STAAD

Basics

CHAPTER – I

Structural Analysis – Basic Concepts (A Revision)

This Chapter covers:

- Basic definitions
- What is Structural Analysis?
- Methods of Analysis
- Need for Computer method of Analysis.

Structures are defined as a frame work that carries external loads. A structure, in general, is composed of interconnected members, connected by rigid joins or frictional hinges.

DEFINITIONS

Beam

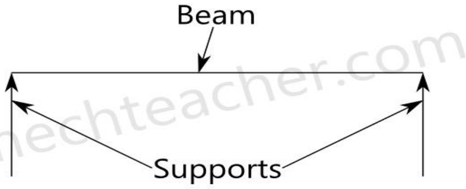
Generally, a beam is a structural horizontal member used to carry vertical loads . Beams are provided to support floor slabs , secondary beams , walls, stairs etc.

Types of Beam

According to the supports provided they are classified as :

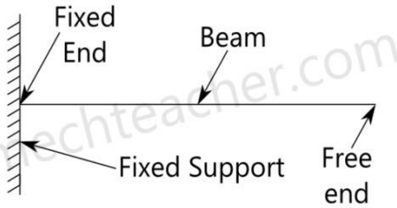
1. Simply supported beam

A beam having its two ends freely resting on masonry walls or pillars is called as simply supported Beam.



2. Cantilever beam

A beam with one end rigidly fixed and the other end free (i.e . not supported) is called cantilever beam .

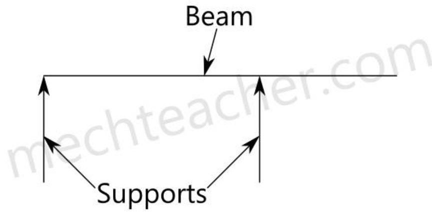


3. Propped cantilever beam

A beam with one end rigidly fixed and the other end resting on masonry walls or pillars is called propped cantilever beam.

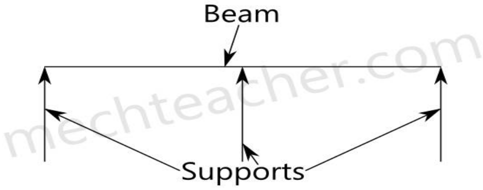
4. Over hanging beam

A beam which is simply supported at two points and having projections at once or both ends beyond the supports is called an overhanging beam.



5. Continuous beam

A beam which has more than two supports is called continuous beam.

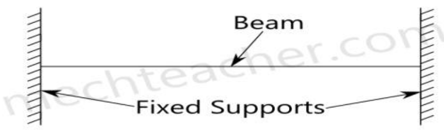


6. Fixed beam

A beam with both of its ends rigidly fixed or built-in into the supporting walls or columns is called fixed beam.

7. Frames

They are made up of beams and columns.



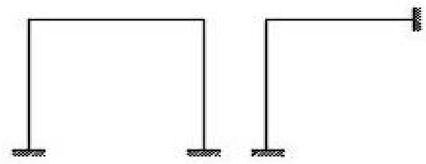
Equilibrium Equations**:**

Statics offers three conditions of equilibrium for co-planner force system :

$\sum H = 0$

$\sum V = 0$

$\sum M = 0$



Classification of structures:

Structures can be classified into following two types:

Statically Determinate Structures

If the number of reactions is equal to the number of available static conditions, then the reactions and the internal forces can be determined using the principle of statics alone . So these types of structures are called statically Determinate Structures .Ex. Simply supported beam, Cantilever beam, Propped cantilever.

Statically Indeterminate Structures

If the number of reaction components is more than the number of available static equations, the reactions and the internal forces cannot be determined using the principle of statics alone. We require additional conditions to determine the reactions in the structure. These types of structures are called Statically Indeterminate structures. Ex. Continuous beams fixed beams etc.

Types of Supports:

Depending on their functions the supports of beams are classified as follows:-

1. Simply support:

It is one which has only a vertical reaction. It allows horizontal movement or rotation. Ex. Masonry Walls.

2. Roller support:

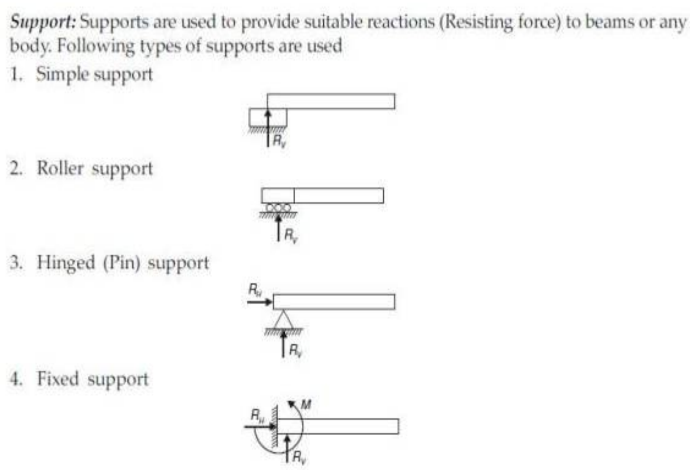
It is similar to the simply support. It offers only a vertical reaction. Rollers are provided to facilitate free horizontal movement. Ex. Roller bearings for trusses.

3. Hinged support:

It is also called pinned support . It offers both vertical and horizontal reaction, but it allows rotation.

4. Fixed support:

It offers vertical, horizontal reaction & moment. Members are restrained against any movement. Members are restrained against any movement or rotation. Ex. Built in Beams, welded



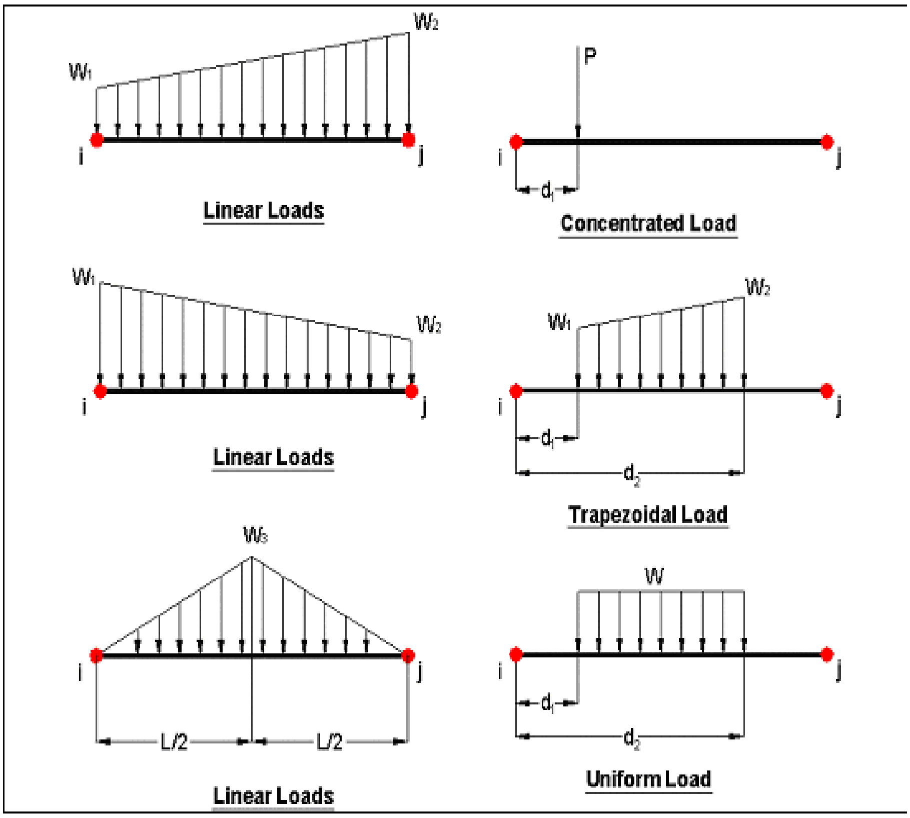
joints.

Effective Span:

Centre –to – distance between the supports is called effective span.

Note: - In STAAD PRO, Nodes are constructed using this effective span only .

TYPES OF LOADS ON BEAMS



Pin jointed frame

A pin jointed frame is a structural framework consisting of members like angles, channel, pipes ext. Jointed together at their ends by rivets or bolts, to from the required shape and to carry the external load.

A truss is a pin jointed frame when it is used to support the roof. Pin-jointed frames are also called as girders when it is used as a beam in bridges etc. It may be called as tower when it is used as mast.

Loadings in pin joined frames:

The loads are generally applied at the joints (called nodes) of the frame and hence its constituent members are always subjected to axial forces and never to bending moment or shear forces.

The axial forces may be either compression or tension forces. Sometimes, the same member may be subjected to different nature of forces under different loading conditions.

Pin – jointed frames broadly classified into two types:

1. Perfect frames and

2. Imperfect frames.

A frame which has sufficient number of number members, to keep it in equilibrium without any change in its shape under the action of external loads, is called a perfect frame.

In short, when a structure satisfies the following equation it is called perfect frames, otherwise it is Imperfect frames.

N =2j – 3

Here,

n = number of members in the frame.

J = number of joints or nodes in the frame.

Also, frames are classified as follows depending upon the determinacy. A frame which could be analyzed completely using three static equations ($\sum H = 0$; $\sum V = 0$; $\sum M = 0$) is called statically **Determinate structure** .

A frame which could not be analyzed using the three static equations ($\sum H = 0$; $\sum V = 0$; $\sum M = 0$) alone is called **statically Indeterminate structure.**

Analysis of Frames:

The perfect frames are analyzed by the following methods, conventionally:

1. **Method of joints.**
2. **Method of sections**
3. **Graphical method.**

Others types of frames like imperfect frames are analyzed using advanced methods.

What is Analysis or Structural Analysis?

Structural Analysis is the determination of the internal forces in the members of the structure due to the external loads acting on it . The external loads may act at the joints or on the members.

The internal forces that we are finding out are called axial forces, shear forces, bending moment or torque.

The structures also are stable apart from its strength to withstand the action of external loads. For that, it should be supported properly and maintain a stable equilibrium. The supports may be of roller, hinged or fixed support.

These supports exert external reactions. If these reactions are determined, then the internal forces can be found out and this process is called as analysis of structures or structural Analysis.

Methods of Analysis

Statically determinate structures can be easily resolved, by calculating the summation of forces in both the directions.

But, statically indeterminate structures cannot be solved simply by having statics equations alone. It needs extra reactions or formulas to solve the structure. These structures can be analyzed through any of the following methods:

1. **Moment area method** – for propped cantilevers and fixed beams.
2. **Three moment equation – for continuous beams.**
3. **Moment distribution method - for continuous beams and frames**
4. **Slope deflection method - for continuous beams and frames**
5. **Method of column analogy – for frames**
6. **Method of consistent deformation – for trusses.**
7. **Substitute frame method, cantilever method – 3D frames.**

Need for Computer Method of Analysis

One can feel the toughness and consider the time factor, for solving indeterminate problems using these methods. He has to keep the entire formulas, calculations, code requirements etc. in figure tip, to analyze and design a structure.

Not only that, you have to see accuracy in calculations and have to keep in mind all the methods & procedure. Also you cannot see the results in depth in manual calculations.

To overcome these difficulties, computer method of solving the problems emerged. Following are advantages of computer aided analysis & design.

- **Analysis and design can be done within a fraction of second**
- **Once constructed the structure, it can be verified for many conditions.**
- **Any type of structure can be resolved.**
- **Very less manual calculations and formula workout.**
- **Accurate and in depth results.**
- **Graphical and printout results can be taken, etc**.

CHAPTER – II

INTRODUCTION TO STAAD PRO

This Chapter Covers:-

- History of Software.
- METHO OF ANALYSIS IN STAAD
- STAAD PROGRAM FLOW CHART
- STARING STAAD PRO
- INTRODUCING STAAD PRO SCREEN
- OVERALL PROCEDURE IN WORKING WITH STAAD
- SETTING DEFAULT CONFIGURATION
- STAAD BASIC DEFINATIONS

About STAAD

STAAD Pro V8i is the most popular structural engineering software. The ultimate power tool for Computerized Structural Engineering. Staad Pro is the use of computer technology to aid in the Structural design of Civil Structures. Staad originally meant Structural Analysis & Design.

