STAAD.*Pro* 2005

GETTING STARTED

AND

TUTORIALS



www.reiworld.com www.reel.co.uk **STAAD.***Pro* **2005** is a proprietary computer program of Research Engineers, International (REI), a division of netGuru, Inc.

The program and this document have been prepared in accord with established industry engineering principles and guidelines. While believed to be accurate, the information contained herein should never be utilized for any specific engineering application without professional observance and authentication for accuracy, suitability and applicability by a competent and licensed engineer, architect or other professional. REI disclaims any liability arising from the unauthorized and/or improper use of any information contained in this document, or as a result of the usage of the program.

RELEASE 2005

Copyright Research Engineers, Interntional Division of netGuru, Inc. Published July, 2005

About STAAD.Pro 2005

STAAD.Pro is a widely used software for structural analysis and design from Research Engineers International.

The STAAD.Pro software consists of the following:

The STAAD.Pro Graphical User Interface (GUI): It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

The STAAD analysis and design engine: It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, Timber and Aluminum design.

About the STAAD.*Pro* 2005 Documentation

The documentation for STAAD.Pro consists of a set of manuals as described below. These manuals are normally provided only in the electronic format, with perhaps some exceptions such as the Getting Started Manual which may be supplied as a printed book to first time and new-version buyers.

All the manuals can be accessed from the Help facilities of STAAD.Pro. Users who wish to obtain a printed copy of the books may contact Research Engineers. REI also supplies the manuals in the PDF format at no cost for those who wish to print them on their own. See the back cover of this book for addresses and phone numbers.

Getting Started and Tutorials: This manual contains information on the contents of the STAAD.Pro package, computer system requirements, installation process, copy protection issues and a description on how to run the programs in the package. Tutorials that provide detailed and step-by-step explanation on using the programs are also provided.

Examples: This book offers examples of various problems that can be solved using the STAAD engine. The examples represent various structural analyses and design problems commonly encountered by structural engineers.

Graphical Environment: This manual contains a detailed description of the Graphical User Interface (GUI) of STAAD.Pro. The topics covered include model generation, structural analysis and design, result verification, and report generation.

Technical Reference: This manual deals with the theory behind the engineering calculations made by the STAAD engine. It also includes an explanation of the commands available in the STAAD command file.

International Design Codes: This document contains information on the various Concrete, Steel, and Aluminum design codes, of several countries, that are implemented in STAAD. Generally, this book is supplied only to those users who purchase the international codes utilities with STAAD.Pro.

OpenSTAAD: This document contains information on the library of functions which enable users to access STAAD.Pro's input and results data for importing into other applications.

<u> Part - I</u>

Getting Started

System Requirements Installation Start-up

Table of Contents

1.	Introduction	1
2.	Hardware Requirements	2
3.	Contents of the STAAD.Pro CD	4
4.	Installation	6
5.	Copy Protection Device	14
6.	Running STAAD.Pro	16
7.	Running STAAD.etc	18
8.	Running Sectionwizard	19
9.	Running STAAD.foundation	20
10.	Running Mesher	21

1. Introduction

STAAD.Pro is an analysis and design software package for structural engineering. This manual is intended to guide users who are new to this software as well as experienced users who want specific information on the basics of using the program.

Part-I of this manual describes the following:

- Hardware Requirements
- Contents of the STAAD.Pro CD
- Installation
- Copy Protection Device
- Running STAAD.Pro

Part II of this manual contains tutorials on using STAAD.Pro. The tutorials guide a user through the processes of:

- Creating a structural model. This consists of generating the structural geometry, specifying member properties, material constants, loads, analysis and design specifications, etc.
- Visualization and verification of the model geometry
- Running the STAAD analysis engine to perform analysis and design
- Verification of results graphically and numerically
- Report generation and printing
- Inter-operability. In other words, using STAAD.Pro in conjunction with other programs created by REI, such as STAAD.etc.

2. Hardware Requirements

The following requirements are suggested minimums. Systems with increased capacity provide enhanced performance.

- PC with Intel-Pentium or equivalent.
- Graphics card and monitor with 1024x768 resolution, 256 color display (16 bit high color recommended).
- 128 MB RAM or higher.
- Windows NT 4.0 or higher operating system. Running it on Windows 95 & Windows 98 systems is not recommended as performance may be degraded. The program works best on Windows 2000 and XP operating systems.
- Sufficient free space on the hard disk to hold the program and data files. The disk space requirement will vary depending on the modules you are installing. A typical minimum is 500MB free space.
- A multi-media ready system with sound card and speakers is needed to run the tutorial movies and slide shows.

Note: Additional RAM, disk space, and video memory will enhance the performance of STAAD.Pro.

Starting with STAAD.Pro Version 2001, the size of structures that the program can handle has been increased significantly. As a result of this, the minimum amount of physical + virtual memory required by the program also has increased to over 600MB. Users may need to ensure that adequate amounts of virtual memory are available, and in Windows NT and 2000 systems, parameters such as paging file sizes should be large enough or span over multiple drives if the free space on any one drive runs low.

Another issue to keep in mind is the location of the "TEMP" parameter as in the "SET TEMP" environment variable in Windows NT and 2000 systems. While performing calculations, depending on the structure size, the program may create gigantic scratch files which are placed in the folder location associated with

2

the "TEMP" parameter. Users may wish to point the "SET TEMP" variable to a folder on a drive that has disk space sufficiently large to accommodate the requirements for large size structures.

Note: The user must have a basic familiarity with Microsoft Windows systems in order to use the software.

3. Contents of the STAAD.Pro CD

Typically, a startup screen appears when the CD is placed in the drive. If it does not, you may initiate it by running SPROCD.EXE located at the root folder of the CD (This can be done by clicking on the file named SPROCD.EXE from Windows Explorer). The SPROCD *Title* screen appears as shown in Figure 1.



Figure 1: The SPROCD Title Screen

The choices offered by the *Title* screen are described below:

Install STAAD Structural Suite

This is the installation module containing the programs *STAAD.Pro Version 2005, STAAD.etc, Sectionwizard* and *STAAD.foundation. STAAD.etc* is a program that enables design of structural components such as base plates, bolt groups, cantilever retaining walls, rectangular footings, etc. *Sectionwizard* is a program for calculating properties such as area, moments of

inertia, section modulii, torsional constants, etc., of various cross sections. *STAAD.foundation* is a program for designing reinforced concrete pile caps and pile groups, mat foundations, individual footings, etc. In order to use *STAAD.etc*, *Sectionwizard* and *STAAD.foundation* to their full capability, users must have purchased them as additional items of software. In the absence of a valid license to use them, those modules will work only in a demonstration mode.

The installation procedure is explained in detail in the next section.

View Readme File

Displays the contents of the README.TXT file on the CD. This file is a text file containing general information pertaining to the installation and operation of the program.

Exit

Exits the SPROCD program.

All online documentation that comes with the program is created in HTML format. These may be accessed using any Internet browser such as Microsoft's Internet Explorer or Netscape Navigator.

A set of multi-media movies which demonstrate the procedure for using STAAD.Pro are accessible after installing the program. They can be accessed from the Help menu of the main screen of the program. These too can be viewed using a web browser.

4. Installation

Please see the Readme.Txt file on the CD for latest information about installation and related issues. Also, if you receive a document titled **Installation Notes**, it will supercede all other related instructions.

Close all applications before installing STAAD.Pro. Typically, a startup screen appears when the CD is placed in the drive. If it does not, you may initiate it by running SPROCD.EXE located at the root folder of the CD (This can be done by clicking on the file named SPROCD.EXE from Windows Explorer). For an explanation of the different facilities offered by the SPROCD program, please refer to the previous section.

Note: In Windows NT, Windows 2000, and Windows XP systems, you have to log in with administrative rights before commencing installation.

To commence installation, select the option named *Install STAAD Structural Suite*. Standard installation procedure available with any software running on Microsoft Windows is followed and hence is self-explanatory.

Users who are installing the commercial version of the program will encounter the following screen.

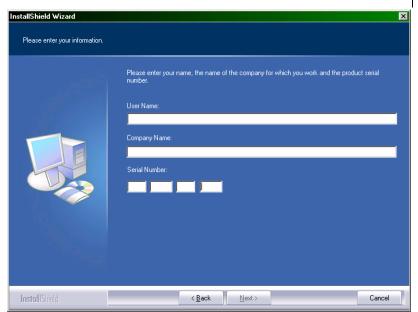


Figure 2: Customer Information & Serial Number

The serial number required for this screen should be available on the back of the CD casing. Users of the Demo Version will not be required to provide the above information.

One of the initial screens you will encounter is the one shown in Figure 3. It pertains to the type of software security system that you purchased with STAAD.Pro. A *Local Security* generally refers to a hardware lock, which is an adapter-like device that is placed on the parallel or USB port of your computer. It could also be a software based system (instead of a hardlock), in which case, it will be a software license which binds STAAD.Pro to the specific computer you are installing it on. *Network Security* refers to a system that supports simultaneous multiple-user access. A separate instruction document containing the steps for network installations is provided to users who have opted for this latter type.

7

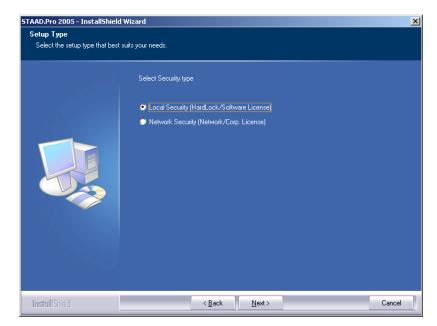


Figure 3: Selection of security system type

If you chose *Local Security*, you are asked to select the type of hardware lock supplied to you, or the software license if that is applicable. The name of the lock is engraved on the cover of the lock. Make sure the type of lock you choose from Figure 4 matches that name. This is absolutely necessary to ensure that the program functions to its full capacity.

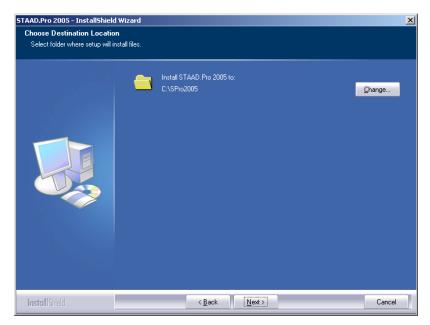
Please note that if you do not have a license for STAAD.etc, Sectionwizard and/or STAAD.foundation, they will work only in the Demonstration mode.

8



Figure 4: Selection of Local Security type

You may install the program in any folder of your choice. A default folder name is supplied to you.





The next dialog box (see next figure) seeks confirmation from you as to whether you wish to install all the programs shown in the list. *Advanced Mesher* is a standalone program for generating finite element meshes for panel type entities like walls and slabs and is available for those who want advanced meshing facilities besides those which are built into the STAAD.Pro software. *OpenSTAAD* is a library of functions which enables users to access input and output data from their STAAD.Pro projects for extraction into their own applications. While *Advanced Mesher* and *OpenSTAAD* are free utilities supplied along with STAAD.Pro, the remainder of the programs in the list require your copy-protection device to support those. If you do not wish to have any specific item(s) installed, uncheck the associated box.

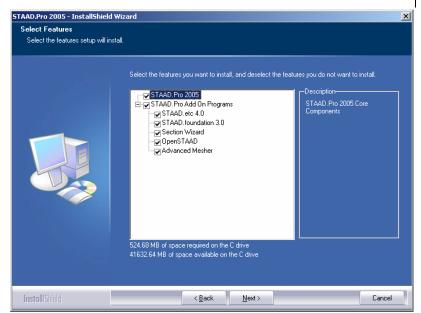


Figure 6: Selection of programs to install

You also have to choose a default unit system. This is to ensure that the length and force units frequently used by you will be available upon entry into the program each time. This is known as the base unit system, and mainly affects the units in which results are displayed, as well as default values for certain quantities. Please refer to one of the tutorials for additional information on these. Of course, it is always possible for you to change the base unit system within the program, at run-time, as frequently as you please. 11

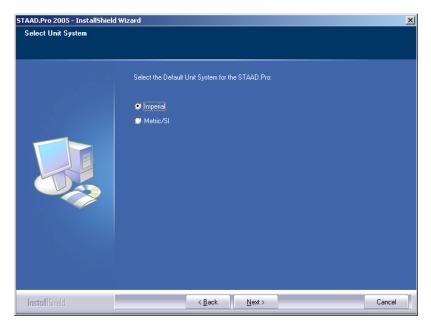


Figure 7: Selection of Default Unit System for STAAD.Pro

Towards the end of the installation process, a message resembling the one shown in Figure 7 will appear. It is pertinent only to users who have received this program as an upgrade from earlier versions of STAAD.Pro, and are already using a security device with those versions. For those users, their hardware lock also needs to be upgraded to enable it to work with STAAD.Pro 2004. That process is done electronically - called re-programming the lock - and there is no need to physically replace the lock (in most of the cases).



Figure 8: Information regarding upgrade of lock

After the installation is complete, please restart your machine for the changes to take effect.

5. Copy Protection Device

As explained in the previous section, a copy protection device in the form of a hardware lock, or a software license, is required to run STAAD.Pro, STAAD.etc and Sectionwizard. The hardware lock must be inserted in the parallel port of your computer and must remain there during the entire duration that you are in one of the programs. If any other device, such as printer cable, hardware lock for other software, etc., is attached to the parallel port, we recommend that you attach the STAAD.Pro / STAAD.etc hardware lock in front of such devices. In case you have multiple locks, and cannot stack them for any reason, REI can replace your parallel port type with a USB type of lock.

The hardware lock is configured for the programs and modules that you have purchased. If you install one of the programs or modules that is not supported by the hardware lock, that component may not be accessible, or will be operable only as a Demonstration version.

The hardware lock driver(s) are automatically installed during the installation process. For computers running on Windows NT, Windows 2000, or Windows XP, you must have administrative rights before installing the program to enable proper installation of the hardware lock driver files.

As can be seen from the tutorials in the later sections of this book, STAAD.Pro consists of various modules, each designed to perform a certain type of task in the model generation, analysis and result verification process. Version 2005 requires the hardlock to be in place during the entire time that any and all of these tasks are being performed.

In other words, from the moment you start the program till the moment you exit it, the lock has to be in place. If the lock is detached at any time in between, the program will stop running, and request that you re-attach the lock. In the event that you are unable to, it will provide the opportunity to save the work and exit the program. To resume your work, you will have to put the lock back in the port and re-start the program.

Another important aspect to note is that if you are upgrading from an earlier version of STAAD.Pro such as 2000, 2001 or 2002, one of the following is applicable with regards to the lock:

- a. The upgrade package should contain a new lock which replaces your old lock.
- b. The upgrade package should contain information outlining how you can re-program your earlier lock so that it becomes compatible with STAAD.Pro 2005.

