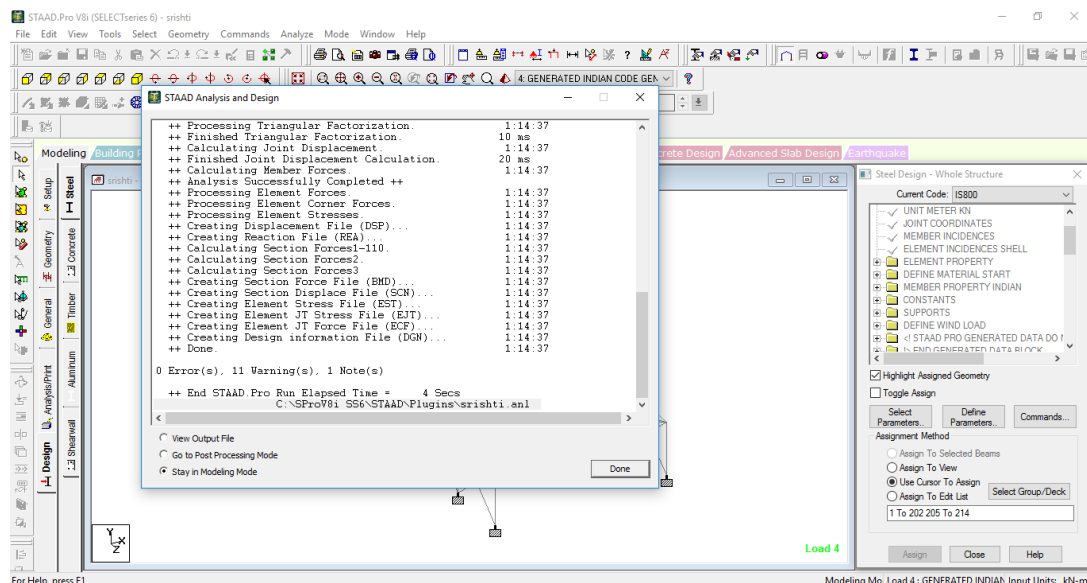
## EXPERIMENT NO. 4 : ANALYSIS AND INTERPRETATION OF RESULTS OF STEEL FRAME.

**AIM:** To analyse and interpret the results of steel frame using STAAD PRO.

### Procedure:

1. After assigning properties to the structure . Goto **loads and loads combination**  
Goto definition and define the loads.  
**Select wind definition → add → close → select wind definition → add → AMERICA code → add → click on calculate as per → OK → close . select exposure select the suitable range → add → select → assign to view → assign.**
2. Go to load case details → add (**dead load, live load and wind load** ) and then add each of them
3. Select dead load → add self weight = 1 (change to steel ) → add → member load = -1.5  
Select live load → add plate load {P = -0.05 KN/m<sup>2</sup>}  
Select GY option → add  
Assign to view → assign
4. Goto **auto load combination** → select all the load combinations. After that see the combination of loads in the **STAADPRO main window**.
5. Goto **commands** → **analyse perform analysis**.
6. **Analyse** → **run analysis**.
7. After the modelling and loads combination of the steel structure we have to analyse the result of the steel structure.
8. For that Goto **commands** → **analyse perform analysis**.

**Analyse** → **run analysis**.



**RESULT :**

Thus the Analysis and interpretation of results of design of steel frame is studied successfully.

**VivaQuestion**

1. Tell about your experiment in short .
2. Explain the full procedure of steel design in brief .
3. Which code is used for live load designing .
4. Which codebook is used to design wind load .
5. Written any five I.S. codes .
6. What are shear forces
7. Explain bending moment diagrams .
8. Why there is need to draw BMD and SFD .
9. How bending works to affect a building .