Staad pro is the product for 3D model generation, analysis and multi-material design. It has an intuitive, user-friendly GUI, visualization tools, powerful analysis and design facilities and seamless integration to several other modelling and design software products.

The software is fully compatible with all Windows operating systems but is optimized for Windows XP. See the new STAAD Pro V8i from the comfort of your own home or office in a FREE interactive online demonstration or watch some online tutorials at your own leisure.

STAAD Pro has been the choice of design professionals around the world for their specific analysis needs

Who Uses

Professions

- **Structural engineers**
- **Consulting engineers/engineering consultants**

Firms

- **Structural engineering**
- **Structural consultant**
- **Multi-discipline E/A and A/E**

- **Departments in construction companies, owner/operators, and government agencies**

Staad Pro was born giant. It was mixture of the expertise of two long experienced companies.

STAAD PRO introduced a really good-looking interface which actually utilized all the exceptional features of Windows XP/7/8/Linux (Each STAAD PRO was working respectively under the windows available at the time of releasing the software to the markets).

Staad Pro with new features surpassed its predecessors & compotators with its data sharing capacities with other major software like AutoCAD & Ms Excel.

The results generation was yet a new feature that you can depend on STAAD Pro to do for you, now, STAAD Pro can generate handsome reports of the inputs and the outputs with the usage of graphical results embedded within, which can be considered as final document presented to the client.

The Concrete & Steel design were among the things that undergone a face lift , specifically the concrete design as Bentley created a new module specially to tackle this issue. This new module Is easy , and straight forward procedure making the concrete design and results generation a matter of seconds ahead of the user.

History

In the Olden Periods, All the Structural calculations to be done by manually. In Early 80's Some of the Private Structural organizations develop some structural programs. Engineers from that organization frame organization **Research Engineers International** Yorba Linda, CA.

Later **Research Engineers International** develops **STAAD Pro** (2000). Initially this product was launches as rental product (Staad –III). These products are used for educational purposes only. From Staad –IV, Staad is launched as Commercial Products. 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 non linear 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.

In recent years it has become part of integrated structural analysis and design solutions mainly using an exposed API called **OpenSTAAD** to access and drive the program using an VB macro system included in the application or other by including OpenSTAAD functionality in applications that themselves include suitable programmable macro systems. Additionally STAAD Pro has added direct links to applications such as **RAM Connection** and **STAAD.Foundation** to provide engineers working with those applications which handle design post processing not handled by STAAD Pro itself. Another form of integration supported by STAAD Pro is the analysis schema of the CIMsteel Integration Standard, version 2 commonly known as CIS/2 and used by a number modeling and analysis applications.

Clients

A. **Entergy, Bechtel Power**
B. **ProtoPower**
C. **Altran Corporation**
D. **BE & K in Alabama**
E. **Kellogg Brown and Root**

Method of Analysis in STAAD

One of the most famous analysis methods to analyze continuous beam is "Moment Distribution Method", which is based on the concept of transferring the loads on the beams to the supports at their ends.

Each support will take portion of the load according to its K; k is the stiffness factor, which equals EI/L. As you can see E, & L is constant as per span, the only variable here is I; moment of inertia. I depend on the cross section of the member. So if you on the cross section of the member. So if you want to use this analysis method, you have to assume a cross section for the spans of continuous beam.

If you want to use this method to analyze a simple frame, it will work but if you go for simple 3d frame this method proves to be very complicated. Hence a new more sophisticated method emerged, which depends fully on matrices this method is called "**Stiffness Matrix Method**", the main formula of this method is , (P)= (K) x (_)

The 3 matrices are as follows,

(P) - is the force matrix, which includes the forces acting on the whole structure, and the reactions at the supports. This Matrix is partially known as “Acting force” on the structures is already known from the different codes, like Dead Load, Live Load, Wind load, etc.., but reactions are unknown.

(K) – is the stiffness factor matrix K-EI/L, and all of these data either assured known or assumed. So this matrix fully known

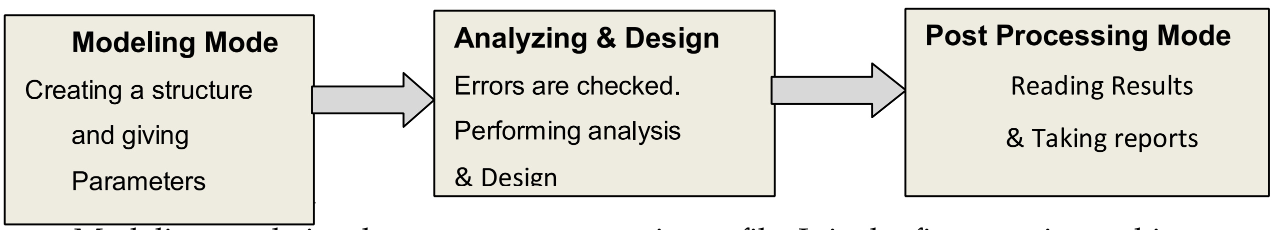
(_) – is the displacement matrix. The displacement of supports are either zeros (fixed supports) or partially zeros (other supports), but the displacements of other nodes are unknown. So this matrix is partially known.

With this three matrices presented as discussed above , the method will solve the system with ordinary matrix methods to get the unknowns. If we solved for the unknown, the reactions will be known, hence shear & moment diagrams can be generated , and the displacement and deflection shapes can be generated .

This method is very hard to be calculated by hand as it needs more time than other methods, so it was not concentrated up until the emergence of computers. Computer program will do tedious & lengthy procedures to solve for this system of matrices, therefore structural software adopted it as the method of analysis. Staad was one of the first to do that.

Overall procedure in working WITH STAAD Pro

- ❖ Preparing the input file
- ❖ Analyze/design the Program
- ❖ Reading the results and verify them



Modeling mode is where we can prepare input file. It is the first step in working in Staad pro. Herewith we will describe our structures. i.e.., the geometry, the cross sections, the material and geometric constants , the support conditions & finally the loading system.

Analyzing & design in the program

After preparing the input file, next is to run the program. STAAD Pro analysis and design engine will start reading input file from left to right & from top to bottom.

The engine will mainly check for two things:

❖ Making sure that the user used the syntax of STAAD Pro commands, or else the engine will produce an error message.

❖ Making sure that all the data needed to form a stable structure exists in the input file, or else the engine will produce an error message.

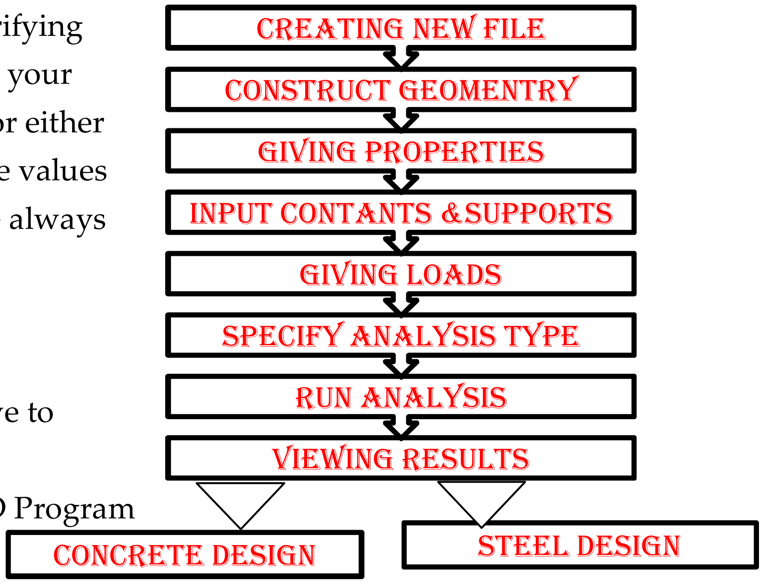
If this two things are correct, STAAD will take values mentioned in the input file (without verification) & produce the output files.

Reading the results& verify them

VIEWING After reading and verifying your results you may decide to go back to your Modeling mode to alter your input file , for either to correct the input file , or to change some values to examine different results. The input file always has extension is "**.std**"

STAAD Program flow chart

Before proceeding further, first we have to



Understand the flow chart of how STAAD Program Flows (or this is the Order to build up an input file).

The Procedure will enable us for

- Put each step in its right position, not before, and not after.
- Make sure that all of the STAAD Pro
  Commands are present in the input file and avoid error messages.

This chapter deals with two part, How to create New File, & Construction of Geometry** (or) Structure of a Problem

(-Geometry is the Skeleton of your Structure, i.e.. it is the arrangement of members(beams & columns) and the plates(slabs, walls & foundations)**

(or)

(A structure is an assembly of individual components such as beams , columns , slabs , plates etc.)

Basic STAAD Definitions

Nodes (or) Joints

Node means a stiffed joint with 6 reactions in STAAD pro. It is located at each end of beam, and each corner of plate. First step in construction of Geometry is building up of Nodes. Each node has Node Number and XYZ Co-ordinate in space.

Beam (or) Member

Beam in STAAD Pro means any member in the structure. It can be beam column ,bracing member Or Truss Member .Beams are actually defined based on Nodes at their Ends. Each Beam has a Beam Number & The Node number at its ends.

For Example, if we want to draw a member of 10 M length following is the Input,

Joint Coordinates

1 0 0 0; 2 10 0 0;

Member Incidence

1 1 2

In joint co-ordinates

Command 1 is Node no

First 0 defines x co-ordinate

Second 0 defines Y Co-ordinate

Third 0 defines Z Co-ordinate

In the 2nd Node

Command 2 is Node no

First 10 defines x co-ordinate

Second 0 defines Y Co-ordinate

Third 0 defines Z Co-ordinate

Plate (or) Element

Plate in STAAD means a thin shell with multi-nodded shape starting from 3 nodes. It can be of any things like slab, wall, or raft foundation. Each plate will have Plate number and Node numbers at each end of it.

Ex:-

Example:-

Element incidences

1 1 2 3 4

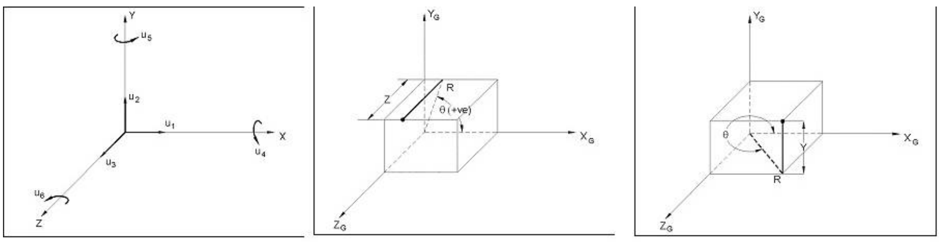
1-Plate no

1,2,3,4- Node no

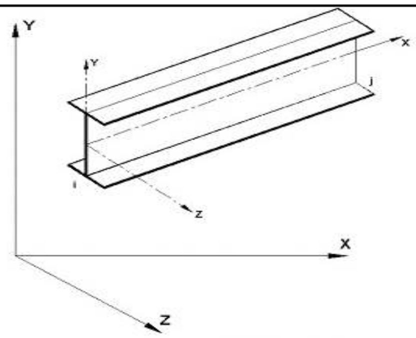
Surfaces

With the new Surface type of entity, the burden of meshing is shifted from the user to the program to some degree. The entire wall or slab is hence represented by just a few "Surface" entities, instead of hundreds of elements. When the program goes through the analysis phase, it will subdivide the surface into elements by itself. The user does not have to instruct the program in what manner to carry out the meshing.