6. Running STAAD.Pro

Click on the STAAD.Pro icon from the STAAD.Pro 2004 program group.

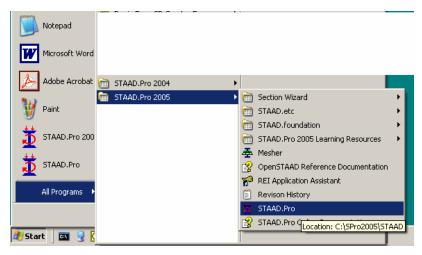


Figure 9: Starting STAAD.Pro

The STAAD.Pro main screen appears as shown in below.	

STAAD.Pro			_ 8 ×
		I m a all on all the parts for	· ᄣᆞ 《]] 코윙성 위] 이 코] 프 램 북
		25 (L &)	× ?
インドドキャロ国際は日下の日	永父生の黒ン をよし	10 A 76 9 4 9 8 8 1	
Pito			
			-
			×
22	New File Recent Files		1
19 19	Space	File Name :	
<u>ک</u> ست	Plane Floor	Structure1	
nem Dala	Truss	Location :	
NEV .		C:\test\	
*	J		
<u>a</u>	A SPACE structure, which is a three-dimen any plane, is the most general.	sional framed structure with loads applied in	
	any pare, is the most garante		
E	Length Units C Inch C Decimeter	Force Units C Pound C Newton	
唐 [1]	Foot C Meter	KiloPound C DecaNewton	
	C Millimeter C Kilometer	C Kilogram C KiloNewton	
	C Centimeter	C Metric Ton C MegaNewton	
16 I			
· · · · · · · · · · · · · · · · · · ·			-
	Display this dialog box at the Startup		
	< Back.	Next> Cancel Help	
-			_

Figure 10: The STAAD.Pro screen

If you are a first time user who is unfamiliar with STAAD.Pro, we suggest that you go through the tutorials shown in Section II of this manual.

7. Running STAAD.etc

To launch the STAAD.etc program, click on the STAAD.etc icon.

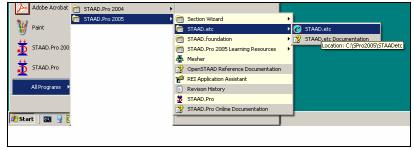


Figure 11: Starting STAAD.etc

For help on using this program, we suggest that you go through the *STAAD.etc Documentation* accessible by clicking on its icon shown in the above figure.

8. Running Sectionwizard

To launch Section*wizard*, choose one of the programs from the Section*wizard* menu.

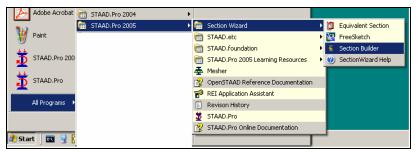


Figure 12: Starting Setionwizard

For help on using this program, please go through *Sectionwizard Help* shown in the above figure.

9. Running STAAD.foundation

To launch STAAD.foundation, click on the STAAD.foundation icon.

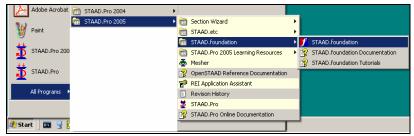


Figure 13: Starting STAAD.foundation

For help on using this program, please go through the *STAAD.foundation Documentation* shown in the above figure.

10. Running Mesher

To launch Mesher, click on the Mesher icon.

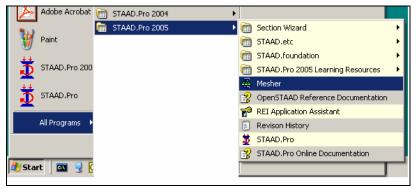


Figure 14: Starting Mesher

Information on using this program is available from the Help menus of the program.

N o t e s

<u>Part - II</u>

Tutorials

Table of Contents

Introduction						
1.	. Tutorial Problem 1: 2D Portal Frame					
	1.1 1.2 1.3 1.4 1.5	Descrip Starting Creating	s of creating the model tion of the Tutorial Problem the Program g a New Structure g the Model using the Graphical Interface	1-2 1-3 1-5 1-10 1-13		
		$\begin{array}{c} 1.5.1 \\ 1.5.2 \\ 1.5.3 \\ 1.5.4 \\ 1.5.5 \\ 1.5.6 \\ 1.5.7 \\ 1.5.8 \\ 1.5.9 \\ 1.5.10 \\ 1.5.11 \\ 1.5.12 \\ 1.5.13 \\ 1.5.14 \\ 1.5.15 \\ 1.5.16 \end{array}$	Generating the Model Geometry Switching On Node And Beam Labels Specifying Member Properties Specifying Material Constants Changing the Input Units of Length Specifying Member Offsets Printing Member Information in the Output File Specifying Supports Viewing the model in 3D Specifying Loads Specifying the Analysis Type Specifying Post-Analysis Print Commands Short-listing the Load Cases to be used in Steel Design Specifying Steel Design Parameters Re-specifying the Analysis Command Re-specifying the Track Parameter	$1-16 \\ 1-23 \\ 1-26 \\ 1-32 \\ 1-33 \\ 1-35 \\ 1-40 \\ 1-43 \\ 1-48 \\ 1-50 \\ 1-60 \\ 1-62 \\ 1-66 \\ 1-68 \\ 1-74 \\ 1-75 \\ 1-75 \\ 1-75 \\ 1-75 \\ 1-75 \\ 1-76 \\ 1-75 \\ 1-75 \\ 1-76 \\ 1-75 \\ 1-75 \\ 1-75 \\ 1-76 \\ 1-75 \\ $		
	1.6 1.7 1.8 1.9 1.10	Creatin Perform Viewin	Specifying the Check Code Command g the Input Command File g the Model using the Command File ning Analysis/Design g the Output File rocessing Going to the Post-Processing Mode Annotating the Displacements Displaying Force/Moment Diagrams Annotating the Force/Moment Diagram Changing the Degree of Freedom for which Forces Diagram is Plotted	1-76 1-79 1-82 1-91 1-94 1-102 1-103 1-106 1-111 1-114 1-117		
1.10.6 Displaying the Dimensions of The Members						

2.	Tuto	orial Pi	roblem	2: R(C Fr	amed	S	tructi	ire			2-1
	2.1 2.2 2.3 2.4 2.5 2.6	Descrip Starting Creating Element	s of creat tion of the the Prog g a New S ts of the S g the STA	e Tutoria ram Structure STAAD.	al Pro e Pro S	blem creen						2-2 2-3 2-6 2-11 2-14 2-15
		2.6.1 2.6.2 2.6.3 2.6.4 2.6.5 2.6.6 2.6.7 2.6.8 2.6.9 2.6.10 2.6.11	Generati Changin Specifyi Specifyi Specifyi Specifyi Specifyi Short-lis Specifyi Specifyi	g the Inj ng Mem ng Geor ng Mate ng Supp ng Load ng the A ting the ng Conc	put U ber P netric rial C orts s nalys load erete I	nits of I ropertie Constant onstant sis Type cases to Design I	Len es ants ts e o be Para	gth used i		ete Desi	ign	2-16 2-28 2-30 2-36 2-38 2-41 2-46 2-63 2-65 2-67 2-71
	2.7 2.8 2.9 2.10 2.11	Creating Perform	g the Inpu g the Mod ing the A g the Outp occessing	lel using nalysis	the C	Comma	nd I	File				2-74 2-77 2-85 2-88 2-96
			Going to Viewing Switchir deflectio Changin Annotati Changin annotate The Noc Displayi Switchir Force/M Changin is plottee	the Def ng betwee on diagra g the siz ng Disp g the un ed le Displa ng Force ing betwee foment c g the siz g the de	lectio een los am ze of t lacem its in aceme e/Mon een los liagra ze of t	n Diag ad cases he defle nents which o ent Tabl ment Di ad cases m he Forc	ram s fo ecti- disp le iagr s fo ce/N	n on diag olacemo cams or viewi Aoment	gram ent value ing the t diagram	n	ram	2-97 2-99 2-101 2-105 2-108 2-111 2-114 2-119 2-122 2-126 2-129
		2.11.12	Annotati		Force/	Momer	nt di	iagram				2-129

3.	Tut	2.11.14 2.11.15 2.11.16 2.11.17 2.11.18 2.11.19 2.11.20	Changing the units in which Force/Moment values are annotated Beam Forces Table Viewing the Force/Moment diagrams from the Beam Graphs Page Restricting the load cases for which results are viewed Using Member Query Producing an on-screen Report Taking Pictures Creating Customized Reports	2-134 2-137 2-141 2-145 2-147 2-152 2-155 2-157 3-1
	3.1 3.2 3.3 3.4 3.5 3.6	Methods Descript Starting Creating Element	s of creating the model tion of the tutorial problem the program g a new structure ss of the STAAD.Pro screen g the STAAD.Pro model	3-2 3-3 3-6 3-11 3-14 3-15
		3.6.1 3.6.2 3.6.3 3.6.4 3.6.5 3.6.6 3.6.7 3.6.8 3.6.9	Generating the model geometry Changing the input units of length Specifying Element Properties Specifying Material Constants Specifying Supports Specifying Primary Load Cases Creating Load Combinations Specifying the analysis type Specifying post-analysis print commands	3-16 3-53 3-55 3-61 3-62 3-67 3-77 3-82 3-85
		Creating Performi	the input command file the model using the command file ng the analysis and design the output file cessing	3-89 3-91 3-98 3-101 3-109
		3.11.1 3.11.2 3.11.3 3.11.4 3.11.5 3.11.6 3.11.7	Viewing stress values in a tabular form Printing the tables Changing the units of values which appear in the above tables Limiting the load cases for which the results are displayed Stress Contours Animating stress contours Creating AVI Files	3-110 3-112 3-113 3-115 3-117 3-123 3-124

3.11.8Viewing plate results using element query3-1273.11.9Producing an onscreen report3-1313.11.10Viewing Support Reactions3-136

4. Tutorial Problem 4: Interoperability
(using STAAD.Pro and STAAD.etc)4-1

4.1	Understanding STAAD.etc	4-2
4.2	Description of the Tutorial Problem	4-3
4.3	Using the Interactive Mode in STAAD.Pro	4-4
4.4	Designing a Footing based on results from STAAD.Pro	4-9
4.5	Designing a Base Plate based on results from STAAD.Pro	4-15
4.7	Saving the Interactive Design as a STAAD.etc File	4-16

5.	Fre	equently Performed Tasks	FPT-1
	1	Selecting nodes, beams, plates, etc.	FPT-1
	2	Viewing the structure from different angles	FPT-8
	3	Switching on labels for nodes, beams, plates, etc.	FPT-12
	4	Displaying a portion of the model by isolating it	
		from the rest of the structure	FPT-18
	5	Creating Groups	FPT-38
	6	Displaying Loads on the screen	FPT-47
	7	Displaying Load Values on the screen	FPT-52
	8	Structural Tool Tip Options	FPT-58
	9	Identifying Beam Start and End	FPT-62
	10	Plotting from STAAD.Pro	FPT-67

Introduction

STAAD.Pro is a general purpose program for performing the analysis and design of a wide variety of types of structures. The basic three activities which are to be carried out to achieve that goal - a) model generation b) the calculations to obtain the analytical results c) result verification - are all facilitated by tools contained in the program's graphical environment. This manual contains four sample tutorials which guide the user through those 3 activities.

The first of those tutorials demonstrates these processes using a simple two-dimensional steel portal frame. It is a good starting point for learning the program. If you are unfamiliar with STAAD.Pro, you will greatly benefit by going through this tutorial first.

For the second tutorial, we have chosen a reinforced concrete frame. We generate the model, perform the analysis, and design the concrete beams and columns. It contains extensive details on the various facilities available for visualization and verification of results.

The modelling and analysis of a slab is demonstrated in the third tutorial. Slabs, and other surface entities like walls are modelled using plate elements. Large surface entities may have to be defined using several elements and this sometimes requires a tool called a mesh generator. This tutorial shows the simple techniques as well as the mesh generation method for generating the finite element model of the slab. It also explains the methods by which one can check the results for plate elements.

A tutorial which demonstrates the inter-operability features between STAAD.Pro and STAAD.etc. is presented in the fourth tutorial. STAAD.etc is a set of modules which can be used to perform component designs such as for a rectangular footing, base plate, cantilever retaining wall, moment connection, bolt group, etc. Users who have purchased STAAD.etc in addition to STAAD.Pro may go through this tutorial to familiarize themselves with the process of utilizing STAAD.etc to perform secondary analysis and design tasks on a structure for which the primary analysis and design is done using STAAD.Pro.

Tutorial Problem 1: 2D Portal Frame

This chapter provides a step-by-step tutorial for creating a 2D portal frame using STAAD.Pro. This tutorial covers the following topics.

- Starting the Program
- Creating a New Structure
- Creating Joints and Members
- Switching On Node and Beam Labels
- Specifying Member Properties
- Specifying Material Constants
- Specifying Member Offsets
- Printing Member Information
- Specifying Supports
- Specifying Loads
- Specifying the Analysis Type
- Specifying Post-Analysis Print Commands
- Specifying Steel Design Parameters
- Performing Analysis and Design
- Viewing the Output File
- Verifying results on screen both graphically and numerically

Section 1

1.1 Methods of creating the model

1-2 Tutorial 1

There are two methods of creating the structure data:

- a. using the command file
- b. using the graphical model generation mode, or graphical user interface (GUI) as it is usually referred to.

The Command File is a text file which contains the data for the structure being modeled. This file consists of simple Englishlanguage like commands. This command file may be created directly using the editor built into the program, or for that matter, any editor which saves data in text form, such as Notepad or WordPad available in Microsoft Windows.

This command file is also automatically created behind the scenes when the structure is generated using the Graphical User Interface. The graphical model generation mode and the command file are seamlessly integrated. So, at any time, you may temporarily exit the graphical model generation mode and access the command file. You will find that it reflects all data entered through the graphical model generation mode. Further, when you make changes to the command file and save it, the GUI immediately reflects the changes made to the structure through the command file.

Both methods of creating our model are explained in this tutorial. Section 1.3 through 1.6 explain the procedure for creating the file using the GUI. Section 1.7 describes creation of the command file using the STAAD.Pro text editor.

1.2 Description of the tutorial problem

The structure for this project is a single bay, single story steel portal frame that will be analyzed and designed. The figure below shows the structure.