# Design of Steel Frame using STAAD Pro.

Week 6-8

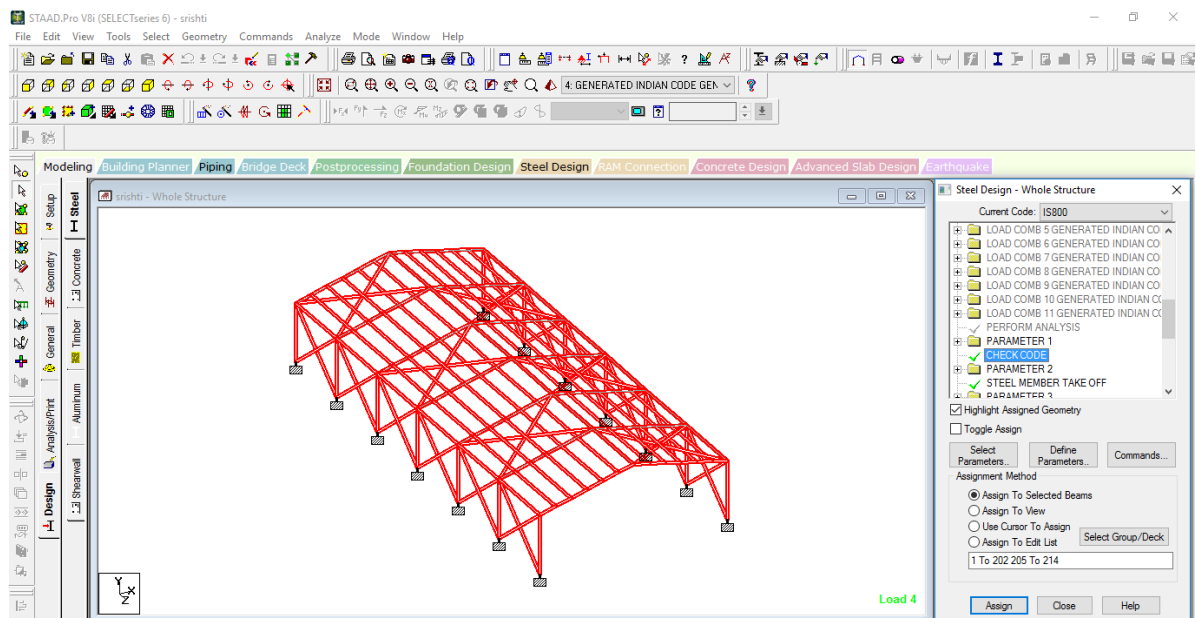
## EXPERIMENT NO. 5 : Design of Steel Frame using STAAD Pro.

AIM: Design of Steel Frame using STAAD Pro.

### Procedure:

#### DESIGN-

- Select the code as IS800.
- Then click on select parameters from which click on yield strength and then click on ok.
- Now click n define parameters and then add and the click on ok.
- Now click on commands, add the member take off and take off option and then click on ok and then assign it by clicking on assign to view option.



**RESULT:** Design of the steel frame is studied successfully.

### VivaQuestion

1. Tell about your experiment in short .
2. Explain the full procedure of steel design in brief .
3. Which code is used for live load designing .
4. Which codebook is used to design wind load .
5. Written any five I.S. codes .