Co-ordinate Systems

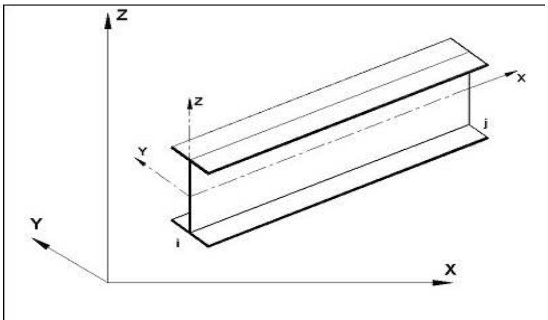


There are two types of co-ordinate systems in STAAD. They are used to build the model and to define the loadings. The Global co-ordinate system is a fixed co-ordinate system in space, which is used utilized to specify the overall geometry & loading pattern of the surface. A local co-ordinate system is related with each member or element.



When Global-Y is Vertical

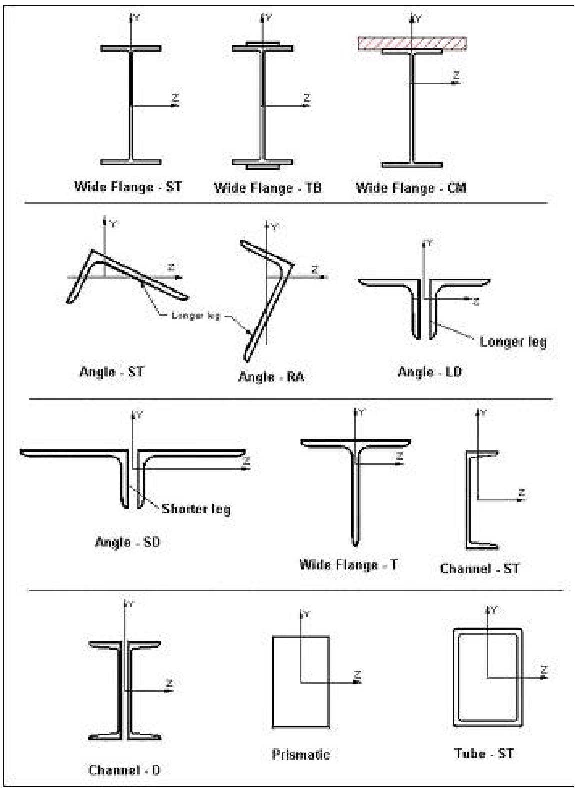
The co-ordinate system is a rectangular co-ordinate system (X, Y and Z) which follows the orthogonal right hand rule. This co-ordinate may be used to define the joint co-ordinates and load directions. The translational degrees of freedom are denoted by u1 , u2 , u3 and the rotational degrees of freedom are denoted by u4, u5 & u6.



When Global-Z is Vertical (SET Z UP is used)

Local Co-ordinate systems

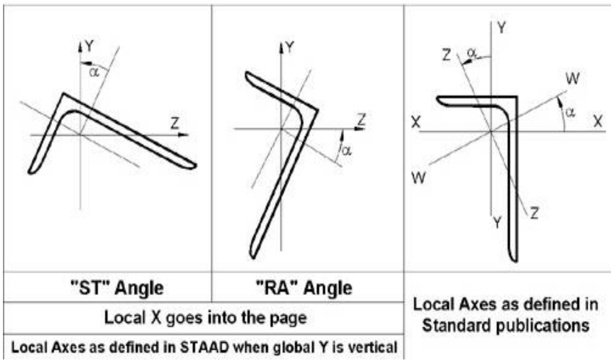
A local co-ordinate systems is associated with each member.
GX,GY,GZ-> Global Axes
X,Y,Z -> Local Area



The following figure shows local axis system (s) for various shapes that are used in the analysis .
Local axis for different cross sections:

Staring Staad

To start STAAD Pro, the following steps to be followed,



- Go to **Start/All Programs/STAAD Pro 8i/STAAD Pro**
- Double click the STAAD Short cut icon available in the Desktop

After double clicking, Staad Initial Screen appears, various parts to be explained as follows,

1. Recent Project
2. Project tab
3. Tutorials
4. License Config
5. Preview area
6. Menu Bar
7. Title bar
8. Staad online

1. Recent Tab:- This tab is open recent project from the list. Select project to open

2. Project Tab:-This tab is used to access the project details

New Project:- This tab is used to create new project.

Open Project:-This tab is used to open existing project.
Open Project wise:-This tab is used to group multiple projects.

Configuration:- Sets default configuration for STAAD.

3. Tutorials:- this tab is used get help (offline(or) online.

4. License Tab:-this tab is used to configure License code to analysis.

5. Preview Tab:- This tab is used to get project preview

Configure:-

The Configure option is available only when there is no file open in STAAD. The Configure dialog box can get from following ways (shown in the figure).

- From the **File** menu, select **Configure** option.
- From the **Project tab**, click **Configuration** option.

From any of the above, the following dialog appears, used to get project preview

Base Unit

There are two base unit systems in the program which control the units (length, force, temperature, etc.) in which, values, specifically results and other information presented in the tables and reports, are displayed in.

The base unit system also dictates what type of default values the program will use when attributes such as Modulus of Elasticity, Density, etc., are assigned based on material types – Steel, Concrete, Aluminium – selected from the program's library (Please refer to Section 5 of the STAAD Technical Reference Manual for details).

These two unit systems are English (Foot, Pound, etc.) and Metric (KN, Meter, etc.)If you recall, one of the choices made at the time of installing STAAD Pro is this base unit system setting. That choice will serve as the default until we specifically change it using this facility.

Background Colour

The drawing window can be set to have either a white background colour or a black background colour. Doing so will also set some default colours in which beams, plates and solids are drawn.

For example, a white background is accompanied by black lines for drawing beams. The user may change the colours of drawing beams, plates and solids through the facilities of the View | Options menu.

Steel Table

When an existing STAAD model is opened, the program reads the contents of the file, and checks the validity of data in that file. One of those data items validated is names of sections assigned from steel tables. Since steel sections are country-specific, such as **British, German**, etc., the program needs to know the country or organization whose steel table is the underlying database for validating the sections being read in from the file. Normally, the input file contains the name of the database as part of the member property command. In the absence of an explicit name, STAAD uses a default. That default is set using this facility.

STAAD Default Design Codes

STAAD supports several major international Design Codes. You may purchase and install one or more of those Design Codes. By default, STAAD always comes with one Design Code, depending on the country where you purchased the software. If you have purchased and installed more than one design code, this is one of the places from where you can set the default, or choose another.

Though all the design codes are installed, only the design codes you have purchased are supported by the security device (hardware lock or license manager).

The installation program will ask you to select one of these as the default code. The default design code is the one selected when the STAAD command file does not contain any design code specification.

Please note that you may not use more than one Design Code in one single run, even if the Input Command File has more than one CODE command. Also note the Design Code specified by the STAAD Design Code tab must match that specified in the Input Command File.

Input File Format

Select whether joint coordinates, member, plate and solid incidences will be written one per input line or multiple per input line. Write Expanded List instructs the program to write out joint, member or element numbers individually, for example: 1 2 3 4 5 instead of 1 TO 5 and consequently, creates voluminous input.

Working Directory:- Select the default folder to store STAAD files. Default folder is "**C:\Users\dell\AppData\Local\Temp**\". User can choose his folder.

Global axis:- Select the axis whether LOCAL (or) GLOBAL. By default Global axis system to be adopted.

STAAD Modeling

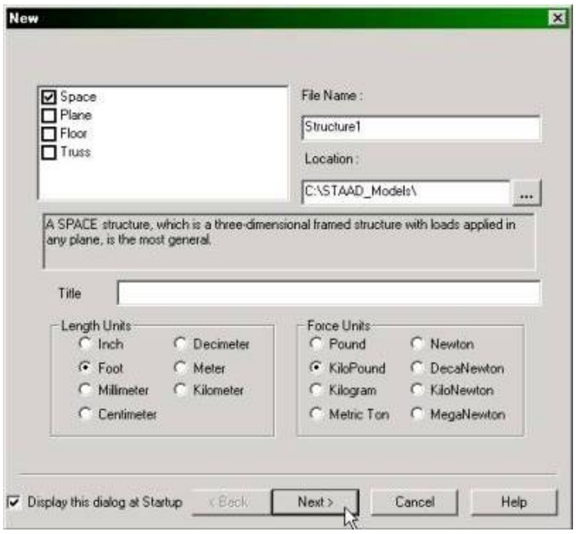
CHAPTER – III

INTRODUCTION TO STAAD PRO

This Chapter Covers:-

- Creating new project.
- Co-ordinate systems
- Creating Structure using
    - structural wizard
    - snap /grid node
    - editor /commands
    - OVERALL PROCEDURE IN WORKING WITH STAAD
    - Using excel work books.
    - Using dxf import
- Using labels

Staring New Project:-



Creating new project in STAAD Pro can be done in the following ways,

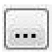
- From the **File** menu, select **new** option.
- From the **File** Toolbar, Click **New** icon.(Project Screen)
- From the **Project tab**, click **New Project** option.

From any of the above, the following dialog appears, used to get project preview

[Note:-STAAD Program can deal with single file at a time, so if you try to create a new file while another file is opened ,Staad will close OLD file.]

The parts of this dialog box are,

File Name: - Specify the name of the new file (no need to type extension, Staad will do that for you) file name is STAAD Pro can take long names.

Location: - Specify in which drive you want to save, and then specify the folder name (sub directory) . To change these settings, Simply click the three dots button  and follow dialogue will appear, choose the folder & click ok

Types of structure: - Staad has four type of structures, that explained as below,

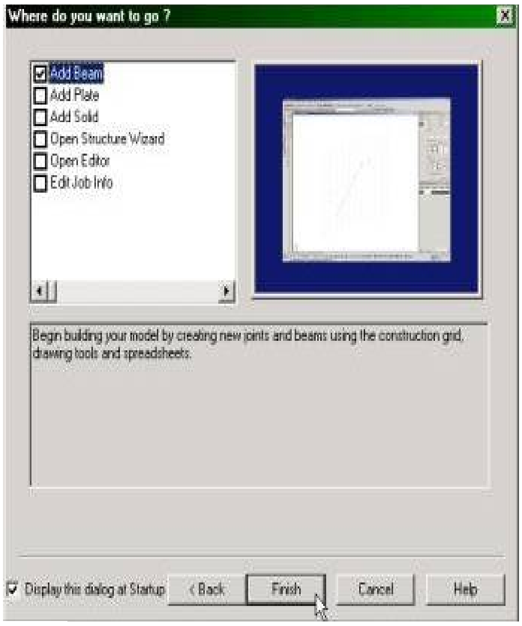
Space: - Three dimensional framed structures with load applied in any plane (The most general).

Plane: - Two dimensional structures framed in the X-Y plane with loads in the same plane.

Note:- based on axis Y plane varying(Local or Global).

Floor-Two (or) Three dimensional structure having no horizontal moment(global x or z) of the structure (FX,FY&MZ are restricted at every joint).

Truss-any structure consists of truss members only, which can have only axial member forces and no bending in the members.



Length & Force Units:-Choose the respective length & Force units i.e.. for example meter & KiloNewton.

Note:- we can change the units at any point of time inside the program (Staaad internally will make the necessary conversion).

When you finished all the options, click next button in order to proceed. Th[redacted] e following dialogue box will be displayed.