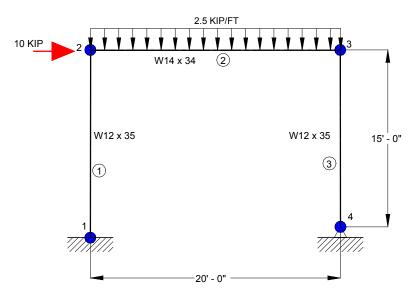


Figure 1.1

An input file called "Tut-01-portal.std" containing the input data for the above structure has been provided with the program. This file contains what would otherwise have resulted had we followed the procedure explained in Section 1.7. 1-4 Tutorial 1

BASIC DATA FOR THE STRUCTURE

ATTRIBUTE	DATA
Member properties	Members 1 & 3 : W12X35
	Member 2 : W14X34
Material Constants	Modulus of Elasticity : 29000 ksi
	Poisson's Ratio : 0.30
Member Offsets	6.0 inches along global X for member 2 at both ends
Supports	Node 1 : Fixed
	Node 4 : Pinned
Loads	Load case 1 : Dead + Live Beam 2 : 2.5 kips/ft downward along global Y
	Load case 2 : Wind From Left 10 kips point force at Node 2
	Load case 3 : 75 Percent of (DL+LL+WL) Load Combination - L1 X 0.75 + L2 X 0.75
Analysis Type	Linear Elastic (PERFORM)
Steel Design	Consider load cases 1 and 3 only.
	Parameters: Unsupported length of compression flange for bending : 10 ft for members 2 and 3, 15 ft for member 1.
	Steel Yield Stress : 40 ksi
	Perform member selection for members 2 and 3

1.3 Starting the program

Select the **STAAD.Pro** icon from the *STAAD.Pro 2005* program group.

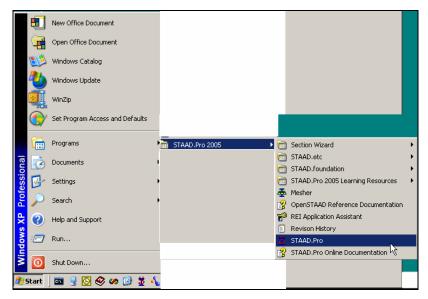


Figure 1.2

1-6 Tutorial 1

The STAAD.Pro Graphical Environment will be invoked and the following screen comes up.

3	STAAD.Pro	_ 8 ×
	s Help	
È	≝⋹⋹⋓⋼⋟⋹⋋⋺⋼⋳⋼⋈⋗∭⋳⋸⋸⋹⋼⋳⋟⋼∭⋳⋼⋼⋼⋳⋼⋼⋫⋟∊⋇∊	麗世
	₽₽₽₽₽₽₽₽++++><\$< E <c<0000000000000000000000000000000000< th=""><th></th></c<0000000000000000000000000000000000<>	
1	▲●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●	
医脊髓髓髓炎大压体炎 中国政府有利的 医尿道病	New File [Receive Face] Shoot Place Place Trans Locdion: Cutort ASPACE disclave, which is a three dimensional framed structure with locads applied in explained. ASPACE disclave, which is a three dimensional framed structure with locads applied in explained. Cutort Image: Continenter Food Matherian Kikopan Matherian Kikopan Matherian Kikopan Matherian Kikopan Matherian Kikopan Matherian Kikopan Matherian Matherian<	

Figure 1.3

This New dialog box will come up every time we start the program. To turn this feature off, simply uncheck the Display this dialog box at the Startup box at the lower left hand corner. This feature can be turned on again at a later time when $File \mid New$ is invoked from the main menu.

Note about the unit system :

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, Aluminum – selected from the program's library (Please refer to Section 5 of the STAAD.Pro 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.

The place from where we can change this setting is under the *File* | *Configure* menu. To get to that option, first close down the dialog box shown in the earlier figure by clicking on **Cancel**. Then, click on the **File** | **Configure** menu option (see figure below) and choose the appropriate unit system you want. For this tutorial, let us choose the **English** units (Kip, Feet, etc.).

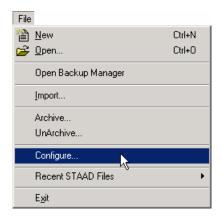


Figure 1.4

1-8 Tutorial 1

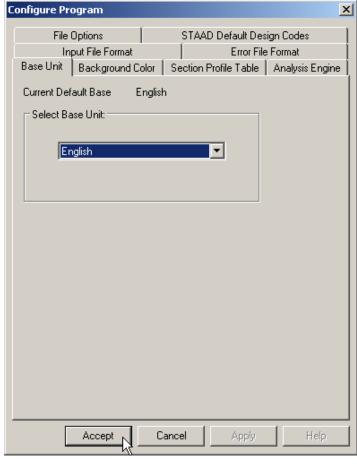


Figure 1.5

Click on the Accept button to close the above dialog box.

Following this, select File | New once again.

File	
웥 <u>N</u> ew	Ctrl+N
🚰 <u>O</u> pen 씨	Ctrl+O
Open Backup Manager	
Import	
Archive	
UnArchive	
Configure	
Recent STAAD Files	•
E <u>x</u> it	

Figure 1.6

The dialog box shown in Figure 1.3 will re-appear.

1-10 | Tutorial 1 **1.4 Creating a new structure**

1. In the *New* dialog box, we provide some crucial initial data necessary for building the model.

The structure type is to be defined by choosing from among *Space*, *Plane*, *Floor* and *Truss*. A *Space* type is one where the structure, the loading or both, cause the structure to deform in all 3 global axes (X, Y and Z). In a *Plane* type, the geometry, loading and deformation are restricted to the global X-Y plane only. A *Floor* type is a structure whose geometry is confined to the X-Z plane. A *Truss* type of structure carries loading by pure axial action. Truss members are deemed incapable of carrying shear, bending and torsion. For our model, let us choose **Plane**.

We choose **Foot** as the length unit and **Kilo Pound** as the force unit in which we will start to build the model. The units can be changed later if necessary, at any stage of the model creation.

We also need to provide a name in the *File Name* edit box. This is the name under which the structure data will be saved on the computer hard disk. The name "Structure?" (? will be a number) is recommended by the program by default, but we can change it to any name we want. Let us choose the name **PORTAL**.

A default path name - the location on the computer drive where the file will be saved – is provided by the program under *Location*. If you wish to save the file in a different location, type in the name, or click the $\boxed{}$ button and specify the desired path.

After specifying the above input, click on the Next button.

				Tutorial 1	1-
ew				×	
Space		File Name :			
Plane ☐ Floor		PORTAL		_	
		Location :			
		C:\test\	<u>·</u>	••	
I					
A PLANE structure is bo	und by a global X-Y (coordinate system wi	ith loads in the same	-	
plane.					
Length Units	C. D. 1	Force Units	C Newton	7	
	C Decimeter	 Found KiloPound 	O DecaNewton		
	OMeter Kilometer	C Kilogram	C KiloNewton		
C Centimeter	Niometer	C Metric Ton	C MegaNewton		
Centimeter		• Metic For	 Meganewion 		
Display this dialog box at					
	< Back	Next>	Cancel H	elp	
		. 0			

Figure 1.7

2. In the next dialog box, we choose the tools to be used to initially construct the model. *Add Beam, Add Plate* or *Add Solid* are, respectively, the starting points for constructing beams, plates or solids. *Open Structure Wizard* provides access to a library of structural templates which the program comes equipped with. Those template models can be extracted and modified parametrically to arrive at our model geometry or some of its parts. If the model is to be created initially using the STAAD command language, the *Open STAAD Editor* box can take us to the STAAD editor. Please remember that all these options are also available from the menus and dialog boxes of the GUI, even after we dismiss this dialog box.

Note: If you wish to use the Editor to create the model, choose Open STAAD Editor, click Finish, and proceed to Section 1.7.

1-12 Tutorial 1

For our model, let us check the **Add Beam** option. Click on the **Finish** button. The dialog box will be dismissed and the STAAD.Pro graphical environment will be displayed.

Where do you want to go ?	×
Add Beam Add Plate Add Solid Open Structure Wizard Open STAAD Editor Edit Job Information	
Begin building your model by creating new joints and beams using the drawing tools and spreadsheets.	e construction grid,
✓ Display this dialog box at the Startup < Back	Cancel Help

Figure 1.8

1.5 Creating the model using the graphical user interface

In order to generate the model graphically, we have to familiarize ourselves with the components of the STAAD.Pro screen. A sample of the STAAD.Pro screen is shown in Figure 1.9.

The screen has five major elements as described below:

Menu bar

Located at the top of the screen, the Menu bar gives access to all the facilities of STAAD.Pro.

Toolbar

The dockable Toolbar gives access to the most frequently used commands. You may also create your own customized toolbar.

Main Window

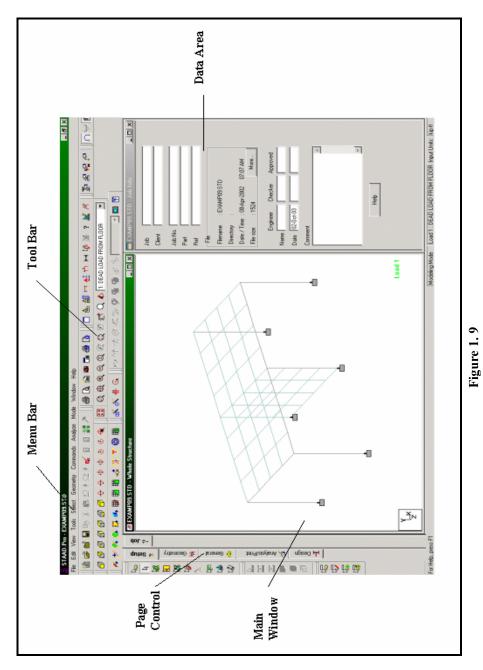
This is the largest area at the center of the screen, where the model drawings and results are displayed in pictorial form.

Page Control

The *Page Control* is a set of tabs that appear on the left-most part of the screen. Each tab on the *Page Control* allows you to perform specific tasks. The organization of the Pages, from top to bottom, represents the logical sequence of operations, such as, definition of beams, specification of member properties, loading, and so on.

Each tab has a name and an icon for easy identification. The name on the tabs may or may not appear depending on your screen resolution and the size of the STAAD.Pro window. However, the icons on the *Page Control* tabs always appear.

The Pages in the *Page Control* area depend on the *Mode* of operation. The *Mode* of operation may be set from the *Mode* menu from the Menu bar.



Elements of the STAAD.Pro Screen

Data Area

The right side of the screen is called the *Data Area*, where different dialog boxes, tables, list boxes, etc. appear depending on the type of operation you are performing. For example, when you select the *Geometry* | *Beam* Page, the Data Area contains the Node-Coordinate table and the Member-incidence table. When you are in the *Load* Page, the contents of the Data Area changes to display the currently assigned Load cases and the icons for different types of loads.

The icons in the toolbar as well as in the *Page Control* area offer ToolTip help. As we move the mouse pointer over a button, the name of the button – called a ToolTip – appears above or below the button. This floating Tool tip help will identify the icon. A brief description of the icon also appears in the status bar.

We are now ready to start building the model geometry. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

1.5.1 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc. From the standpoint of the STAAD command file, the commands to be generated for the structure shown in section 1.2 are :

```
JOINT COORDINATES
1 0. 0.; 2 0. 15.; 3 20. 15.; 4 20. 0.
MEMBER INCIDENCE
1 1 2; 2 2 3; 3 3 4
```

Steps:

1. We selected the *Add Beam* option earlier to facilitate adding beams to create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X,Y,Z) are represented in the icon in the lower left hand corner of the drawing area.

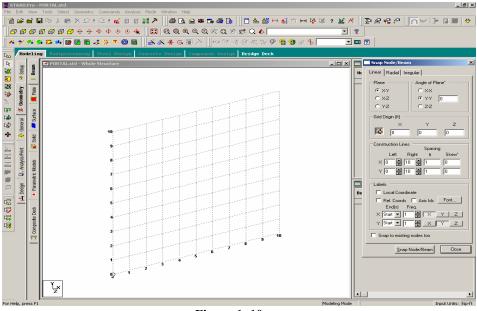


Figure 1. 10

2. A Snap Node/Beam dialog box also appears in the data area on the right side of the screen. The Linear tab is meant for placing the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board. The Radial tab enables construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modelled as piece-wise linear straight line segments. The Irregular tab can be used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane. We will use the Linear tab.

In our structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. So, in this dialog box, let us keep X-Y as the *Plane* of the grid. The size of the model that can be drawn at any time is controlled by the number of *Construction Lines* to the left and right of the origin of axes, and the *Spacing* between adjacent construction lines. By setting 20 as the number of lines to the right of the origin along X, 15 above the origin along Y, and a spacing of 1 feet between lines along both X and Y (see next figure) we can draw a frame 20ft X 15ft, adequate for our structure. Please note that these settings are only a starting grid setting, to enable us to start drawing the structure, and they do not restrict our overall model to those limits.

1-18 Tutorial 1

📲 Snap Node/Bean	n 🛛 🗶				
Linear Radial Irreg	jular				
Plane	Angle of Plane*				
© X-Y	O XX				
⊙ x-z	• Y-Y 0				
O Y-Z	O Z-Z				
Grid Origin (ft)					
×	Y Z				
	0 0				
Construction Lines-					
Left Bi	Spacing ight ft Skew*				
× 0 🖨 20					
Y 0 15 1 0					
Labels					
Local Coordina					
End(s) Freq.					
X Start V 1 X Y Z					
Y Start ▼ 1 ♥ X Y Z					
Snap to existing nodes too					
<u>S</u> nap N	ode/Beam Close				

Figure 1. 11

3. To start creating the nodes, let us first activate the **Snap Node/Beam** button by clicking on it. Then, with the help of the mouse, click at the origin (0, 0) to create the first node.

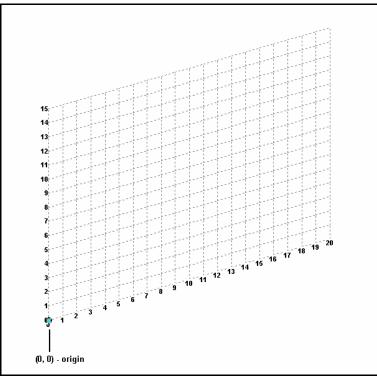


Figure 1.12

4. In a similar fashion, click on the following points to create nodes and automatically join successive nodes by beam members.

(0, 15), (20, 15), and (20, 0)

The exact location of the mouse arrow can be monitored on the status bar located at the bottom of the window where the X, Y, and Z coordinates of the current cursor position are continuously updated.

1-20 Tutorial 1

When steps 1 to 4 are completed, the structure will be displayed in the drawing area as shown below.