# ANALYSIS AND INTERPRETATION OF THE RESULTS OF DESIGN OF STEEL FRAME.

Week 9

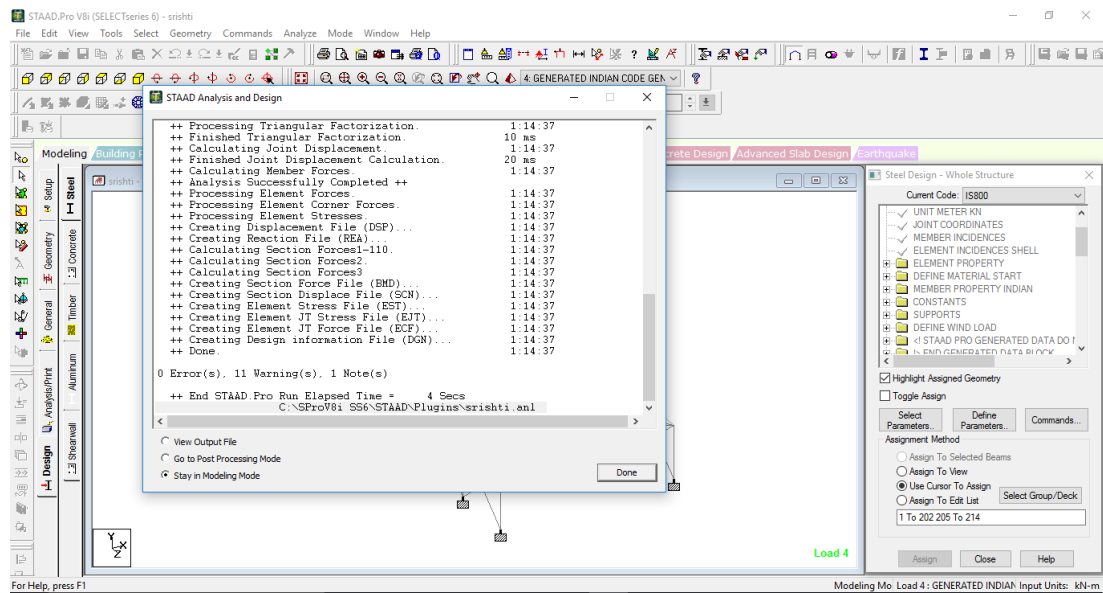


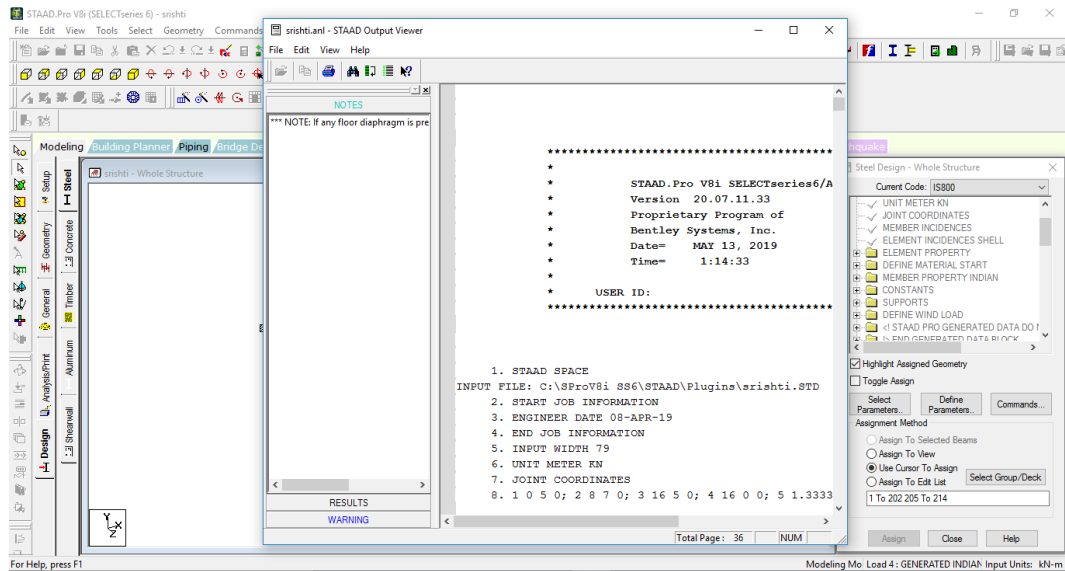
## EXPERIMENT NO. 6: ANALYSIS AND INTERPRETATION OF THE RESULTS OF DESIGN OF STEEL FRAME.

**Aim :** Analysis and Interpretation of Results of Analysis of Steel Frame .

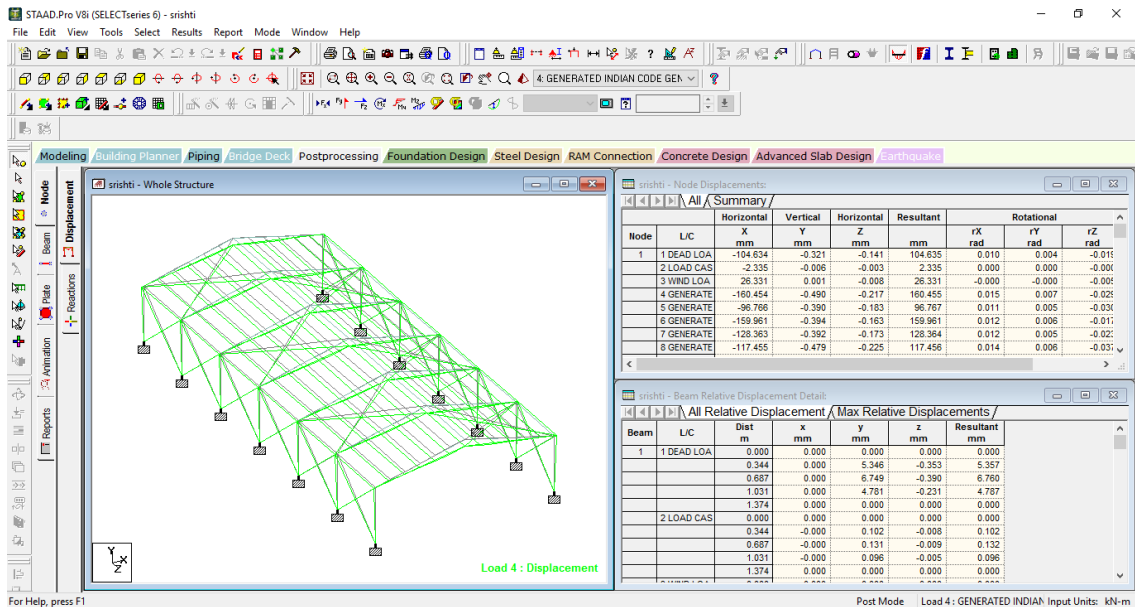
### Procedure:

- Go the commands and click analyse to perform the analysis.
- Now click on Analyse and then click on Run Analysis to view the output result.





- Select the post processing option and then select the result.



## RESULT :

Thus the Analysis and interpretation of results of design of steel frame is studied successfully.

### **VivaQuestion**

- 1. Tell about your experiment in short .**
- 2. Explain the full procedure of steel design in brief .**
- 3. Which code is used for live load designing .**
- 4. Which codebook is used to design wind load .**
- 5. Written any five I.S. codes .**
- 6. What are shear forces**
- 7. Explain bending moment diagrams .**
- 8. Why there is need to draw BMD and SFD .**
- 9. How bending works to affect a building .**

Practice and Reserved Day  
Week 10-11

Lab Report Assessment & Self Study  
Week 12-13

Lab Test, Viva, Quiz, Overall Assessment,  
Skill Development Test (Competency)  
Week 14-17