Add Beam:- Begin building your model by creating new joints and beams using the construction grid, drawing tools and spreadsheets. The Add Beam icon supports four

commands:-

- **Add Beam for Node to Node**
- **Add Curved Beam**
- **Add Beam Between Mid Points**
- **Add Beam Using Perpendicular Intersection**

Add plate:- Begin building your model by creating new joints and 3-noded and 4-noded plate elements using the construction grid, drawing

tools and spreadsheets. The Add Plate icon supports two commands:-

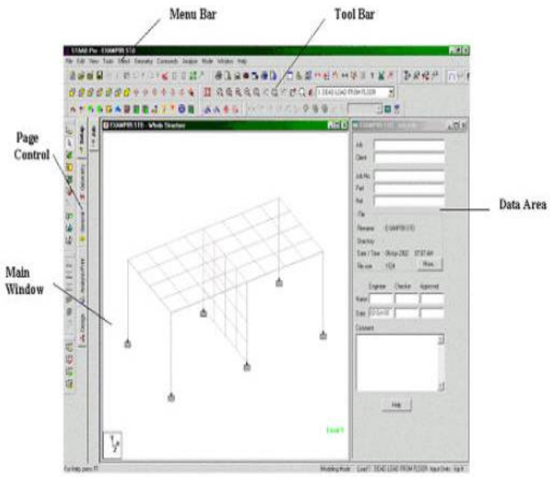
- **Add Quadrilateral Plate**
- **Add Triangular Plate**

Add Solid:- Begin building your model by creating new joints and 8-noded solid/brick elements using the construction grid, drawing tools and spreadsheets. The Add Solid icon supports five commands:- 

- **Add 8 Noded Solid**
- **Add 7 Noded Solid**
- **Add 8 Noded Solid**
- **Add 6 Noded Solid**
- **Add 5 Noded Solid**

Structural Wizard: - Begin building your model by using standard, parametric structural templates for trusses, surfaces, bay frames and much more.

Structural Editor: - Begin building your model using STAAD syntax commands (non-graphical interface) through the STAAD editor



Edit Job information:- Provide information about the job (i.e. client's name, job title, engineers involved, etc.) before building your model.

Here you to choose your option & click finish. You will get main screen. The screen part to be explained below,

1. Graphical Screen:- From This screen user has to create Geometry(or) structure.

2. Page Control:-This tab is used to execute Control & data commands

- These are the tabs that appear at the left of the main window.

- Each Page Control menu has its own sub-menu.
- Each page control has its own function , which will help the user to accomplish one of the tasks required.

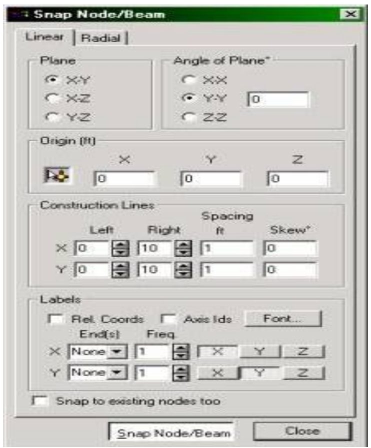
- Setup
  - Job
- Geometry
  - Beam
  - Plate
  - Surface
  - Solid
  - Parametric Models
  - Composite Deck
- General
  - Property
  - Spec
  - Support
  - Load
  - Material
- Analysis/Print
  - Pre-print
  - Analysis
  - Post-print

The Sequence of the page control and the sub page is :

3. Snap/Node Beam:- Snap/Grid Node | Beam is used to specify the grid and snap settings as well as to create members and nodes automatically by snapping to grid points.

Description

When this option is selected, the Snap Node/Beam dialog box appears, as shown below.

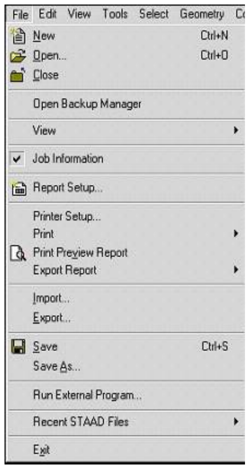


4. Tool bar:- STAAD offers a set of "**dock able**" and "**floating**" toolbars for quick access to frequently used commands. By default, the toolbar icons appear at the top of the STAAD screen immediately below the menu bar and to the left of the Page Control area. his tab is used to configure License code to analysis.

5. Menu Bar:- The menu allows the user to perform related operations such as creating a new structure model, opening the model of an existing structure, saving, printing, viewing different files, etc.

Description:-

For example, File menu is shown below in the next two figures. The File menu has different menu options depending on whether an input file is currently open or not. The following figure shows the File menu when no input file is open. The menu options are explained in the following pages. The following figure shows the File menu when an input file is open.



Various methods of constructing the structure

Following are the various methods to create a structure:

1. **Using Structure Wizard.**
2. **Using the Snap/Grid.**
3. **Using Editor/Commands.**
4. **Using Spreadsheet like Excel.**
5. **Using DXF import.**

Note:

User can use any one of these 5 methods to construct the geometry Alternatively, user can't accomplish the whole process of creating geometry with any of these methods alone; instead, he can combine any of the methods with anyone to render the final shape.

Methods 1: Using Structure Wizard:

Structure wizard is a library of predefined structure that allows us to create our required structure using simple procedures.

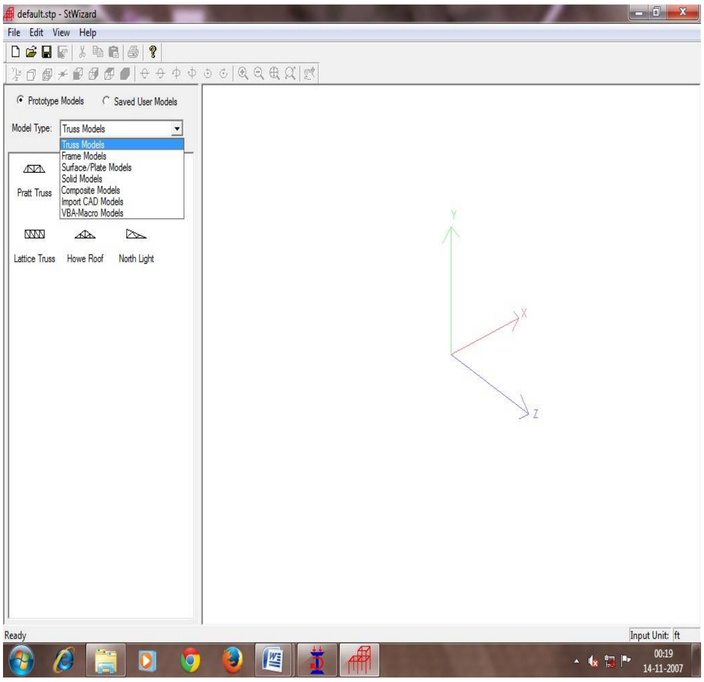
Follow these procedures to create a simple structure using wizard:

- Start **STAAD Pro**.
- Choose **Space** give **File Name** & **Location**
- Units : **Meter** and **Kilo Newton** ,Click Next
- Click Edit Job Info , Finish
- Click **Geometry**- Run **Structure Wizard**.

  (or)
- Start **STAAD Pro**.
- Choose **Space** give **File Name** & **Location**
- Units : **Meter** and **Kilo Newton** ,Click Next

- Click **Open Structure Wizard** , Finish

The following screen will appear.

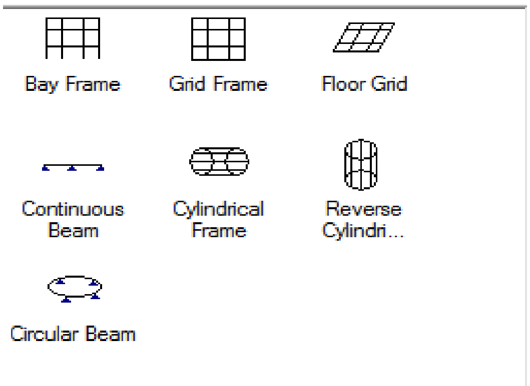


Following are the Model Types in wizard:

- **Truss Models**
- **Frame Models**
- **Surface/Plate Models**
- **Solid Models**
- **Composite Models**
- **Import CAD Models**
- **VBA – Macro Models**

Choose Frame Models, and following are the structure available:

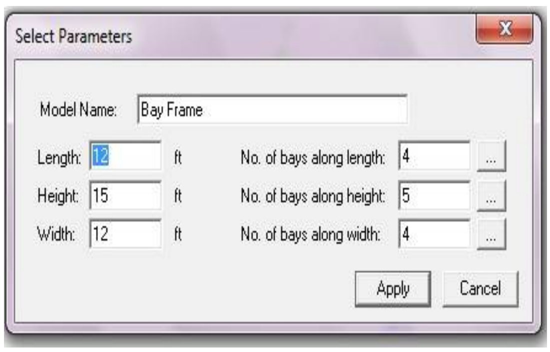
- **Bay Frame**
- **Grid Frame**
- **Floor Grid**
- **Continuous Beam**
- **Reverse Cylindrical Frame**
- **Circular Frame**



Double Click on the Bay Frame icon to setup the dimensions. The following dialogue box will be displayed.

Now specify the following inputs.

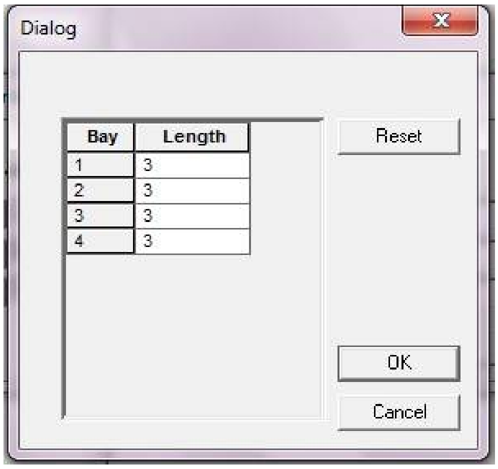
- **Length (Length is in X direction)**
- **Height (Height is in Y direction)**
- **Width (Width is in Z direction)**
- **Number of bays along length.**
- **Number of bays along height**
- **Number of bays along width**



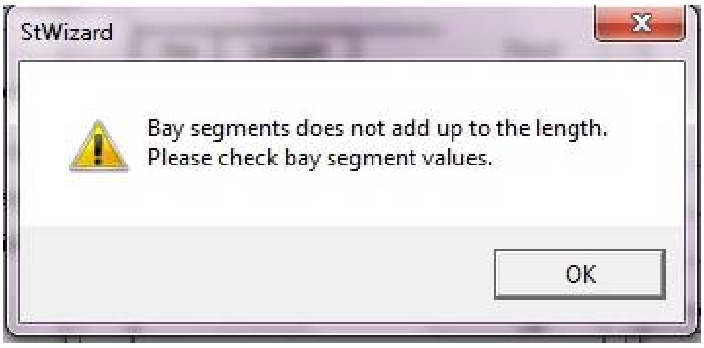
Note:

1. You should give total Height and the total Width of the structure in above dialog box.
2. All the number should be positive
3. If you don't want one of the dimension, simply set it to be zero, the structure will become two-dimensional.

If the spans are unequal, Click the button with the three dots (to the right of Number of bays filed) to set the distances of each span.



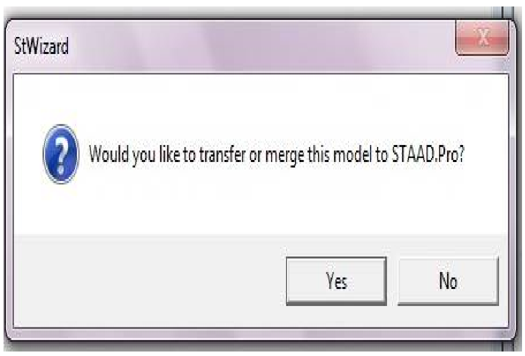
Make sure that the sum of the spans equals the spans equals the total of the dimension: otherwise STAAD Pro will produce an error message warning to you, to correct this error. Check the figure below.



Click OK to accept the number, in the dialog box.

Then click Apply on the Select Parameters box. The structure will be shown on the right.

Finally, in the wizard screen select File- Merge model with STAAD Pro model. You will see the following dialog box:



Confirm Yes. Then the screen will appear to ask, whether to move the present model to Origin of the main screen.

Click OK. Your model will be transferred to main screen.

Note:

If you already have a structure in the main screen, and again want to insert another structure from wizard, follow the same above procedure.