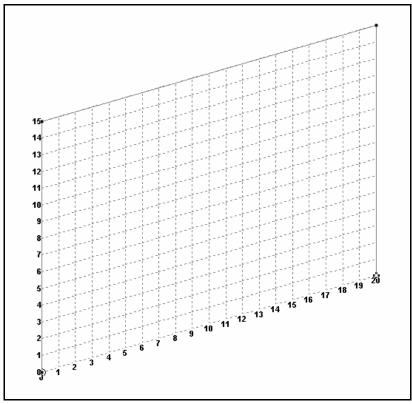


Figure 1. 13

5. At this point, let us remove the grid from the structure. To do that, click on the **Close** button in the *Snap Node/Beam* dialog box.

🗱 Snap Node/Beam 🛛 🗙	1
Linear Radial Irregular	L
Plane Angle of Plane*	L
● X-Y	L
Oxz ⊙yy o	L
O YZ O ZZ	L
Grid Origin (ft)	L
X Y Z	L
	L
Construction Lines	L
Spacing Left Right ft Skew*	L
	L
Y 0 15 1 0	L
	L
Labels	L
Local Coordinate	L
Rel. Coords Axis Ids Font End(s) Freq.	
X Start V 1 X Y Z	L
Y Start ▼ 1 ♥ X Y Z	L
	L
Snap to existing nodes too	
Snap Node/Beam Close	\succ

Figure 1. 14

The grid will now be removed and the structure in the main window should resemble the figure shown below.

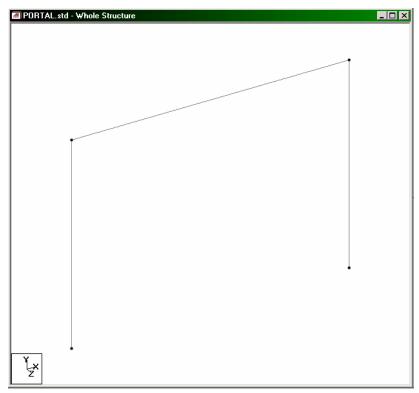


Figure 1.15

It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events. To save the file, pull down the **File** menu and select the **Save** command.

1.5.2 Switching on node and beam labels

1. Node and beam labels are a way of identifying the entities we have drawn on the screen. In order to display the *node* and *beam numbers*, right click anywhere in the drawing area. In the pop-up menu that comes up, choose **Labels**. Alternatively, one may access this option by selecting the *View* menu followed by the *Structure Diagrams* option from the top menu bar, and the *Labels* tab of the dialog box that comes up.

View			
Zoom	•		
🛫 Pan			
View Selected Objects Only			
Always <u>F</u> it in Current Window			
Tables		X Cut	Ctrl+X
🔣 Whole Structure		Ba Copy	Ctrl+C
Ogientation	F4	🔒 Easte	
🔁 Structure Diagrams 🔪		Selec	t Cursor
		Selec	tion Mode
Open View		🝅 Ta <u>k</u> e	Picture
New View			
View <u>M</u> anagement	•	🗡 Add <u>B</u>	-
Toolbars		<u> </u>	·S
Option <u>s</u>		<u>O</u> pen	View
Set Structure <u>C</u> olors		<u>L</u> abel	s
_		Orient	tation F4
Structural Tool Tip Options		🔁 Struct	ture <u>D</u> iagrams
📣 3D Rendering		💡 Mode	l View Details
Refresh	F5	🔥 3D Re	endering

Figure 1.16

2. In the *Diagrams* dialog box that appears, turn the **Node Numbers** and **Beam Numbers** on and then click on **OK**.

Diagram s		X					
Force Limits Structure	Animation	Design Results					
Nodes Node Numbers (N) Node Points (K) Supports (S) Dimension (D)	Properties C References (J) C Sections (X) None	² hysical Members Member Number (^M) Design Group (^G) Design Envelope (^E) Design Brief (^B)					
Beams Beam Numbers (B) Beam Orientation (O) Beam Spec (A) Releases (R)	Loading Display Options Load Values (V) Display Floor Loading (I Display Floor Load Dist Display Wind Load Cor Display Wind Load (W)	ribution (Y) htributory Area (H)					
Beam Ends (E) Start Color End Color Plates	General Show Axes Window (^A Show Axes At Org (I) Material (M) Show Diagram Info (G)) 🧖 Master Slave (L)					
 Plate Numbers (P) Plate Orientation (T) 	Surfaces Surface Numbers (^P)	Surface Orientation (^T)					
Solids Solid Numbers (C)	 Always Use STAAD.Pri Always Use Current Lal ode (Use labeling cursors to 0) 	bel Settings.					
	For quick access to the labels using keyboard hotkeys, press Shift + the letter shown in brackets.						
	OK Cancel	Apply Help					

Figure 1.17

The following figure illustrates the node and beam numbers displayed on the structure. The structure in the main window should resemble the figure shown below.

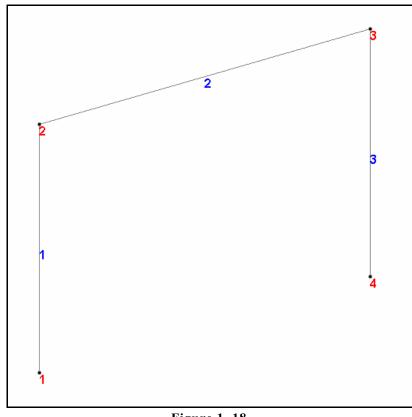


Figure 1. 18

If you are feeling adventurous, here is a small exercise for you. Change the font of the node/beam labels by going to the *View* menu and selecting the *Options* command, and then selecting the appropriate tab (Node Labels / Beam labels) from the *Options* dialog box.

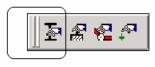
1.5.3 Specifying member properties

Our next task is to assign cross section properties for the beams and columns (see figure in section 1.2). For those of us curious to know the equivalent commands in the STAAD command file, they are :

MEMBER PROPERTY AMERICAN 1 3 TABLE ST W12X35 2 TABLE ST W14X34

Steps:

1. To define member properties, click on the **Property Page** icon located on the top toolbar.



Property Page

Figure 1. 19

Tutorial 1 1-27

Alternatively, one may go to the *General* | *Property* page from the left side of the screen as shown below.

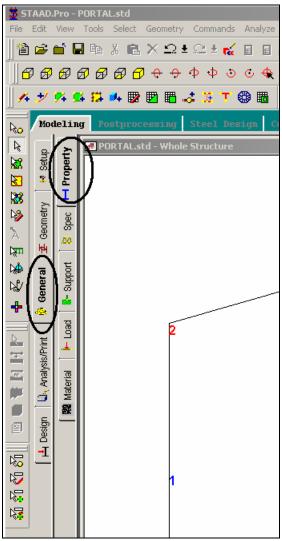


Figure 1. 20

1-28 Tutorial 1

2. In either case, the *Properties* dialog box comes up (see figure below). The property type we wish to create is the W shape from the AISC table. This is available under the **Section Database** button in the *Properties* dialog box as shown below. So, let us click on the **Section Database** button.

📽 Properties - Wh	ole Structure	×				
Section Beta Angle						
Ref Section	Material					
I						
Highlight Assigned	d Geometry					
	E dit	Delete				
Values 🕻	Section Database	Define				
Materials	Thickness	User Table				
	Assignment Method					
C Assign To Selected Beams © Use Cursor To Assign C Assign To Edit List C Assign To View						
Assign To Edit List S Assign To View						
	Assign Clos	e Help				

Figure 1. 21

3. In the Section Profile Tables dialog box that comes up, select W Shape under the American option. Notice that the Material box is <u>checked</u>. Let us keep it that way because it will enable us to subsequently assign the material constants E, Density, Poisson, etc. along with the cross-section since we want to assign the default values.

Choose **W12X35** as the beam size, and **ST** as the section type. Then, click on the **Add** button as shown in the figure below. Detailed explanation of the terms such as ST, T, CM, TC, BC, etc. is available in Section 5 of the STAAD Technical Reference Manual.

Section Profile Tables				×
Steel Timber Aluminum				
🦪 American 📃	W Shape			
W Shape M Shape S Shape B Shape Channel Channel MC Channel MC Channel MC Channel MSS Rectangle HSS Rectangle HSS Rectangle MSS Rectangle Mareican Steel Joist American Cold Formec Multer Cold Formed	Select Beam ₩12×22 ₩12×26 ₩12×30 ₩12×30 ₩12×35 ₩12×45 ₩12×50 ₩12×50 ₩12×55 ₩12×55 ¥12×55 ¥12×55 ¥12×55 ¥12×55 ¥12×55 ¥12×55 ¥12×55 ¥12×55	Type Specification ST (Single Section from Table) T (Tee Section Cut from Beam) CM (Composite Section) CT (Concrete Thickness) FC (Concrete Grade) CW (Concrete Width) CD (Density of concrete) CT (To Cover Plate) BC (Bottom Cover Plate) TB (Top & Bottom Cover Plate) WP (Cover Plate Width)	ft kip/ft2 ft kip/ft3 ft	
		TH (Cover Plate Thickness)	ft	
		Add Assign	Close	Help

Figure 1. 22

4. To create the second member property (ST W14X34), select the W14X34 shape and click on the Add button.

After the member properties have been created, let us **Close** the *Section Profile Tables* dialog box.

1-30 Tutorial 1

- 5. The next step is to associate the properties we just created with selected members in our model. Follow these steps.
 - a. Select the first property reference in the *Properties* dialog box (W12X35).

I

- b. Make sure that the "Use Cursor to Assign" button is selected under the Assignment Method box.
- c. Click on the Assign button. The cursor changes to
- d. Using the cursor, click on members 1 and 3.
- e. Finally, click on the Assign button again, or click on the 'Esc' button on your keyboard to stop the assignment process.

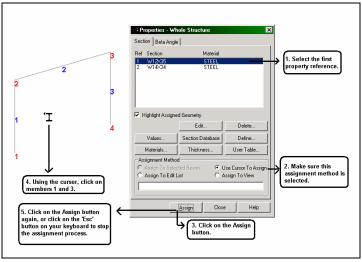
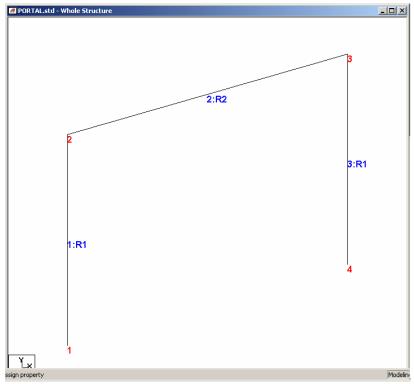


Figure 1.23

6. In a similar fashion, assign the second property reference (W14X34) to member 2.



After both the properties have been assigned to the respective members, our model should resemble the following figure.

Figure 1.24

Let us once again save our structure by pulling down the **File** menu and selecting the **Save** command.

1.5.4 Specifying material constants

In Section 1.5.3, we kept the *Material* check box "on" while assigning the member properties. Consequently, the material constants got assigned to the members along with the properties, and the following commands were generated in the command file:

CONSTANTS E 29000 MEMB 1 TO 3 POISSON 0.3 MEMB 1 TO 3 DENSITY 0.000283 MEMB 1 TO 3 ALPHA 6.5e-006 MEMB 1 TO 3

Hence, there is no more a need to assign the constants separately. However, if we hadn't assign them as before, we could go to the menu option *Commands* | *Material Constants* and assign them explicitly as shown in the figure below.

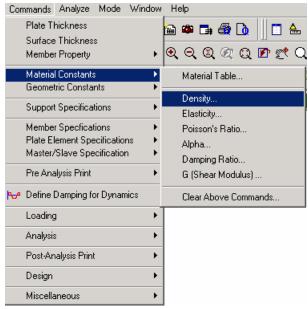


Figure 1.25

1.5.5 Changing the input units of length

For specifying member offset values, as a matter of convenience, it is simpler if our *length units* are *inches* instead of *feet*. The commands to be generated are:

UNIT INCHES KIP

Steps:

1. To change the length units from feet to inch, click on the **Input Units** icon from the appropriate toolbar.



Figure 1. 26

Alternatively, one may select the *Tools* | *Set Current Input Unit* menu option as shown in the next figure.

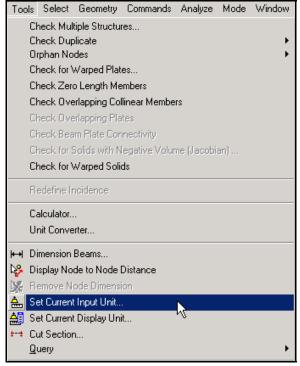


Figure 1.27

2. In either case, the following dialog box comes up. Set the *Length Units* to **Inch** and click on the **OK** button.

Set Current Inp	ut Units			>	×
Length Units-		Force U	nits ——		
Inch	C Decimeter	O Pou	und	C Newton	
C Foot	C Meter	• Kilo	Pound	C DecaNewton	
O Millimeter	C Kilometer	C Me	tric Ton	C KiloNewton	
C Centimete	er	O Kilo	ogram	C MegaNewton	
	ОК	Car	ncel		

Figure 1.28

1.5.6 Specifying member offsets

Since beam 2 actually spans only the clear distance between the column faces, and not the center to center distance, we can take advantage of this aspect by specifying offsets. Member 2 is OFFSET at its START joint by 6 inches in the global X direction, 0.0 and 0.0 in Y and Z directions. The same member is offset by negative 6.0 inches at its END joint. The corresponding STAAD commands are:

MEMBER OFFSET 2 START 6.0 0.0 0.0 2 END -6.0 0.0 0.0

Steps:

- Since we know that member 2 is the one to be assigned with the offset, let us first select this member prior to defining the offset itself. Select member 2 by clicking on it using the Beams Cursor
 The selected member will be highlighted. (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting members.)
- 2. To define member offsets, click on the **Specification Page** icon located in the top toolbar.

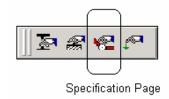


Figure 1. 29

1-36 Tutorial 1

Alternatively, one may go to the *General* | *Spec* Page from the left side of the screen.

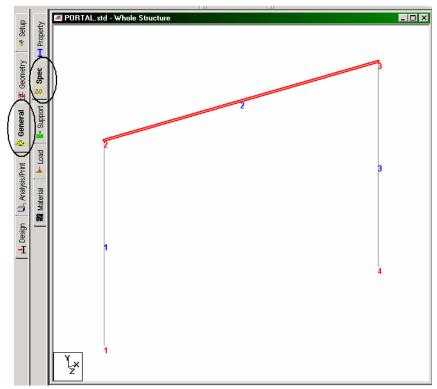


Figure 1. 30

3. In either case, the *Specifications* dialog box shown below comes up. Member Releases and Offsets are defined through the **Beam** button in this dialog box as shown below.