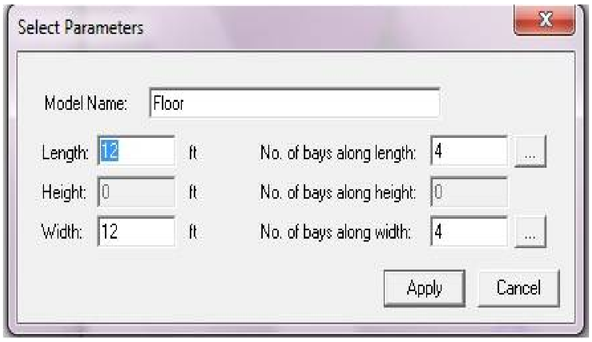
Finally, you will be required to choose where to paste the current model.

Click **Reference PT**.

[Reference point is used to paste a geometry coming from structure Wizard to a structure in the STAAD Pro windows.]

- ❖ The following screen will appear, asking you to specify the node to handle created geometry from, Select one of the Nodes, and click **OK**.
- ❖ The shape of the pointer will change.
- ❖ Click on the desired node at the structure in STAAD Pro main window. STAAD Pro will return back to the old dialogue box with the selected coordinate in X, Y & Z.
- ❖ Click **OK** to accept the results .STAAD Pro will display a message to inform the user that **Duplicate nodes ignored**, as shown below. This message means, that two nodes (one form the Original structure and one from the created geometry Click **OK**.
- ❖ The same issue applies to the beams; a new message will appear telling, **Duplicate beams ignored** as shown in the dialogue box shown below. Click **OK**. Finally the geometry is pasted in the right place.

Likewise, now we shall create other structure in frame models.



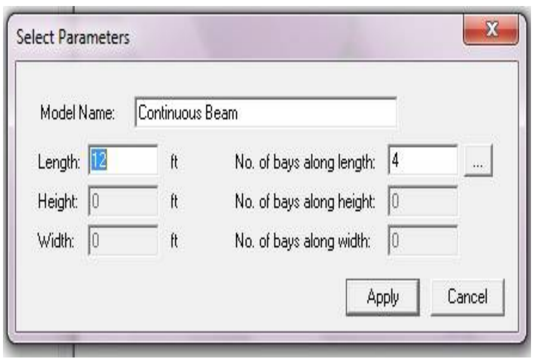
Frame models->Grid Frame:

Grid Frame is just like Bay Frame with one exception. It creates ground beams in the X-Z plane of the difference. All other procedures are same. Following diagram shows the difference.

Frame Models -> Floor Grid :

Floor Grid is a two –dimensional structure in the X-Z plane only. The purpose of floor grid is to create a mesh o beams in the X and Z direction.

Double –Click on the Floor Grid icon, the following dialogue box will be shown.



Frame Models -> Continuous Beam:

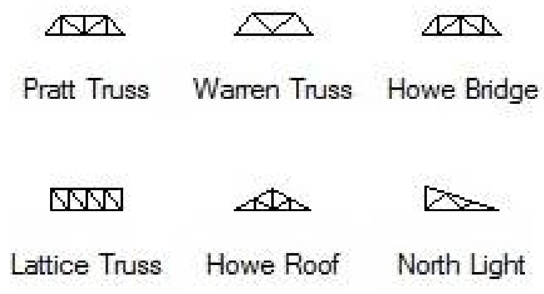
Continuous Beam is one dimensional structure in the X direction only.

Double click continuous Beam icon, the following dialogue box will be shown. Note that Height (Y Axis) and Width (Z Axis) are not available for editing.

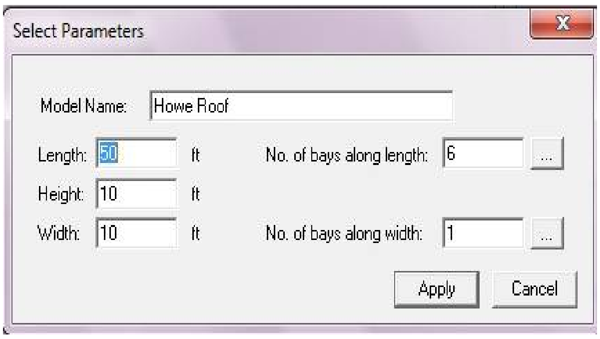
Note:- All other frame models are dealt in advanced chapters.

Truss models:

Choose Truss Models under model types and following are the structures available:



Double click on any icons, you will get same dialogue box for all six shapes , as shown in the dialogue box below,



As you can see from the dialogue box, you can change the following parameters,

- ❖ **Total Length (in X direction)**
- ❖ **Total Height (in Y direction)**
- ❖ **Total WIDTH (IN Z direction), for 3d trusses only**
  - ▪ **Note:-If want 2D set it to)**
- ❖ **Number of bays in along length**

(**Note**:-the parameter will decide the shape of the truss)

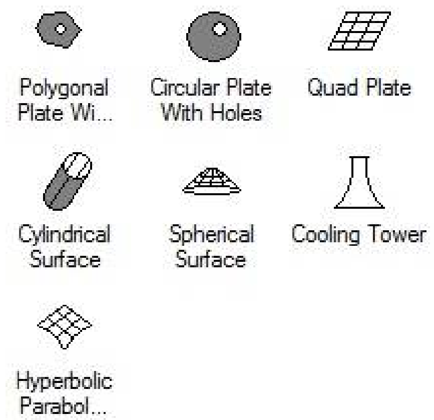
- ❖ **Number of bays along width.**

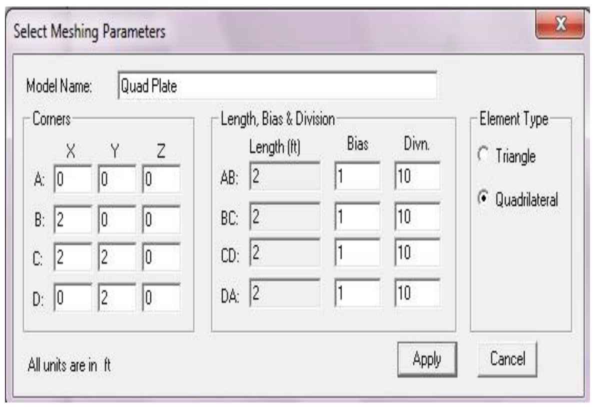
  (**Note**:-set it to zero if you want 2D truss)

  The rest of the procedure is same as the Frame Models.

Surface /Plate Models:

Choose Surface/Plate Models under





model types and following are the Structures Available:

To create 3-Noded or 4-Noded plates , double click on **Quad Plate** , you will see the following dialog box ,

From the element type (Upper right portion of the dialogue box) specify if you want triangle shape (3-noded) or Quadrilateral shape (4-noded).

You have 4 corners to specify A, B , C & D which they will be the corner of desired plate. The XYZ here doesn't mean the real XYZ of the space we are using in Staad Pro window , but rather XYZ of the structure wizard. This point can attach to any other point in the Staad Pro Window. Also the use of the XYZ is a very good way to tell Structure Wizard in which plane you will create your plate.

As an example for the last one ,check the following 4 corners :

A=0, 0, 0;

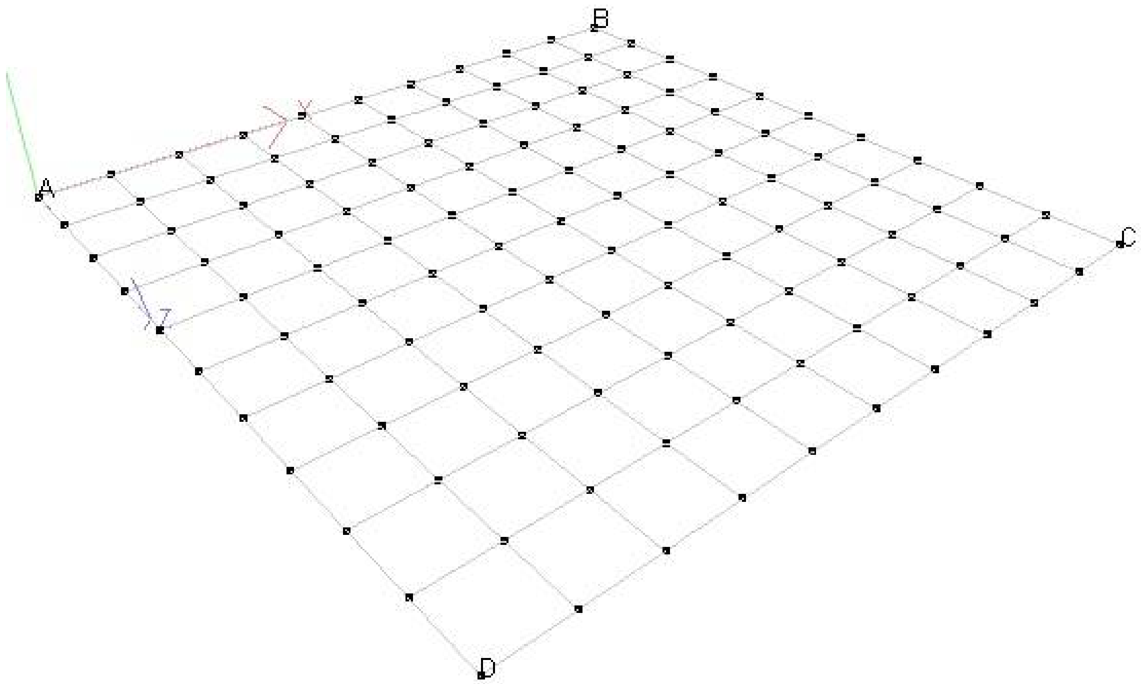
B=10, 0, 0;

C=10, 0, 10;

D=0, 0, 10

The result will show as the shape below:

As you can see the Y co-ordinate is always 0, hence the plate is in the X-Z plane , this is a good geometry for slab.



While giving the coordinate of the 3 or 4 nodes, you must be consistent, either rotate clockwise (CW) or counter clock wise (CCW).

In the Bias and Division parts, specify the number of diversions each side of the plate will be divided to. By default Bias=1, means the divisions are equally spaced. Dividing a plate we will more than one plate (one plate here means one entity). Example would be if you have a plate 10x10 m plate divided by 10 divisions from each side, therefore the total number of smaller plates will be 10 plates in each of 1mx 1m .

Click Apply. Then follow the same procedure as seen previous to transfer model to main screen.

Note:-

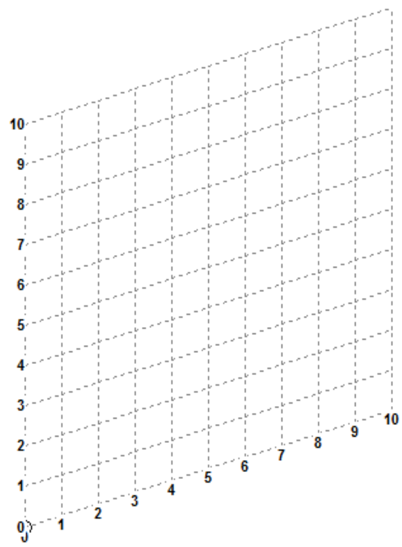
- All other model types like solid, Composite models etc are default in advanced chapters.
- Click on the following toolbar buttons in Structure Wizard shown below & practice with the following views ant rotations after creating the Structures.

Methods 2: Using Snap/Grid:

Snap/Grid is a method of creating structure using snap/grids that allows us to create our required structure using simple procedures.

Follow these procedures to create a simple structure using wizard:

- Start **STAAD Pro**.
- Choose **Space** give **File Name** & **Location**
- Units : **Meter** and **Kilo Newton** ,Click Next
- Click **Add Beam(or)Add Plate(or)Add Solid** , Finish



Staad main screen will appear (**Add Beam/Add Plate/Add Solid** is explained in the next page).Follow any one of the below methods to have the following screen:

Automatically snap/grid enable to make by default type (**beam/plate/solid**).By default 10x10 grid is appears in the XY Plane. Because 10x10 grid not enough to complete a structure, we have to customize the Grid by the following ways,

- If you want to modify the default, click **edit** button or to create new grid click **Create** button .he Floor Grid icon, the following dialogue box will be shown.

Here there are three type of Grids settings available .They are

- **Linear grid(For Linear Structures)**
- **Radial grid(For Curve Structures)**
- **Irregular grid(For Irregular Structures)**

Linear Grid

Follow the steps below to customize the linear Grid

- **Decide the plane that you want to work (XY, YZ or ZX plane).**
- **Specify Angle of Plane (Leave it 0 for now) & Pick angle plane(X-X,Y-Y or Z-Z plane).**
- **Specify Origin point (Preferable to leave it at 0, 0, 0).**
- **Specify Left & Right Counts in the construction line area.**
- **Specify the spacing & skew in the construction lines**

After completing the above steps click OK to close.

Radial Grid

Follow the steps below to customize the Radial Grid

- **Decide the plane that you want to work (XY, YZ or ZX plane).**
- **Specify Angle of Plane (Leave it 0 for now) & Pick angle plane(X-X, Y-Y or Z-Z plane).**
- **Specify Origin point (Preferable to leave it at 0, 0, 0).**
- **Specify Start Angle, Sweep & Bays Counts in the construction line area.**
- **Specify the Radis1,Radius 2 & Bays in the construction lines**

After completing the above steps click OK to close.

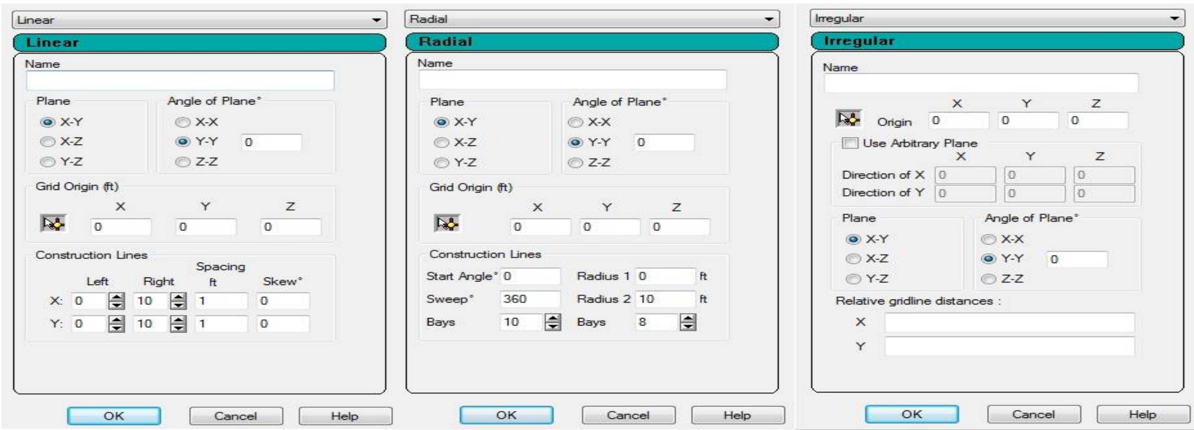
Irregular Grid

Follow the steps below to customize the Irregular Grid

- **Specify Origin point (Preferable to leave it at 0, 0, 0).**
- **Specify Distance offset in arbitrary plane as xyz co-ordinate values.**
- **Decide the plane that you want to work (XY, YZ or ZX plane).**
- **Specify Angle of Plane (Leave it 0 for now) & Pick angle plane(X-X, Y-Y or Z-Z plane).**
- **Specify the Relative grid line distance in the X & Y axis.**

After completing the above steps click OK to close.

After completing any of the above choose grid line & make Geometry by



using following steps,

- **Select from the page control/Geometry tab (or)**
- Select from the menus **Geometry/Snap Grid Node** then **Beam/Plate/Solid**
- From the **Geometry toolbar** or **Geometry menu,** select one of two available **Snap node/Beam or Snap Node/Plate.**

Adding Beams

Make Sure that **SNAP NODE/BEAM** is on.

To start drafting **Beams**, go to start **NODE Coordinate** and click , a node will be inserted here , and click , a second new mode will be added and according a new beam will be created. Keep on doing this until you are done, the click close button

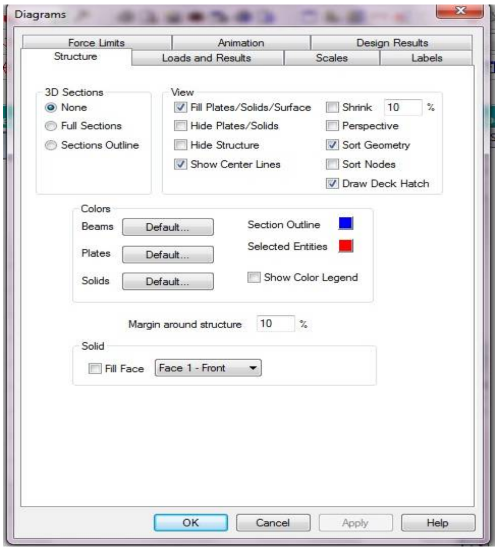
Note:-Once you starting clicking nodes , controlling point will restrict to start your next beam from the last node reached. In order avoid this hold Control Key and click on the desired node other than the last node anf you can start next beam from that Node.

Adding Plates

Make sure that **Snap Nose /Plate** is On.

To start drafting Plates, Go to the Start Node Coordinate and click, a node is inserted here. Repeat this process for four points, a plate will be added .Then you are done click **close**.

Note:- Once you finish the first plate , the controlling point will restrict you to start next plate from the last node reached. To avoid this , hold the Ctrl key and click on the coordinate desired other than the last node and you can start your next plate from that node.

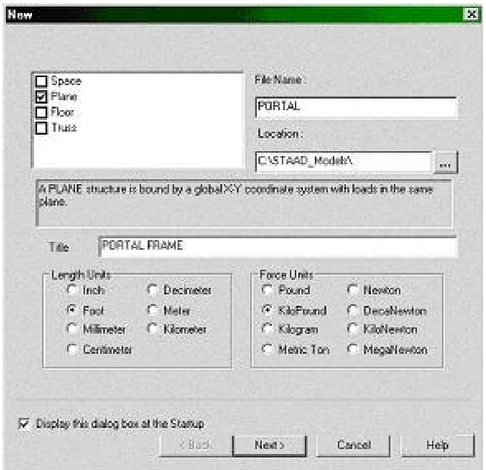


<u>Fill Plates</u>

To view your plates for the procedure below,

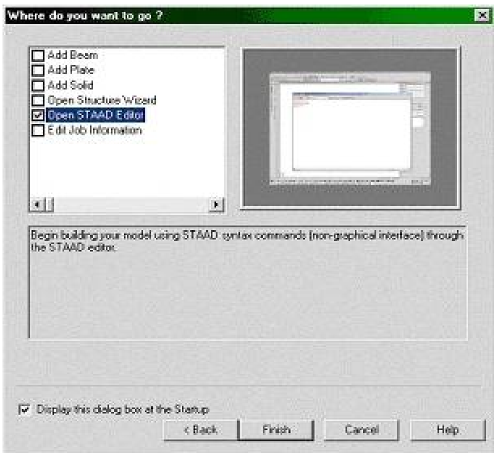
- ➢ In the Staad Pro Window, right click anywhere, menu screen will appear, select from it Structure Diagrams. The following dialogue box will appear :
- ➢ Under View, Click **Fill Plates/Solids/Surfaces** On, click **Sort Nodes** On.

Methods 3: Using Editor/Commands:



This Method is used to create Structure by writing Program Codes using Editor Window. The advantages of Editor are as follows,

- ➢ Knowledge of the STAAD language can be very useful in utilizing the large number of facilities available in the program.
- ➢ The user can easily make changes to the input data if he/she has a good understanding of the command language and syntax of the input.
- ➢ The input file represents the user's thought about what he/she wants to analyze or design. With the knowledge of the STAAD command language, the user or any other person can verify the accuracy of the work.

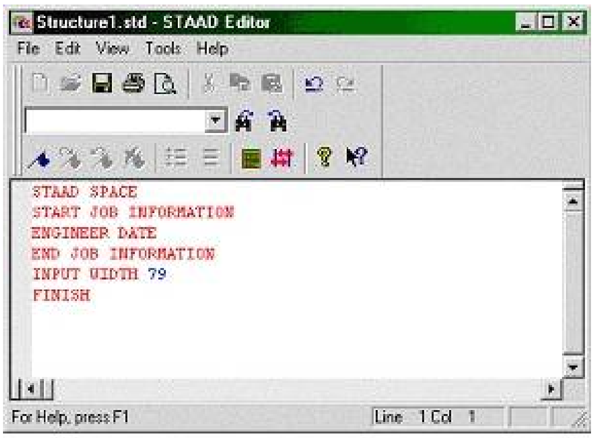


The commands used in the input file are explained in Section 5 of the STAAD Technical

Reference Manual. Users are urged to refer to that manual for a better understanding of the language.

Follow these procedures to create a simple structure using wizard:

- Start **STAAD Pro**.
- Choose **Space** give **File Name** & **Location**
- Units : **Meter** and **Kilo Newton** ,Click Next
- Click **Open Staad Editor** , Finish



[Note:-Alternatively, any standard text editor such as Notepad or WordPad may also be used to create the command file. However, the STAAD Pro command file editor offers the advantage of syntax checking as we type the commands. The STAAD Pro keywords, numeric data, comments, etc. are displayed in distinct colors in the STAAD Pro editor.]

Some of the basic pasic Programming Concept to be explained below,

NODES

To create we have to use **Joint Coordinates** Specification method

These commands allow the user to specify and generate the coordinates of the JOINTs of the structure. The JOINT COORDINATES command initiates the specification of the coordinates. The General format for Joint Co-ordinate is explained below:

JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) band-spec

$\mathbf{i_1, x_1, y_1, z_1, (i2, x_2, y_2, z_2, i_3)}$

Description

The command

- The command JOINT COORDINATES specifies a Cartesian coordinate system. Joints are defined using the global X, Y and Z coordinates.

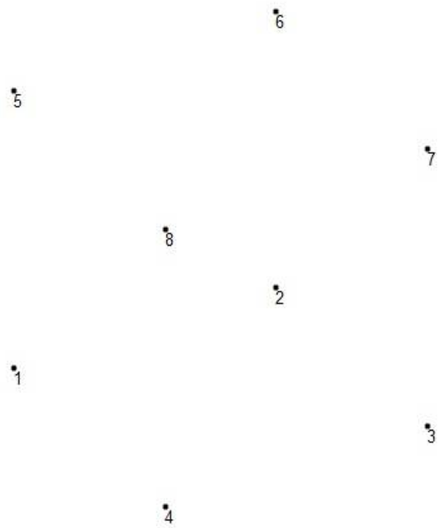
- The command JOINT COORDINATES CYLINDRICAL specifies a Cylindrical Coordinate System. Joints are defined using the r, q and z coordinates.
- JOINT COORDINATES CYLINDRICAL REVERSE specifies a Reverse Cylindrical Coordinate system (see Figure 1.4). Joints are defined using the r, y and q coordinates.

* i_1 = the joint number for which the coordinates are provided. Any integer number (maximum 6 digits) within the limit is permitted.

X1, y1 and z1 = X, Y & Z (R, & Z for cylindrical or R, Y & for cylindrical reverse) coordinates of the joint.

For PLANE analyses z1 is an optional data item when defining input for individual joints. Z1 is always required for joint generation. The following are used only if joints are to be generated.

* i_2 = the second joint number to which the joint coordinates are generated.



X2, y2, and z2 = X, Y & Z (R, & Z for cylindrical or R, Y & for cylindrical reverse) coordinates of the joint i_2.