*** Specification	ns - Whole Structur	e 🗙
Specification		
Highlight Ass	signed Geometry	
E dit	Delete	
Node	Beam	Plate
🔲 Toggle Spe		
Assignment Me		
C Assign	Fo Selected Beams	
_	rsor To Assign	
C Assign 1		
Assign	Close	Help

Figure 1.31

1-38 Tutorial 1

4. In the *Beam Specs* dialog box that opens, select the **Offset** tab. We want to define the offset at the start node in the X direction. Hence, make sure that the **Start** option is selected under *Location*. Then, enter **6.0** in the **X** edit box. Since we have already selected the member, let us click on the **Assign** button.

Member Speci	fication					ļ	×
Inacti		, Fire f	Proofing		Imperfe	ection	1
Release	Offset	Cable	Truss	Cor	mpression	Tension	Ļ
Location	•		Direction Global	•			
	Cifset ×	s 6.0	in				
	Y	0	in				
	z	0	in				
		Add	Close		Assign	Help	

Figure 1. 32

5. To apply the offset at the end node, repeat steps 3 and 4, except for selecting the **End** option and providing **-6.0** in the **X** edit box.

Tutorial 1 1-39

After both the *Start* and *End* offsets have been assigned, the model will look as shown below.

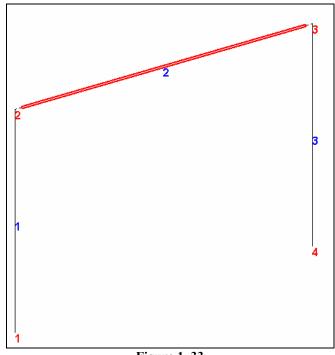


Figure 1.33

Click anywhere in the drawing area to un-highlight the member.

Let us save the work again by pulling down the **File** menu and selecting the **Save** command.

1.5.7 Printing member information in the output file

We would like to get a report consisting of information about all the members including start and end joint numbers (incidence), member length, beta angle and member end releases in the STAAD output file. The corresponding STAAD command is:

PRINT MEMBER INFORMATION ALL

Steps:

1. Since the information is required for all the members, select all the members by going to **Select** | **By All** | **All Beams** menu option.

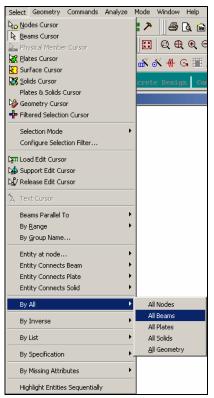


Figure 1.34

2. Then, go to **Commands** | **Pre Analysis Print** | **Member Information** from the top menu bar as shown in the figure below.

Commands Analyze Mode Wir	ndow Help
Plate Thickness Surface Thickness	🖮 🍅 📑 鎟 🗋 📗 🗂 📥
Member Property	ା ଷ୍ର୍ଷ୍ 🖉 ପ୍ 🖻 🛒 ଠ
Material Constants Geometric Constants	• * † † @ # ** 9 💁
Support Specifications	•
Member Specfications Plate Element Specifications Master/Slave Specification	6 6 6
Pre Analysis Print	Problem Statistics
₩ Define Damping for Dynamics	Joint Coordinates
Loading	Element Information
Analysis	Member Information
Post-Analysis Print	Support Information
Design	Material Properties
Miscellaneous	Member Properties
	Entire Table
	Print All
	Clear Above Commands

Figure 1.35

1-42 Tutorial 1

3. Notice that the assignment method is set **To Selection**. Press the **OK** button in this dialog box.

Print Member Information	×
This command requires no additional parameters.	
Assign O To View O To Selection	
• To view • To selection	
	-
OK Cancel Help	
Figure 1. 36	

Click anywhere in the drawing area to un-highlight the members.

Let us also save our structure again by using the **Save** option of the **File** menu.

1.5.8 Specifying Supports

The specifications of this problem (see section 1.2) call for restraining all degrees of freedom at node 1 (FIXED support) and a pinned type of restraint at node 4 (restrained against all translations, free for all rotations) The commands to be generated are :

SUPPORTS 1 FIXED ; 4 PINNED

Steps:

1. To create a support, click on the **Support Page** icon located in the top toolbar as shown below.



Figure 1.37

1-44 Tutorial 1

Alternatively, one may go to the *General* | *Support* Page from the left side of the screen.

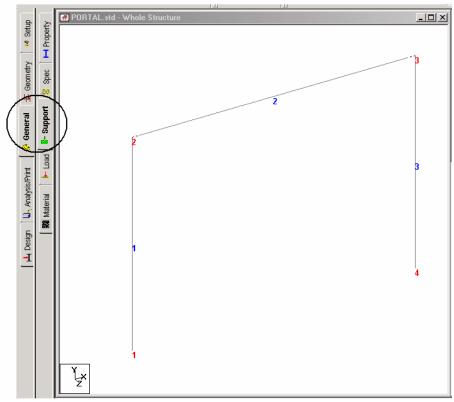


Figure 1.38

2. In either case, the *Supports* dialog box comes up as shown in the next figure. Since we already know that node 1 is to be associated

with a *Fixed* support, using the **Nodes Cursor**, select **node 1**. It becomes highlighted. (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting nodes.)

3. Then, click on the **Create** button in the *Supports* dialog box as shown below.

👷 🕂 Sup	ports - Whol	e Structure		×
Ref	Description			
S1	No support			\times
0	Edit [nment Method Assign To Se Assign To Vir Use Cursor T Assign To Ec	elected Nodes ew o Assign	Dele	ite
	<u>A</u> ssign	Close	Help	

Figure 1.39

1-46 Tutorial 1

4. In the *Create Support* dialog box that opens, select the **Fixed** tab (which also happens to be the default) and click on the **Assign** button as shown below. This creates a FIXED type of support at node 1 where all 6 degrees of freedom are restrained.

Create St	upport				X
Enford Fixe	ed But d	Multilinear Spring Pinned	_	Foundation ed But	Inclined Enforced
B	estraint	FX	2	MX	
	V	FY	2	MY	
	V	FZ	2	ΜZ	
	Add	Cancel		Assign	Help

Figure 1.40

5. To create a PINNED support at node 4, repeat steps 2 to 4, except for selecting **node 4** and selecting the **Pinned** tab in the *Create Support* dialog box.

Tutorial 1 1-47

After the supports have been assigned, the structure will look like the one shown below.

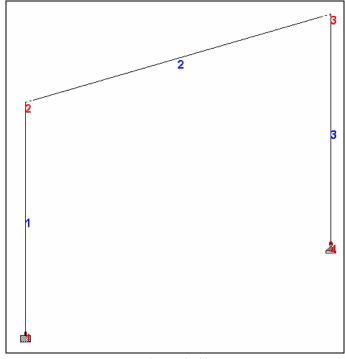


Figure 1.41

After assigning both the supports, let us save our structure using the **File** | **Save** option.

1.5.9 Viewing the model in 3D

Let us see how we can display our model in 3D. To do this, either right-click and select **Structure Diagrams** or go to *View* | *Structure Diagrams* menu.

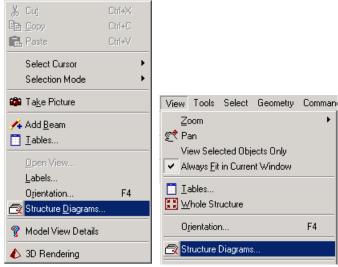


Figure 1.42

In the ensuing dialog box, the *Structure* tab page allows you to set up structural view parameters as explained below.

The options under 3D Sections control how the members are displayed. Selecting None displays the structure without displaying the cross-sectional properties of the members and elements. Selecting Full Sections displays the 3D cross-sections of members, depending on the member properties. Sections Outline displays only the outline of the cross-sections of members.

Let us select **Full Sections** to draw the 3D sections. You can also change the color of the sections by clicking on the *Section Outline* color button under the *Colors* section. Then, click on **OK**.

		Tutorial 1 1-49
	Diagrams	
	Force Limits Animation Design Results Structure Loads and Results Scales Labels	
(3D Sections View Image: None Image: Fill Plates/Solids/Surface Shrink 10 % Image: Full Sections Image: Fill Plates/Solids Perspective Image: Fill Plates/Solids Perspective Image: Sections Dutline Image: Fill Plates/Solids Image: Fill Plates/Solids Perspective Image: Fill Plates/Solids Image: Fill Plates/Solids/Solids Image: Fill Plates/Solids/Solids Image: Fill Plates/Solids/Solids Image: Fill Plates/Solids/Solids Image: Fill Plates/Solids/Solids/Solids Image: Fill Plates/Solids/Solids/Solids/Solids Image: Fill Plates/Solids/S	_
	Margin around structure 10 %	

Figure 1. 43

The resulting diagram is shown below.

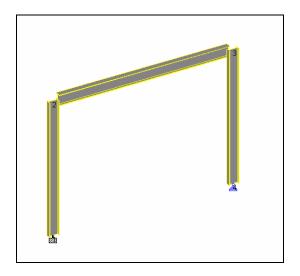


Figure 1.44

1.5.10 Specifying Loads

Three load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial. The corresponding commands to be generated are listed below.

UNIT FEET KIP

LOADING 1 DEAD + LIVE MEMBER LOAD 2 UNI GY -2.5

LOADING 2 WIND FROM LEFT JOINT LOAD 2 FX 10.

LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL) 1 0.75 2 0.75

Steps:

The creation and assignment of load cases involves the following two steps:

- a. First, we will be creating all 3 load cases.
- b. Then, we will be assigning them to the respective members/nodes.

Creating load cases 1 and 2

1. To create loads, first click on the **Load Page** icon located on the top tool bar.

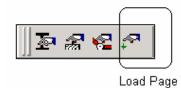


Figure 1.45

Tutorial 1 1-51

Alternatively, one may go to the *General* | *Load* Page from the left side of the screen.

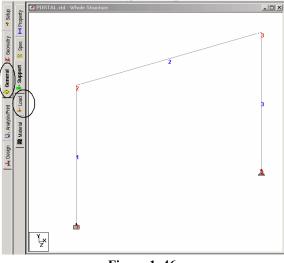


Figure 1.46

Before we create the first load case, we need to change our length units to feet. To do that, as before, utilize the Input Units icon (see section 1.5.5).

Notice that a window titled "Load" appears on the right-hand side of the screen. To create the first load case, highlight **Load Cases Details** and then click on the **Add** button in the *Load* dialog box.

Load	×	→	1. Highlight Load Cases Details.
New Add	Edt Delete © Use Carsor To Assign © Assign To Edit List	→	2. Click on the Add button.
Assign	Close Help		

Figure 1.47

1-52 Tutorial 1

3. The Add New Load Cases dialog box comes up.

The drop-down list box against *Loading Type* is available in case we wish to associate the load case we are creating with any of the ACI, AISC or IBC definitions of Dead, Live, Ice, etc. This type of association needs to be done if we intend to use the program's facility for automatically generating load combinations in accordance with those codes. Notice that there is a check box called *Reducible per UBC/IBC*. This feature becomes active only when the load case is assigned a Loading Type called *Live* at the time of creation of that case. Please refer to STAAD.Pro 2004 Release Report for further details.

As we do not intend to use the automatic load combination generation option, we will leave the *Loading Type* as **None**. Enter **DEAD** + **LIVE** as the *Title* for *Load Case 1* and click on the **Add** button.

Add New : Load Cases	×
Primary Load Generation Define Combinations Auto Load Combination	Primary
	Number 1 Loading Type : None
	Reducible per UBC/IBC
	Title DEAD + LIVE
	Add to Close Help

Figure 1.48

The newly created load case will now appear under the *Load Cases Details* option.

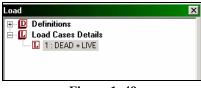
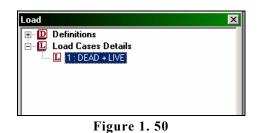


Figure 1.49

Tutorial 1-53

To create the Member load, first **highlight** DEAD + LIVE. You will notice that the *Add New Load Items* dialog box shows more options now.



4. In the *Add New Load Items* dialog box, select the **Uniform Force** option under the *Member Load* item. Specify **GY** as the Direction, enter -2.5 as the Force and click on the **Add** button.

Add New : Load Items		x
Selfweight	Uniform Force	
a Nodal Load		K.
🖉 Member Load	Force	
Uniform Force	d1 0 ft	
📔 Uniform Moment		
Concentrated Force	W1 -2.5 kip/ft d2 0 ft	
Concentrated Momer	W1	
- Linear Varying	d3 0 ft	
📔 Trapezoidal		
	Direction	
Pre/Post Stress		
Fixed End	OX(Local) OGX OPX	
area Load		
🛷 Floor Load	C Y (Local) © GY C PY	
🧳 Plate Loads	O Z (Local) O GZ O PZ	
Surface Loads		
Solid Loads		
Temperature Loads Seismic Loads		
Time History		
Section 2010		
Snow Load		
Response Spectra		
Repeat Load		
S Frequency		
	Add Close Help	

Figure 1.51

The next step is to create the second load case which contains a joint load.

5. Highlight Load Cases Details in the *Load* dialog box. In the *Add New Load Cases* dialog box, once again, we are not associating the load case we are about to create with any code based Loading Type and so, leave that box as **None**. Specify the *Title* of the second load case as **WIND FROM LEFT** and click on the **Add** button.

Add New : Load Cases		X Load	X
Primary Load Generation Define Combinations Auto Load Combination	Primary Number 2 Loading Type: None Reducible per UBC/BC Title VIND FROM LEFT 2. Specify the Title 3. Click on Add Add Cloce	Pedprinter P Definitions P P P Definitions P P P P P P P P P P P P P P P P P P P P P P P	
		New Add Edit Delete	
		T Tassle Land	

Figure 1. 52

6. Next, to create the Joint load, highlight WIND FROM LEFT.

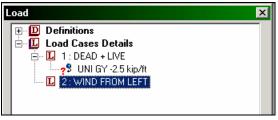


Figure 1. 53

7. In the *Add New Load Items* dialog box, select the **Node** option under the *Nodal Load* item. Specify **10** for **Fx**, and click on the **Add** button.

Add New : Load Items		x
Add New : Load Items	Fx 10 kip Mx 0 kipft Fy 0 kip My 0 kipft Fz 0 kip Mz 0 kipft	×
•	Add Close Help	

Figure 1.54

Creating load case 3

Load cases 1 and 2 were primary load cases. Load case 3 will be defined as a load combination. So, the next step is to define load case 3 as $0.75 \times (\text{Load } 1 + \text{Load } 2)$, which is a load combination.

8. To do this, once again, highlight the Load Cases Details option. In the *Add New Load Cases* dialog box, click on the Define Combinations option from the left-hand side. Specify the *Title* as 75 Percent of [DL+LL+WL].