Example 1

To create a room of size 20 feet length & 15 feet width, Open the staad Editor same as above & Enter following codes,

STAAD SPACE

INPUT WIDTH 79

uni ft kn

Joint coordinates

1 0 0 0

2 10 0 0

3 10 0 10

4 0 0 10

5 0 10 0

6 10 10 0

7 10 10 10

8 0 0 10

FINISH

Close & save the editor, The Result Screen is shown above.

(Note:- Use SHIFT+K to view nodes & SHIFT +N to node numbers)

To create multi-storey structures, user cannot be able to all calculate by using method. To avoid this, we have gone for **REPEAT & REPEAT ALL** functions

- The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified coordinate increments.
- The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command, or all joint data if no previous REPEAT ALL command has been given. Note: Use "REPEAT ALL 0" to start a section of data to be repeated if necessary. (When using the REPEAT and REPEAT ALL commands, joint numbering must be consecutive and should begin with 1.)
- To create a room of size 20 feet length & 15 feet width, Open the staad Editor same as above & Enter following codes,

JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) band-spec

i_1, x_1, y_1, z_1, (i_2, x_2, y_2, z_2, i_3)

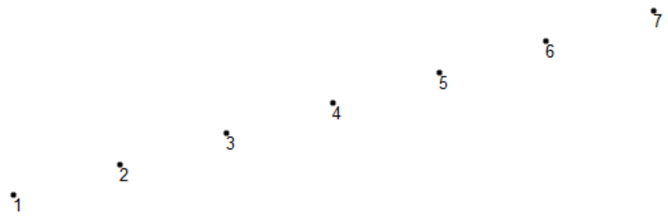
REPEAT n, xi_1, yi_1, zi_1, (xi_2, yi_2, zi_2... xi_n, yi_n, zi_n)

REPEAT ALL n, xi_1, yi_1, zi_1, (xi_2, yi_2, zi_2,..., xi_n, yi_n, zi_n)

n is limited to 150

STAAD SPACE

INPUT WIDTH 79



uni ft kn

Joint coordinates

1 0 0 0 7 60 0 0

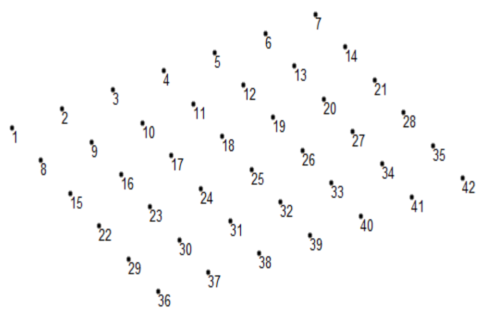
FINISH

Note:-Here Staad takes 1 0 0 0 as staring node 7 60 0 0 as ending node. That means Staad Calculates Difference between Nodes & calculates Axis Difference & Inserts all point in between 1 & 7.Instand of that ,if you give 1 0 0 0 ; 7 60 0 0 that means only 2 nodes staring Ending.

Now Single column span is created, to copy in to floor level we have to use repeat function. the syntax is

Repeat 5 0 0 10

Add the above syntax after 1 0 0 0 7 60 0 0 Above command copy the last step 5 time in the z axis. The output is shown above.



NOCHECK= Do not perform check for multiple structures or orphan joints.

JTORIG xOrigin yOrigin zOrigin

band-spec = (NOREDUCE BAND)

The JTORIG command should be entered on a separate command line. Basically after the joint coordinates are entered or generated, then the xOrigin, yOrigin, and zOrigin values are added to the coordinates. For example a cylinder could be generated about the Y axis then moved by this command to its proper place. To create multiple offset structural parts, enter additional JOINT COORDINATES commands, each one followed by its JTORIG command. An example showing the use of this command is provided later in this section.

The multiple JOINT COORDINATES command concept allows UNIT changes and PERFORM ROTATION commands in between; such that these commands would apply to a selected portion of the joints. However, the PERFORM ROTATION command applies to all prior defined joints, not just those in the previous JOINT COORDINATE command.

NOREDUCE BAND causes the program to execute without performing a bandwidth reduction.

Methods 4: Using Spread Sheet like Excel:

In This Method you to type each node joint or node coordinate in the spread sheet, which is given in STAAD to construct the required Structure.

- Start Space
- Choose Space, give File name & location.
- Units **Meter** and **Kilo Newton**, Click next

Select **Geometry/Nodes** from the menu, then type all joint coordinates, side by side node is seen on the main screen. After constructing nodes, **Right click** on the screen and choose **Add Beam** or from Geometry menu select **Add Beam/Add Beam from point to Point**.

Hint:-

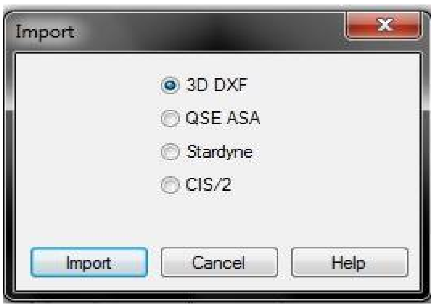
If you want to connect columns & beams quickly first select all nodes, then go to **Geometry/Connect Beams along X/Z axis** for beams & **Geometry/Connect along Y Axis** for Columns.

You can also add Plates like this from Geometry menu.

Methods 5: Using DXF Import:

You should know AutoCAD somewhat, before using this method. Following are the steps to create a structure using this method,

Procedure-A

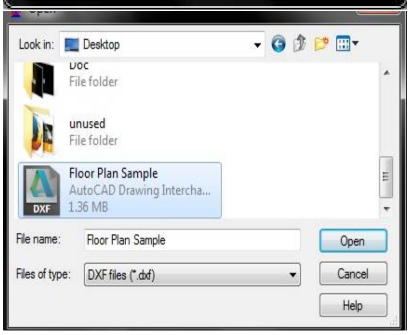


- Start **AutoCAD**
- Draw your structure(2d or 3d)
- Close & save file in the DXF format.

  Note:- AutoCAD default Format is Dwg, You have to change file type to DXF from the Type dialogue.

To import the DXF file saved above, you to follow any one of the method explained below,

a) From Structure Wizard,
b) **From File Import**



Procedure-B

a) **From Structure Wizard:-**

- From the model Type select **Import Cad Models**.
- Double click on the scan **DXF** icon, a dialogue box will appear to make you select file name.

➢ The **DXF** will be scanned and opened.

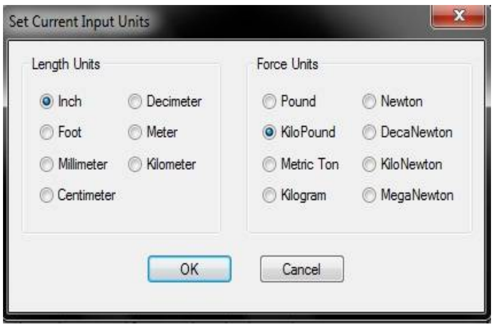
b) From Structure Wizard:-

➢ Start Staad Pro.
➢ Start New Project using above Procedure.
➢ After Getting new project.
➢ From File Menu Select **File/Import**.

Staad asks now about the type to be imported:

❖ Select **3D DXF** and **import**. STAAD Pro will ask the location of your DXF file.
❖ Select the file, and Click Open. The following dialogue box will appear.
❖ Now Select one of the three choices, and click ok:
  - **No Change**:-The XYZ orientation of STAAD matches the XYZ in AutoCAD.
  - **Y up**:-You are telling STAAD to consider Y is up in Staad, and hence to convert Y in Autocad Accordingly. (This is the right choice in almost all of the cases.
  - **Z up**:- You are telling Staad to consider Z is up in Staad, and hence to convert Z in AutoCAD accordingly.

❖ The Following of the dialogue will appear:-
❖ Select the Proper Length & Force Unit and click OK, the structure will transferred.



Note:-

1. In AutoCAD, use always Line, in drafting Beams & Columns
2. Staad will consider one line as equal to one Beam or Column, hence long line covering more than one node Will be considered as one object. According cut your lines to be small and connecting two Nodes only
3. Use the latest AutoCAD version with the latest Staad Versions.

Viewing:

Various Viewing tools can be explained below,

	View from +Z, you consider it the Front View
	View from -Z, you consider it the Back View
	View from +X, you consider it the Left View
	View from -X, you consider it the Right View
	View from +Y, you consider it the Top View
	View from -Y, you consider it the Bottom View
	Isometric -it the Isometric View
Rotations	
	Rotate up & Down, Rotating around the X axis
	Rotate left & Right, Rotating around the Y axis
	Rotate Left & Spin Right, Rotating around the Z axis

You can use the arrows in your key board also.

Use:-

- Right arrow and Left Arrow to rotate around Y axis.
- Up Arrow and down arrow to rotate around X axis.

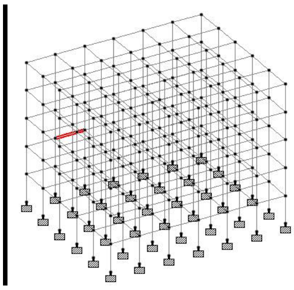
Selecting:

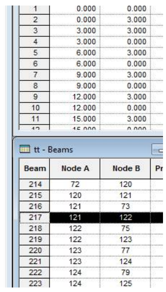
Various Selecting tools can be explained below,

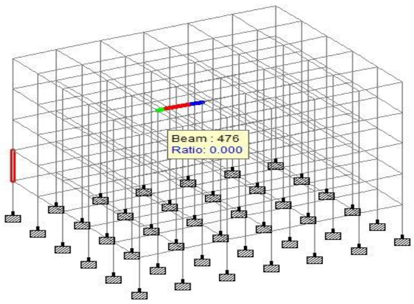
	Nodes Cursor, to select nodes.
	Beams Cursor, to select Beams.
	Plates Cursor, to select plates.
	Surface Cursor, to select Surfaces.
	Solid Cursor, to select Solid.

Single Selection:

- Click ON the desired node, Beam, (or) Plate, it will be highlighted by running in to red.
- From the **Data/Area**, click on the numbers of the node, Beam o Plate , it will highlighted. Check the picture below,

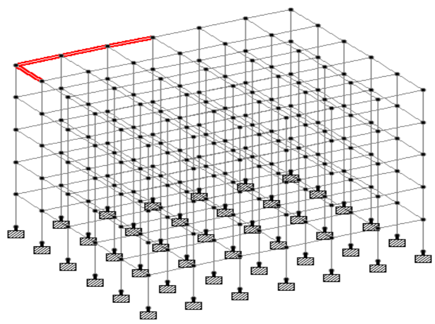




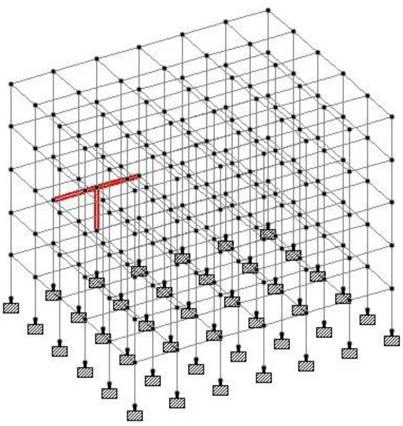


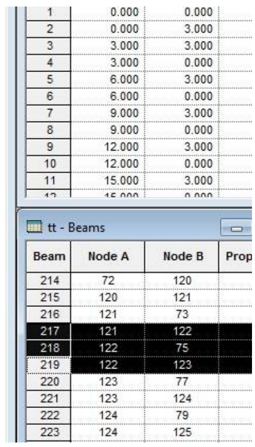
Multiple Selections:

- Select any Node, Beam, or Plate First, then hold down the ctrl key, and click other nodes, Beams and Plates.



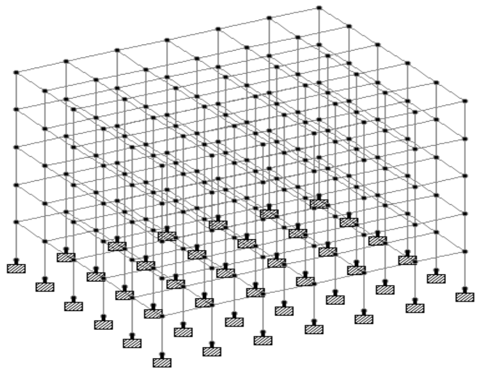
- From the Data/Area, Click on the number of the Node, Beam, or Plate. It will be highlighted. Then hold Ctrl key, and click on other numbers; it will highlighted as Well.





- Make a window around the needed nodes; Beams, (or) Plates by clicking in any empty place of the Window, and Holding down the left Button, moving to the other corner and releasing the button, whatever inside the window will be selected based on cursor we are using.\

➢ joint coordinates command. It is useful in instances such as when the centre of cylinder is not at (0, 0, 0) but at a different point in space.



Select While Viewing 3D Geometry:

Using both Viewing commands and Selecting methods leads to effectively select multiple **Nodes, beams, or Plates**, in 3D Geometry. Looking at a 3D model from different viewing points will enable the user to select Nodes/Beams or Plates in the plane shown and any things behind it.

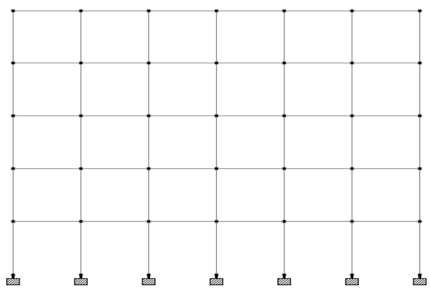
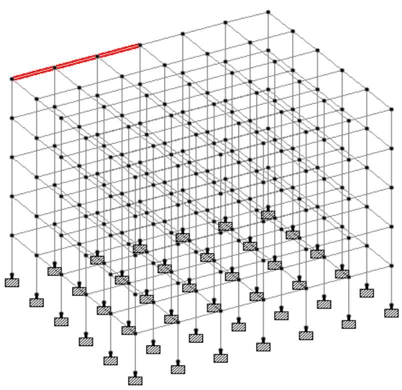
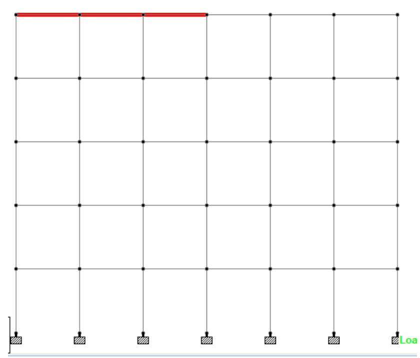


Figure 13d Isometric

View Figure 2 View From +Z

Now click on the Beams as shown,

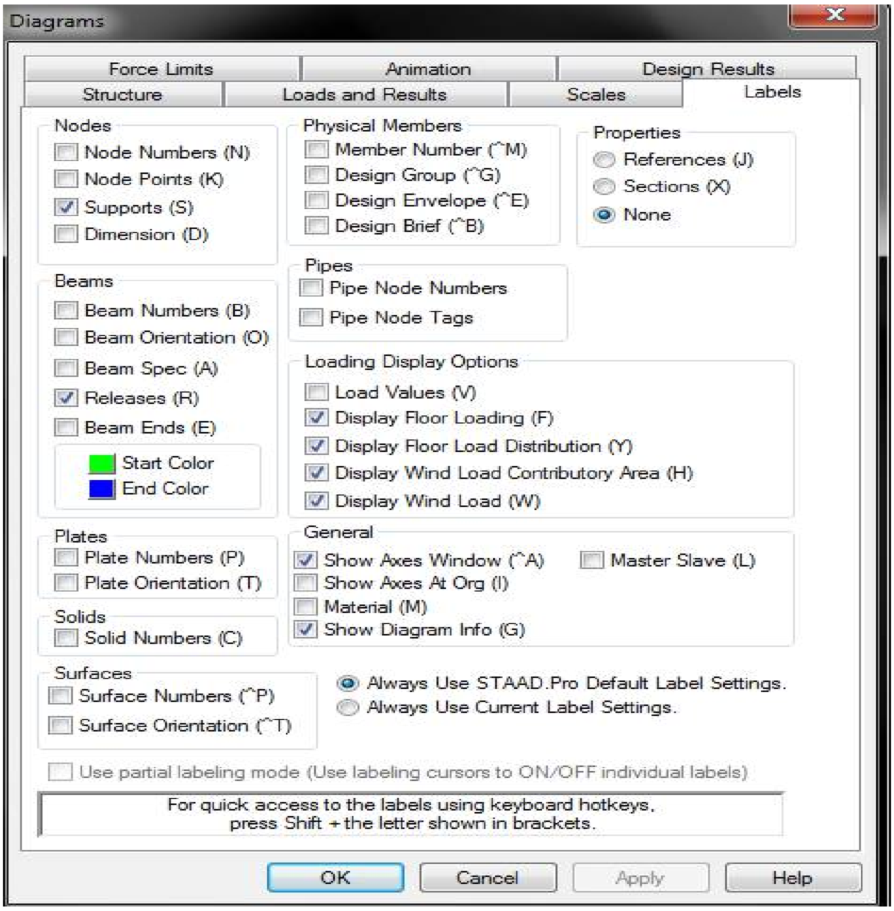




Using Labels:

In the STAAD Pro window, **Right click** on any place, a menu will appear, select from it **labels**, a large dialogue box will appear as below,

❖ Turn on Node Numbers & Node Points, Beam Numbers, Plate Numbers to see element nos.



CHAPTER – IV

Other Functions useful For Construction of Geometry

This Chapter covers:

- Inserting nodes
- Placing plates, surface & Solids
- Translational Repeat
- Circular Repeat
- Mirror, Rotate, move
- Adding Beams (connecting & intersecting)
- Cut section
- Renumber
- Delete, undo/redo, dimension etc

INTRODUCTION

We have discussed four methods in the previous chapter that are used to create the basic geometry. However, it alone cannot fulfill the creation of some complex requirements of any structural engineer.

So when you complete this chapter, you will be finishing all the requirements to construct any type of Structure.

In this module, we will discuss essential functions, which will enable the user to complete any unusual requirements in the building up the geometry.

All of the functions to be discussed require to Inserting Nodes, Plates, Surfaces & Solids.

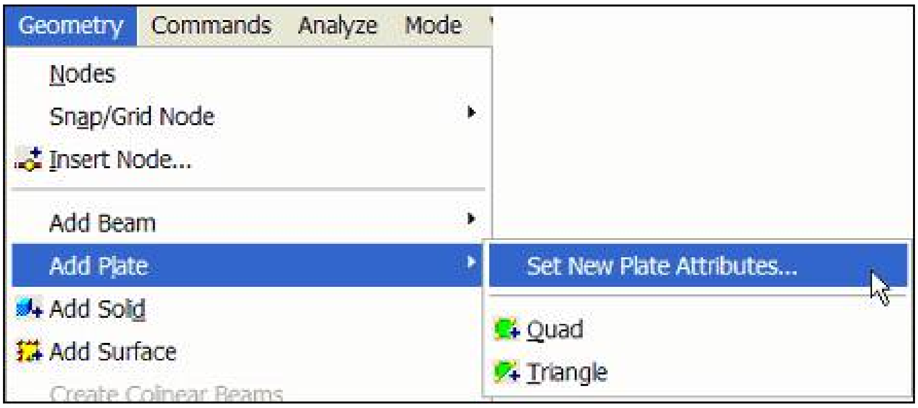
Inserting Plates

This option allows you to add Triangular or Quadrilateral plate elements by connecting existing nodes.

Description

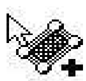
To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated. Methods to create as follows,

- ❖ To add Quadrilateral plate, select Quad from **Geometry/Add Plate/Quad** menu.



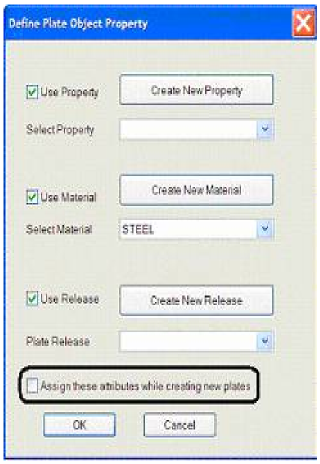
For triangular plates, select **Geometry/Add Plate/Triangle** from the sub-menu. The cursor changes to Quad Plate or Triangular Plate as shown below.

- ❖ From tool bar click, Plate icon appears, this 4 nodded plates. To get more options, hold down the left mouse button when clicking on the button. Icons that have this property are identified with a black triangle in their lower right corner:-



Set New Plate Attribute

Similar to the "**Set New Member Attribute**" command introduced in STAAD Pro 2004, a new command, in which the user can define the property, material and releases to each new plate element as it is created, has been introduced.



In order to define the attributes for a plate element before they are created, go to **Geometry | Add Plate | Set New Plate Attributes** from the main menu.

A dialog box will prompt for various attributes of the plate to be pre-defined. A summary of the specific attributes are defined in the table below.

Button	Function
Create New Property	Prompts the "Plate Thickness" dialog box where the thicknesses of the plate at each of the corner nodes can be defined.
Create New Material	Define the various material properties of the plate including Poisson's ratio, modulus of elasticity, shear modules, etc.
Create New Release	Define the degrees of freedom to be released at each node of the plate as well as define the plate to be plane stress, no in-plane rotation or no stiffness.

Multiple properties, releases and materials can be created and saved for future use. To choose from various pre-defined types, simply select the appropriate definition using the "Select Property", "**Select Material**" or the "**Plate Release**" drop-down boxes.

For the program to recognize the pre-defined attributes, the "**Assign these attributes while creating new plates**" check box must be checked. Any new plate element created from here on (whether created individually, through a mesh or Structure Wizard) will now possess these attributes.

Surfaces:-

For any surface type of structural component, modeling requires breaking it down into a series of plate elements for analysis purposes. This is what is known in stress analysis parlance as meshing. When a user chooses to model the surface component using plate elements, he/she is taking on the responsibility of meshing. Thus, what the program sees is a series of elements. It is the user's responsibility to ensure that meshing is done properly.

With the new Surface type of entity, the burden of meshing is shifted from the user to the program to some degree. The entire wall or slab is hence represented by just a few "Surface" entities, instead of hundreds of elements. When the program goes through the analysis phase, it will subdivide the surface into elements by itself. The user does not have to instruct the program in what manner to carry out the meshing.

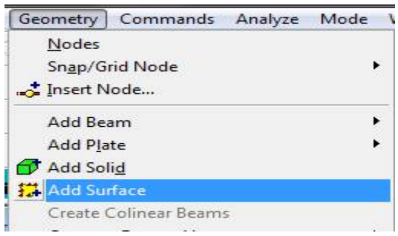
Inserting Surfaces

This option allows us to add Surface elements by connecting existing nodes.

Description

When we select this option, the cursor changes to a Surface cursor as shown below.To create new solid elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the element being generated.

To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated. Methods to create as follows,



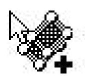
- ❖ To add Quadrilateral plate, select Quad from **Geometry/Add Surface** menu..
- ❖ From tool bar click, **Add Surface** icon appears, this creates 5 nodded Surfaces.

Inserting Solids

This option allows us to add solid elements by connecting existing nodes.

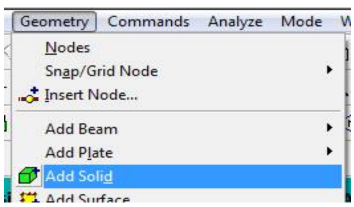
Description

When we select this option, the cursor changes to a Solid cursor as shown below.



To create new solid elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the element being generated.

To create new elements, simply click on the existing nodes in the right sequence. A rubber-banded area shows the boundary of the plate being generated. Methods to create as follows,



- ❖ To add Quadrilateral plate, select Quad from **Geometry/Add Solid** menu..
- ❖ From tool bar click, Add Solid icon appears, this creates 8 nodded Solids. To get more options, hold down the left mouse