Add New Load Cases	X	Load
2. Select Define Combinations.	Define Combinations Load No: 3 Name : DL+LL+WL] Type • Normal General Format : a _i * L _i • SRSS Factor 1 • Algebraic Component • ABS Default 2 • Default 2 • Default 2 • Default 2 • SRS Factor 1 • Algebraic Component • Cond Cases: • Load Combination Definition: [A] = Algebraic. • Default 2 • SRS Factor • SRS • SRS • Coad Combination Definition: [A] = Algebraic. • SRS •	 Product State St
	Add Close Help	New Add Edit

Figure 1.55

In the *Define Combinations* box, the default load combination type is set to be Normal, which means an algebraic combination. The other combination types available are called SRSS (square root of sum of squares) and ABS (Absolute). The SRSS type offers the flexibility of part SRSS and part Algebraic. That is, some load cases are combined using the square root of sum of squares approach, and the result is combined with other cases algebraically, as in

```
A + SQRT(B*B + C*C)
where A, B and C are the individual primary cases.
```

We intend to use the default algebraic combination type (Normal).

9. In the *Define Combinations* box, select both load cases from the left side list box (by holding down the 'Ctrl' key) and click on the button. The load cases appear in the right side list box. Then, enter **0.75** in the *Factor* edit box. (These data indicate that we are adding the two load cases with a multiplication factor of 0.75 and that the load combination results would be obtained by algebraic summation of the results for individual load cases.) Press the **Add** button.

Add New Load Cases		x
Primary	Define Combinations	
 Load Generation Define Combinations Auto Load Combination 	Load No: 3 Name : [DL+LL+WL]	
	\mathfrak{S} Normal General Format: $\alpha_i * L_i$ C SRSS Factor 1 C ABS Default [$2i_i$] 1	
	Available Load Cases: Load Combination Definition: (A) = Algebraic.	
	>	
	<u> </u>	
		J
	Add Close Help	

Figure 1.56

Now that we have completed the task of creating all 3 load cases, let us **Close** the *Add New Load Cases* dialog box.

1-58 Tutorial 1

Our next step is to associate load case 1 with member 2. Follow these steps.

- a. Select the first load reference in the *Load* dialog box (UNI GY -2.5 kip/ft).
- b. Make sure that the "Use Cursor to Assign" button is selected under the Assignment Method box.
- c. Click on the Assign button. The cursor changes to نام
- d. Using the cursor, click on member 2.
- e. Finally, click on the Assign button again, or type the 'Esc' button on your keyboard to stop the assignment process.

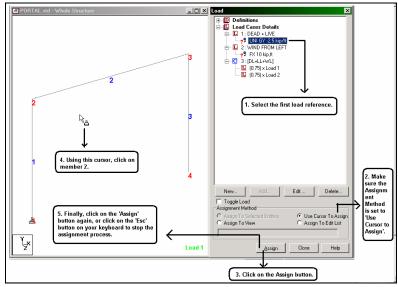
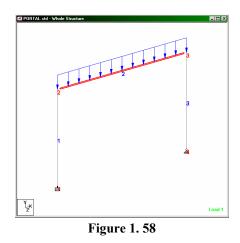


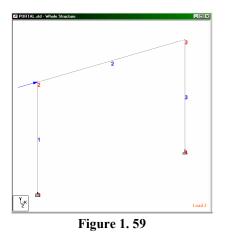
Figure 1.57

After the member load has been assigned, the model will look as shown below.



In a similar fashion, assign the second load case (FX 10 kip, ft) to Node 2.

After assigning the joint load, the model will look as shown below.



Let us once again save our model by pulling down the **File** menu and selecting the **Save** command or by holding the 'Ctrl' key and pressing the 'S' key.

1.5.11 Specifying the analysis type

The analysis type we are required to do is a linear static type. We also need to obtain a static equilibrium report. This requires the command:

PERFORM ANALYSIS PRINT STATICS CHECK

Steps:

1. To specify the Analysis command, go to **Analysis/Print** Page from the left side of the screen. By default, the **Analysis** sub-page from the second row is in focus as shown below.

dn	PORTAL.std - Whole Structure	_ _ _×
🙆 General 🚡 Geometry	Analysis /Print Commands Perform Cable Analysis Poelta Analysis Nonlinear Analysis Dha Perform Analysis Perform Imperfection Analysis Print Option	

Figure 1.60

2. In the *Analysis/Print Commands* dialog box that appears, make sure that the **Perform Analysis** tab is selected. Then, check the **Statics Check** print option. Finally, click on the **Add** button followed by the **Close** button.

Analysis/Print Con	nmands	×
Perform Cable Ana Perform An		Change
	Print Option No Print Load Data Statics Check Statics Load Mode Shapes Both	
After Current	Add Assign Close	Help

Figure 1.61

Let us save the data once again using the File | Save option.

1.5.12 Specifying post-analysis print commands

We would like to obtain the member end forces and support reactions written into the output file. This requires the specification of the following commands:

PRINT MEMBER FORCES ALL PRINT SUPPORT REACTION LIST 1 4

Steps:

1. The dialog box for specifying the above commands is nested in the **Post-Print** sub-page of the **Analysis/Print** page.

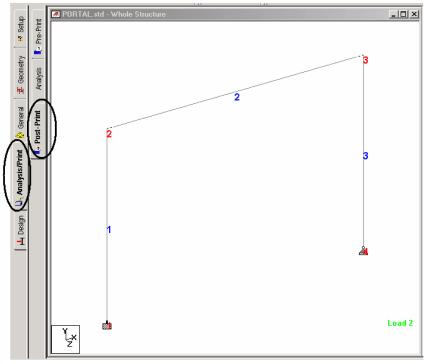


Figure 1. 62

- 2. Next, **select all the members** by rubber-banding around them using the mouse.
- 3. Click on the **Define Commands** button in the data area on the right hand side of the screen.

Highlight Assigned Geometry		
🔲 Toggle Assign		
Define Commar		
Assignment Method		
C Assign To Selected Beams		
Assign To View		
 Use Cursor To Assign 		
C Assign To Edit List		
Assign Close Help		

Figure 1.63

1-64 Tutorial 1

4. In the *Analysis/Print Commands* dialog box that appears, select the **Member Forces** tab and click on the **Assign** button followed by the **Close** button.

Figure 1.64

5. Repeat steps 2 to 4 except for selecting both the supports and selecting the Support Reactions tab in the *Analysis/Print Commands* dialog box. (Recall that the supports can be selected by turning the *Nodes Cursor* on, holding the 'Ctrl' key down, and clicking on the supports.) After clicking on the Assign button, Close the dialog box.

Tutorial 1 1-65

At this point, the *Post Analysis Print* dialog box should resemble the figure shown below.

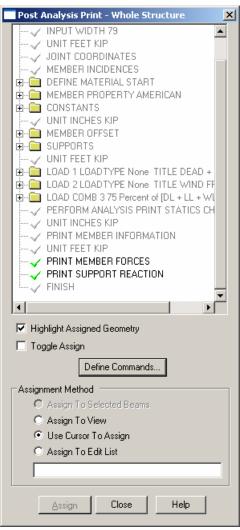


Figure 1.65

Save the work using the File | Save menu option.

1.5.13 Short-listing the load cases to be used in steel design

The steel design has to be performed for load cases 1 and 3 only per the specification at the beginning of this tutorial. To instruct the program to use just these cases, and ignore the remaining, we have to use the LOAD LIST command.

The command will appear in the STAAD file as :

LOAD LIST 1 3

Steps:

In the menus on the top of the screen, go to Commands | Loading
 | Load List option as shown below.

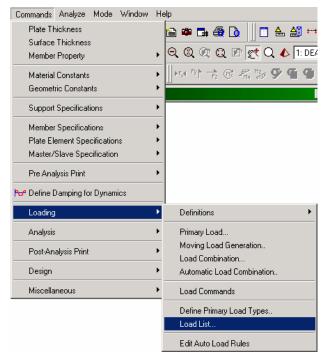


Figure 1.66

A Load List dialog box comes up. From the Load Cases list box on the left, double click on 1: DEAD + LIVE and 3: 75 Percent of [DL+LL+WL] to send them to the Load List box on the right, as shown below. Then click on the OK button to dismiss the dialog box.

Load List		x
Load Cases 2: WIND FROM LEFT Use the > button to transfer selected	Load List 1: DEAD + LIVE 3: 75 Percent of [DL+LL+WL] << Use the < button to remove selected load	OK Cancel Help
load cases to the load list. Use >> to transfer all.	cases from the load list. Use << to remov all.	

Figure 1.67

1.5.14 Specifying steel design parameters

The specifications listed in section 1.2 of this tutorial require us to provide values for some of the terms used in steel design because the default values of those terms are not suitable. The corresponding commands to be generated are:

PARAMETER CODE AISC FYLD 5760 ALL UNT 10.0 MEMB 2 3 UNB 10.0 MEMB 23 TRACK 2 MEMB 2 3 SELECT MEMB 2 3

Steps:

1. To specify steel design parameters, go to **Design** | **Steel** Page from the left side of the screen. Make sure that under the **Current Code** selections on the top right hand side, **AISC ASD** is selected.

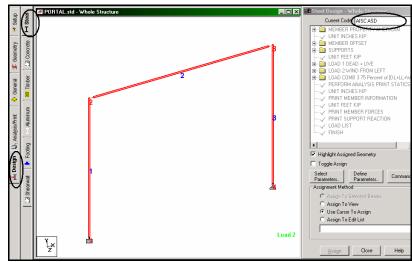


Figure 1.68

2. Click on the **Define Parameters** button in the *Steel Design* dialog box.

✓ Highlight Assigned Geometry		
Toggle Assign		
Select Parameters Commands		
Assignment Method		
C Assign To Selected Beams		
C Assign To View		
Use Cursor To Assign		
C Assign To Edit List		
Assign Close Help		
Figure 1 60		

Figure 1.69

 In the *Design Parameters* dialog box that comes up, select the FYLD option. Then, provide the Yield Strength as 5760 Kip/ft² and click on the Add button.

Design Parameters		x
COMPOSITE CONDIA CONDIA CONHEIGHT CYCLES DFF DJ1 DJ2 DLRATIO DMAX DMAX DMIN EFFWIDTH FPC FSS FU FVLD MPACT KX KY V72 V	FYLD Yield strength of steel: 5760 kip/ft2	
	After Add Assign Close Help	

Figure 1.70

1-70 Tutorial 1

4. To define the remaining parameters, repeat step 3 except for selecting the parameters and providing the values listed below.

Parameter	Value
UNT	10
UNB	10
TRACK	2

- 5. When all the parameters have been added, click on the **Close** button in the *Design Parameters* dialog box.
- 6. The next step is to assign these parameters to specific members of the model. From looking at the requirements listed in the beginning of this tutorial, we know that the FYLD parameter is to be assigned to all the members, while the remaining parameters are to assigned to members 2 and 3.

As before, use the Use Cursor to Assign method to assign these parameters.

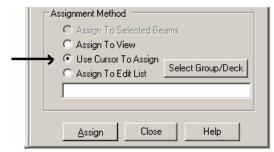


Figure 1.71

After all the design parameters have been assigned, the *Steel Design* dialog box will look as shown below.

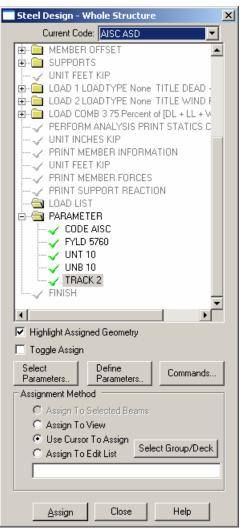


Figure 1.72

1-72 Tutorial 1

7. To specify the SELECT command, click on the **Commands** button in the *Steel Design* dialog box as shown below. The SELECT command is an instruction to the program to fetch and assign the least-weight cross-section which satisfies all the code requirements (PASSes) for the member.

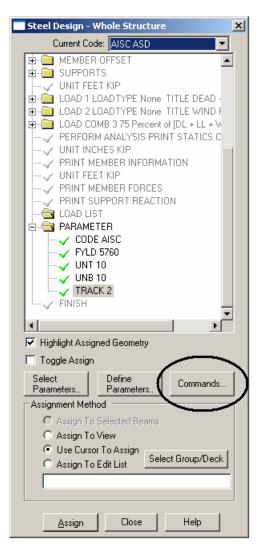


Figure 1.73

8. In the *Design Commands* dialog box that appears, click on the **SELECT** option. Then, click on the **Add** button followed by the **Close** button.

Design Commands		×
CHECK CODE FIXED GROUP GROUP MEMBER TAKE OFF SELECT SELECT OPTIMIZED SELECT OPTIMIZED SELECT WELD SELECT WELD TAKE OFF	SELECT Select least weight section size based on specifications of the desired code. The selection is done using the results from the most recent analysis and iterating on section sizes until a least weight size is obtained. This command has no additional parameters.	
	Current Add Assign Close Help	

Figure 1.74

9. Once again, we need to associate this command with members 2 and 3. You may either use the Use Cursor to Assign method or first select members 2 and 3 and then use the Assign to Selected Beams option.

After the parameters are assigned, click anywhere in the drawing area to un-highlight the members.

Let us save our structure by pulling down the **File** menu and selecting the **Save** command.

1.5.15 Re-specifying the analysis command

When the analysis & design engine executes the member selection operation we specified in the previous step, a new set of properties will end up being assigned to those members. This has the effect of changing the stiffness distribution for the entire structure. Since the structure is statically indeterminate, we ought to re-analyze it if we want the nodal displacements, member forces, etc. to reflect this new stiffness distribution. The command to be generated is hence:

PERFORM ANALYSIS

Steps:

 To specify the Analysis command, repeat step 1 of Section 1.5.11 of this tutorial. In the *Analysis/Print Commands* dialog box that comes up, select the **Perform Analysis** tab. Since we are not interested in a statics check report once again, let us check the **No Print** option. Finally, click on the **Add** button followed by the **Close** button.

We should again save the work using File | Save.

1.5.16 Re-specifying the TRACK parameter

The final calculation we need to do is make sure the current set of member properties pass the code requirements based on the up-todate member forces. This will require that we do a code checking operation again. To restrict the output produced to a reasonable level, we specify the TRACK parameter again as:

TRACK 1 ALL

Steps:

- 1. To define and assign **1.0** for the **TRACK** parameter, repeat steps 1 to 4 of Section 1.5.14 of this tutorial.
- 2. Next, **select all the members** by rubber-banding around them using the mouse. (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting members.) Then, assign this parameter to all the members.

1.5.17 Re-specifying the CHECK CODE command

The analysis operation carried out in response to the command in Section 1.5.15 will create a new set of member forces. These forces will very likely be quite different from those which were used in the member selection operation (see the commands of section 1.5.14). Consequently, we have to verify that the structure is safely able – from the standpoint of the design code requirements – to carry these new forces. A code checking operation, which uses the up-to-date cross sections of the members, and the latest member forces, will provide us with a status report on this issue. The command to be generated is hence:

CHECK CODE ALL

Steps:

 If you have wandered away from the Steel Design page, from the Commands menu on top of the screen, select Design | Steel Design. 2. Click on the **Commands** button in the *Steel Design* dialog box as shown below.

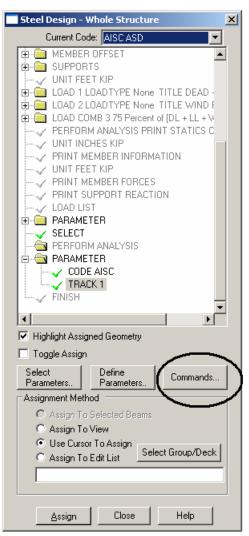


Figure 1.75

1-78 Tutorial 1

3. In the *Design Commands* dialog box that appears, click on the **Check Code** tab. Then, click on the **Add** button followed by the **Close** button.

Design Commands		×
CHECK CODE FIXED GROUP GROUP MEMBER TAKE OFF SELECT SELECT OPTIMIZED SELECT WELD SELECT WELD SELECT WELD TRUSS TAKE OFF	CHECK CODE Check the adequacy of members based on specifications of the desired code.	
	This command has no additional parameters.	
	Current Add Assign Close Help	

Figure 1.76

4. Since the CHECK CODE command has to be assigned to all the members, the easiest way to do that is to click on the **Assign to View** button.

Assignment Method	1
C Assign To Selected Beams	L
💿 Assign To View 🧲	⊢
🔿 Use Cursor To Assign	L
🔿 Assign To Edit List	L
	L
	L
	L
Assign Close Help	L

Figure 1.77

We have now completed the tasks for assigning the input for this model. Let us save the file one last time using the **File** | **Save** option.

1.6 Viewing the input command file

Steps:

Let us now take a look at the data that has been written into the file that we just saved earlier. The contents of the file can be viewed either by clicking on the **STAAD Editor** icon or, by going to the *Edit* menu and choosing *Edit Input Command File* as shown below.

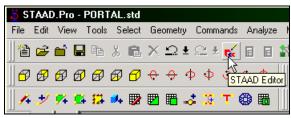


Figure 1.78

💭 Undo Assign Support	Ctrl+Z Ctrl+Y
券 Cu <u>t</u> ≌a <u>C</u> opy ₽ Paste ★ <u>D</u> elete	Ctrl+X Ctrl+C Ctrl+V Del
Take Picture Copy Picture	
ត Edit Input Command File	- k -
Create New VB Macro Edit Existing VB Macro	

Figure 1.79

A new window will open up with the data listed as shown here:

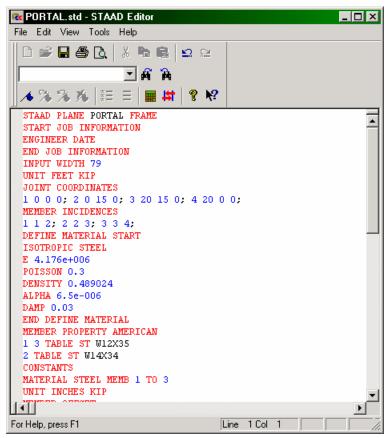


Figure 1.80

This window and the facilities it contains is known as the *STAAD Editor*.

We could make modifications to the data of our structure in this *Editor* if we wish to do so. Let us *Exit* the *Editor* without doing so by selecting the **File** | **Exit** menu option of the editor window (not the File | Exit menu of the main window behind the editor window).

As we saw in Section 1.1, we could also have created the same model by typing the relevant STAAD commands into a text file using either the STAAD editor, or by using any external editor of our choice. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to section 1.8 where we perform the analysis and design on this model.

1-82 Tutorial 1

1.7 Creating the model using the command file

Let us now use the command file method to create the model for the above structure. The commands used in the command file are described later in this section.

The STAAD.Pro command file may be created using the built-in editor, the procedure for which is explained further below in this section. 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. A typical editor screen is shown below to illustrate its general appearance.

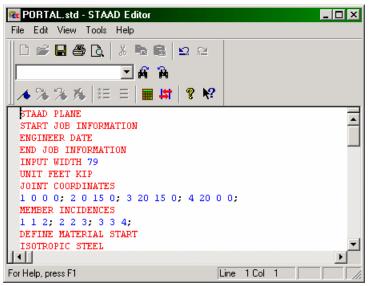


Figure 1.81

To access the built-in editor, first start the program using the procedure explained in Section 1.3. Next, follow step 1 of Section 1.4.

Location : C:Mest
A PLANE structure is bound by a global XY coordinate system with loads in the sam plane.
plane.
□ Length Units □ □ Force Units
Foot O Meter KiloPound O DecaNewtor
C Centimeter C Metric Ton C MegaNewtor

Figure 1.82

You will then encounter the dialog box shown in the figure shown below. In that dialog box, choose **Open STAAD Editor**.

Where do you want to go ?	×
Add Beam Add Plate Open Stucture Wizard Open Stucture Wizard Edit Job Information	
Begin building your model using STAAD syntax commands (non-graphical interface the STAAD editor.	e) through
✓ Display this dialog box at the Startup < Back	Help

Figure 1.83

1-84 Tutorial 1

At this point, the editor screen similar to the one shown below will open.

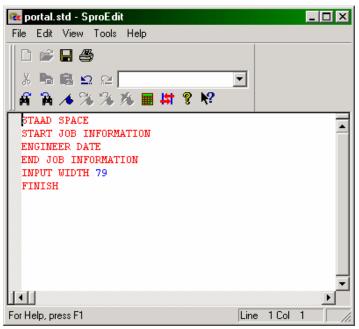


Figure 1.84

Delete all the command lines displayed in the editor window and type the lines shown in bold below (You don't have to delete the lines if you know which to keep and where to fill in the rest of the commands). The commands may be typed in upper or lower case letters. Usually the first three letters of a keyword are all that are needed -- the rest of the letters of the word are not required. The required letters are underlined. ("PLANE" = "PLA" = "plane" = "pla")

STAAD PLANE PORTAL FRAME

Every STAAD.Pro input file has to begin with the word **STAAD**. The word PLANE signifies that the structure is a plane frame (in the XY plane). The remainder of the words are the title of the problem, which is optional.

If a line is typed with an asterisk in the first column, it signifies that the line is a comment line and should not be executed. For example, one could have put the optional title above on a separate line as follows.

* PORTAL FRAME

<u>UNIT FEET KIP</u>

Specify the force and length units for the commands to follow.

<u>JOI</u>NT <u>COO</u>RDINATES 1 0. 0.; 2 0. 15.; 3 20. 15.; 4 20. 0.

Joint numbers and their corresponding global X and Y coordinates are provided above. For example, 3 20 15. indicates that node 3 has an X coordinate of 20 ft and a Y coordinate of 15 ft. Note that the reason for not providing the Z coordinate is because the structure is a plane frame. If this were a space frame, the Z coordinate would also be required. Semicolons (;) are used as line separators. In other words, data which is normally put on multiple lines can be put on one line by separating them with a semicolon.

<u>MEMBER INCIDENCE</u> 1 1 2;2 2 3;3 3 4

The members are defined by the joints to which they are connected.

<u>MEM</u>BER <u>PRO</u>PERTY AMERICAN 1 3 <u>TA</u>BLE <u>ST</u> W12X35 2 <u>TA</u>BLE <u>ST</u> W14X34

Members 1 and 3 are assigned a W12X35 section from the built-in AMERICAN steel table. Member 2 has been assigned a W14X34. The word ST stands for standard single section. Sections 5.20.1 through 5.20.5 of the STAAD Technical Reference Manual explain the convention for assigning member property names.

UNIT INCHES CONSTANTS E 29000.0 ALL POISSON 0.3 ALL

The length unit is changed from FEET to INCHES to facilitate input of the modulus of elasticity (E). The keyword CONSTANT is required before material properties such as E, density, Poisson's ratio, coefficient of thermal expansion (ALPHA) etc. can be provided. See Section 5.26 of the STAAD Technical Reference Manual for more information.

<u>MEMBER OFF</u>SET 2 <u>STA</u>RT 6.0 0. 0. 2 <u>END</u> -6.0 0. 0.

The beam member is physically connected to the 2 columns at the face of the column, and not at the column centerline. This creates a rigid zone, about half the depth of the columns, at the 2 ends of the beam 2. This rigid zone is taken advantage of using member offsets (It is the user's choice whether or not he/she wishes to use these). So, the above commands define that member 2 is eccentrically connected or OFFSET at its START joint by 6 inches in the global X direction, 0.0 and 0.0 in Y and Z directions. The same member is offset by negative 6.0 inches at its END joint. See Section 5.25 of the STAAD Technical Reference Manual for more information.

PRINT MEMBER INFORMATION ALL

The above command is self-explanatory. The information that is printed includes start and end joint numbers (incidence), member length, beta angle and member end releases.

<u>SUP</u>PORTS 1 <u>FIX</u>ED ; 4 <u>PIN</u>NED

A fixed support is located at joint 1 and a pinned support (fixed for translations, released for rotations) at joint 4. More information on the support specification is available in Section 5.27 of the STAAD Technical Reference Manual.

<u>UNIT FT</u>

The length unit is changed to FEET to facilitate input of loads.

<u>LOA</u>DING 1 DEAD + LIVE <u>MEM</u>BER <u>LOA</u>D 2 <u>UNI</u> GY -2.5

The above commands identify a loading condition. DEAD + LIVE is an optional title to identify this load case. A UNIformly distributed MEMBER LOAD of 2.5 kips/ft is acting on member 2 in the negative global Y direction. Member Load specification is explained in Section 5.32 of the STAAD Technical Reference Manual.

<u>LOA</u>DING 2 WIND FROM LEFT <u>JOI</u>NT <u>LOA</u>D 2 FX 10.

The above commands identify a second load case. This load is a JOINT LOAD. A 10 kip force is acting at joint 2 in the global X direction.

LOAD COMBINATION 3 75 PERCENT OF (DL+LL+WL) 1 0.75 2 0.75

This command identifies a combination load with an optional title. The second line provides the components of the load combination case - primary load cases and the factors by which they should be individually multiplied.

PERFORM ANALYSIS PRINT STATICS CHECK

This command instructs the program to proceed with the analysis and produce a report of static equilibrium checks. Section 5.37 of the STAAD Technical Reference Manual offers information on the various analysis options available.

<u>PRINT MEMBER FOR</u>CES ALL <u>PRI</u>NT <u>SUP</u>PORT <u>REA</u>CTION LIST 1 4

The above print commands are self-explanatory. The member forces are in the member local axes while support reactions are in the global axes.

LOAD LIST 1 3 <u>PARAMETERS</u> <u>CODE AISC</u> <u>UNT</u> 10.0 MEMB 2 3 <u>UNB</u> 10.0 MEMB 2 3 <u>FYLD</u> 5760 ALL <u>TRA</u>CK 2.0 MEMB 2 3 <u>SEL</u>ECT <u>MEM</u>BER 2 3

The above sequence of commands is used to initiate the steel design process. The command PARAMETERS is followed by the various steel design parameters. Parameters are specified typically when their values differ from the built-in program defaults. Specifications of the AISC ASD code are to be followed. A parameter list for the AISC code is available in Table 3.1 of the Technical Reference Manual. ALL members have 10 ft unsupported length for the top and bottom flange (UNT and UNB). UNT and UNB are used to compute the allowable compressive stress in bending. The yield strength of steel is specified as 5760 ksf (40 ksi) since it is different from the default value of 36 ksi. The TRACK parameter controls the level of description of the output, 2.0 being the most detailed. The LOAD LIST command lists the load cases (1 and 3) to be used in the design. The SELECT MEMBER command asks the program to come up with the most economical section for members 2 and 3 in the context of the above analysis.

PERFORM ANALYSIS

When the analysis & design engine executes the member selection operation we specified in the previous step, a new set of properties will end up being assigned to those members. This has the effect of changing the stiffness distribution for the entire structure. Since the structure is statically indeterminate, we ought to re-analyze it if we want the nodal displacements, member forces, etc. to reflect this new stiffness distribution. The above command instructs the program to do another cycle of analysis.

PARAMETER TRACK 1 ALL

The TRACK parameter is re-specified. It controls the level of information produced in the steel design output. We have lowered it from 2.0 we specified earlier to 1.0 since we aren't interested in the highest level of detail at this time.

CHECK CODE ALL

The analysis operation carried out earlier will create a new set of member forces. These forces will very likely be quite different from those which were used in the member selection operation. Consequently, we have to verify that the structure is safely able – from the standpoint of the design code requirements – to carry these new forces. A code checking operation, which uses the up-todate cross sections of the members, and the latest member forces, will provide us with a status report on this issue.

<u>FINI</u>SH

A STAAD run is terminated using the FINISH command.

Save the file and return to the main screen.

This concludes the session on generating our model as a command file using the built-in editor. If you wish to perform the analysis and design, you may proceed to the next section of this manual. The on-screen post-processing facilities are explained in Section 1.10.

(Remember that without successfully completing the analysis and design, the post-processing facilities will not be accessible.)

1.8 Performing Analysis/Design

STAAD.Pro performs Analysis and Design simultaneously. In order to perform Analysis and Design, select the **Run Analysis** option from the **Analyze** menu.



If the structure has not been saved after the last change was made, you should save the structure first by using the *Save* command from the *File* menu.

When you select the *Run Analysis* option from the *Analyze* menu, the following dialog box appears:

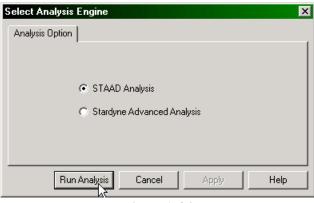


Figure 1.86

We are presented with the choice of 2 engines : the STAAD engine and the STARDYNE Advanced Analysis engine. The STARDYNE Analysis engine is suitable for advanced problems such as Buckling Analysis, Modal Extraction using various methods, etc. However, if the calculations call for steel or concrete design, UBC load generation, etc., we have to select the STAAD engine. So, let us ensure that the radio button is on the **STAAD** engine. 1-92 Tutorial 1

Click on the Run Analysis button.

As the analysis progresses, several messages appear on the screen as shown in the figure below.

茸 STAAD Analysis and Design	
<pre>++ Processing and setting up Load Vector. ++ Processing Element Stiffness Matrix. ++ Processing Global Stiffness Matrix. ++ Finished Processing Global Stiffness Matrix. ++ Processing Triangular Factorization. ++ Finished Triangular Factorization. ++ Calculating Joint Displacements. ++ Finished Joint Displacement Calculation. ++ Calculating Member Forces. ++ Analysis Successfully Completed ++ ++ Calculating Section Forces</pre>	21:19:52 ▲ 21:19:52 21:19:52 0 sec 21:19:52 0 sec 21:19:52 0 sec 21:19:52 0 sec 21:19:52
<pre>++ Performing Steel Design ++ Start Steel Design ++ Start Steel Design ++ Finished Design ++ Creating Reaction File (DSP) ++ Creating Reaction Forces ++ Creating Section Forces File (BMD) ++ Creating Section Displace File (SCN) ++ Creating Design information File (DGN) ++ Done.</pre>	21:19:53 0 sec 21:19:53 21:19:53
0 Error(s), 0 Warning(s) ** End STAAD.Pro Run Elapsed Time = 2 Secs ** Output Written to File: PORTAL.anl	•
 View Dutput File Go to Post Processing Mode Stay in Modelling Mode 	Done

Figure 1.87

Notice that we can choose from the three options available in the above dialog box:

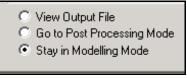


Figure 1.88

Tutorial 1-93

These options are indicative of what will happen after we click on the *Done* button.

The *View Output File* option allows us to view the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands we specified during the model generation process. It also tells us whether any errors were encountered, and if so, whether the analysis and design was successfully completed or not. Section 1.9 offers additional details on viewing and understanding the contents of the output file.

The *Go to Post Processing Mode* option allows us to go to graphical part of the program known as the Post-processor. This is where one can extensively verify the results, view the results graphically, plot result diagrams, produce reports, etc. Section 1.10 explains the Post processing mode in greater detail.

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

1.9 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis were performed properly. For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

We can access the output file using the method explained at the end of the previous section. Alternatively, we can select the **File** | **View** | **Output File** | **STAAD Output** option from the top menu. The STAAD.Pro output file for the problem we just ran is shown in the next few pages.

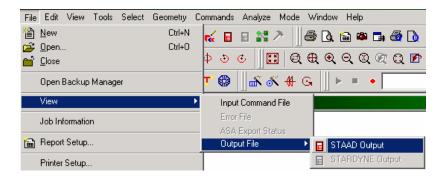


Figure 1.89

The STAAD.Pro output file is displayed through a file viewer called *SproView*. This viewer allows us to set the text font for the entire file and print the output file to a printer. Use the appropriate *File* menu option from the menu bar.

rcframe.anl	- SproVie	w
File Edit View	v Help	
Open	Ctrl+O	N?
Print	Ctrl+P	_
Print Preview		
Print Setup		
Page Setup		
Font Setup		
🗸 Top Most		
Recent File		
Exit		

Figure 1.90

By default, the output file contains a listing of the entire Input also. You may choose not to print the echo of the Input commands in the Output file. Please select *Commands* | *Miscellaneous* | *Set Echo* option from the menu bar and select the *Echo Off* button.

It is quite important that we browse through the entire output file and make sure that the results look reasonable, that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the postprocessing mode – the graphical screens where results can be viewed graphically. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
*
                   STAAD.Pro
                   Version
                               Bld
                   Proprietary Program of
                   Research Engineers, Intl.
                                                       *
         +
                   Date=
                   Time=
              USER ID:
         *****
 1. STAAD PLANE PORTAL FRAME
 2. START JOB INFORMATION
 3. ENGINEER DATE
 4. END JOB INFORMATION
 5. INPUT WIDTH 79
 6. UNIT FEET KIP
 7. JOINT COORDINATES
 8. 1 0 0 0; 2 0 15 0; 3 20 15 0; 4 20 0 0
 9. MEMBER INCIDENCES
10. 1 1 2; 2 2 3; 3 3 4
11. DEFINE MATERIAL START
12. ISOTROPIC STEEL
13. E 4.176E+006
14. POISSON 0.3
15. DENSITY 0.489024
16. ALPHA 6.5E-006
17. DAMP 0.03
18. END DEFINE MATERIAL
19. MEMBER PROPERTY AMERICAN
20. 1 3 TABLE ST W12X35
21. 2 TABLE ST W14X34
22. CONSTANTS
23. MATERIAL STEEL MEMB 1 TO 3
24. UNIT INCHES KIP
25. MEMBER OFFSET
26. 2 START 6 0 0
27. 2 END -6 0 0
28. SUPPORTS
29. 1 FIXED
30. 4 PINNED
31. UNIT FEET KIP
32. LOAD 1 DEAD + LIVE
33. MEMBER LOAD
34. 2 UNI GY -2.5
35. LOAD 2 WIND FROM LEFT
36. JOINT LOAD
37. 2 FX 10
38. LOAD COMB 3 75 PERCENT OF {DL+LL+WL]
39. 1 0.75 2 0.75
40. PERFORM ANALYSIS PRINT STATICS CHECK
        PROBLEM STATISTICS
 NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =
                                            4/
                                                   3/
                                                         2
 ORIGINAL/FINAL BAND-WIDTH= 1/ 1/
                                           6 DOF
                           2, TOTAL DEGREES OF FREEDOM =
 TOTAL PRIMARY LOAD CASES =
                                                            7
 SIZE OF STIFFNESS MATRIX =
                               1 DOUBLE KILO-WORDS
 REQRD/AVAIL. DISK SPACE = 12.0/ 3884.9 MB, EXMEM = 488.4 MB
      STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.
                                                         1
      DEAD + LIVE
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING 1 )
   SUMMATION FORCE-X =
                         0.00
   SUMMATION FORCE-Y =
                         -47.50
   SUMMATION FORCE-Z =
                          0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
              0.00 MY=
                                0.00 MZ=
                                                -475.00
  MX=
```

Tutorial 1 1-97

```
***TOTAL REACTION LOAD ( KIP FEET ) SUMMARY (LOADING
                                                     1)
   SUMMATION FORCE-X =
                          0.00
   SUMMATION FORCE-Y =
                            47.50
   SUMMATION FORCE-Z =
                            0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
                                  0.00 MZ=
               0.00 MY=
                                                 475.00
  MX=
MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING
                                                 1)
        MAXIMUMS AT NODE
  X = 1.82363E-01
                        2
  Y = -1.46578E - 02
                        3
  Z = 0.0000E+00
                        0
  RX= 0.00000E+00
                        0
  RY= 0.00000E+00
                        0
  RZ= -4.82525E-03
                        2
      STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.
                                                            2
      WIND FROM LEFT
***TOTAL APPLIED LOAD ( KIP FEET ) SUMMARY (LOADING
                                                    2)
   SUMMATION FORCE-X =
                           10.00
   SUMMATION FORCE-Y =
                            0.00
   SUMMATION FORCE-Z =
                             0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
               0.00 MY=
                                  0.00 MZ=
                                                -150.00
  MX=
***TOTAL REACTION LOAD ( KIP FEET ) SUMMARY (LOADING
                                                  2)
   SUMMATION FORCE-X = -10.00
   SUMMATION FORCE-Y =
                            0.00
   SUMMATION FORCE-Z =
                             0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX=
               0.00 MY=
                                  0.00 MZ=
                                                 150.00
MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING
                                                 2)
        MAXIMUMS AT NODE
  X = 7.27304E-01
                        2
  Y = 2.47268E-03
                        2
  z = 0.0000E+00
                        0
  RX= 0.00000E+00
                        0
  RY= 0.00000E+00
                        0
  RZ= -5.48842E-03
                        4
************ END OF DATA FROM INTERNAL STORAGE ************
 41. UNIT INCHES KIP
 42. PRINT MEMBER INFORMATION ALL
MEMBER INFORMATION
-----
MEMBER
         START END
                            LENGTH
                                       BETA
         JOINT JOINT
                            (INCH)
                                       (DEG)
                                             RELEASES
           1
                  2
                            180.000
                                       0.00
  1
  2
           2
                           228.000
                                       0.00
                  3
                            180.000
                                       0.00
  з
           3
                  4
************ END OF DATA FROM INTERNAL STORAGE ************
```

```
1-98 Tutorial 1
```

		FORCES	STRUCTU	RE TYPE =	= PLANE			
		ARE KI	P FEET					
MEMBER	LOAD	JT	AXIAL	SHEAR-Y	SHEAR-Z	TORSION	MOM-Y	MOM-Z
1	1	1	23.18	-3.99	0.00	0.00	0.00	-11.48
	_		-23.18	3.99				
	2	1 2	-4.10 4.10	7.68 -7.68				
	3	1	14.30	2.77	0.00	0.00	0.00	
		2	-14.30	-2.77	0.00	0.00	0.00	
2	1	2	3.99	23.18	0.00	0.00	0.00	36.81
		3	-3.99	24.32	0.00	0.00	0.00	-47.72
	2	2 3	2.32 -2.32	-4.10 4.10	0.00	0.00 0.00	0.00	-45.27 -32.69
	3	2	4.73	14.30	0.00	0.00	0.00	-6.34
	5	3	-4.73	21.32	0.00	0.00	0.00	-60.31
3	1	3	24.32	3.99	0.00	0.00	0.00	59.88
	•		-24.32	-3.99	0.00	0.00	0.00	0.00
	2	3 4	4.10 -4.10	2.32 -2.32	0.00 0.00	0.00 0.00	0.00	34.74 0.00
	3	3	21.32	4.73	0.00	0.00	0.00	70.97
			-21.32	-4.73	0.00	0.00	0.00	0.00
45. 1 SUPPOI	PRINT RT REA	SUPPORT	REACTION	LIST 1 4 FEET	STRUCTUR	************* E TYPE = PI	LANE	
45. 1 SUPPOI	PRINT RT REA LOAD	SUPPORT ACTIONS - FORCE-X	REACTION	LIST 1 4 FEET Y FORCE	STRUCTURI E-Z M	E TYPE = PI OM-X MC	LANE DM-Y MC	DM Z
45. 1 SUPPOI	PRINT RT REA LOAD 1	SUPPORT CTIONS - FORCE-X 3.99	REACTION UNIT KIP FORCE- 23.1	LIST 1 4 FEET Y FORCE 8 0.	STRUCTURI 5-Z Ma	E TYPE = PI OM-X MC 0.00 C	LANE DM-Y MC	48
45. 1 SUPPOI	PRINT RT REA LOAD	SUPPORT ACTIONS - FORCE-X 3.99 -7.68	REACTION UNIT KIP FORCE- 23.1 3 -4.1	LIST 1 4 FEET Y FORCE 8 0. 0 0.	STRUCTUR E-Z Ma .00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67	
45. 1 SUPPOI	PRINT RT REA LOAD 1 2 3 1	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99	REACTION UNIT KIP 5 FORCE- 23.1 4 -4.1 14.3 9 24.3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 0 0. 2 0.	STRUCTURI 5-Z Ma 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C	LANE DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1	PRINT RT REA LOAD 1 2 3 1 2	SUPPORT FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32	REACTION UNIT KIP 5 FORCE- 23.1 4 -4.1 14.3 9 24.3 2 4.1	LIST 1 4 FEET Y FORCE 8 0. 0 0. 0 0. 2 0. 0 0.	STRUCTUR -Z M -00	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4	PRINT RT REA LOAD 1 2 3 1 2 3	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 4.1 21.3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4	PRINT RT REA LOAD 1 2 3 1 2 3 3	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 4.1 21.3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ******	PRINT RT REA LOAD 1 2 3 1 2 3 3	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73 **** END JIST 1 3	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 4.1 21.3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0	PRINT RT REA LOAD 1 2 3 1 2 3 3 *******	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73 **** END ALST 1 3 TER LISC	REACTION UNIT KIP 5 FORCE- 9 23.1 4 -4.1 14.3 9 24.3 9 24.3 9 24.3 9 24.3 9 21.3 0 0F LATEST	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 49. 1	PRINT RT REA LOAD 1 2 3 1 2 3 4 ******* LOAD I PARAME CODE A FYLD 5	SUPPORT CTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73 	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 21.3 OF LATEST 3 1 TO 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 49. 1 50. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 ******* LOAD I PARAME CODE A FYLD 5 UNT 10	SUPPORT ACTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 **** END MIST 1 3 TER LIST 1 3 TER	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 21.3 OF LATEST 3 1 TO 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ACTIONS - FORCE-X 3.99 -7.68 -2.77 -3.99 -2.32 -4.73 	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 1 2 2 3 3 3 2 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00
45. 1 SUPPOI OINT 1 1 4 ****** 46. 1 47. 1 48. 0 48. 0 49. 1 50. 1 51. 1 52. 1	PRINT RT REA LOAD 1 2 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 1 2 2 2 3 3 1 2 2 3 3 3 1 2 2 3 3 3 1 2 3 3 1 2 3 3 1 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 1 2 2 3 3 2 3 3 2 3 3 1 2 3 3 1 2 2 3 3 2 3 3 3 2 3 3 2 3 3 2 3 3 3 3	SUPPORT ICTIONS - FORCE-X 3.99 -7.66 -2.77 -3.99 -2.32 -4.73 ISC ISC ISC ISC ISC MEME 2 MEME 2 MEME 2 MEME 2	REACTION UNIT KIP FORCE- 23.1 -4.1 14.3 24.3 24.3 24.3 24.3 24.3 21.3 OF LATEST 3 3 3 3 3 3 3	LIST 1 4 FEET Y FORCE 8 0. 0 0. 2 0. 2 0. 2 0.	STRUCTURI -Z Ma 00 0 00 0 00 0 00 0 00 0 00 0	E TYPE = PI OM-X MC 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C 0.00 C	DM-Y MC 0.00 -11 0.00 67 0.00 42 0.00 0 0.00 0	48 7.93 2.34 0.00

Tutorial 1 1-99

****				ч I	PROPERTIES IN INCH UNIT
	=====================================				AX = 8.85
DESIGN CODE *	ST W14 		 	I	AY = 3.39 AZ = 3.47
AISC-1989 *				=== ===	SY = 5.82 SZ = 42.05
*	<leng< td=""><td>GTH (FT)= 1</td><td>9.00> </td><td></td><td>RY = 1.49 RZ = 5.73</td></leng<>	GTH (FT)= 1	9.00>		RY = 1.49 RZ = 5.73
PARAMETER		XIP-FEET)	L1		STRESSES
IN KIP INCH	1	11	LI L1		IN KIP INCH
KL/R-Y= 153.2	•	L3		L3	FA = 6.36
KL/R-Z= 39.7	6 +	13			fa = 0.45
UNL = 120.00 CB = 1.00		L3	L1		FCZ = 21.68 FTZ = 24.00
CMY = 0.8	5 13			L3	FCY = 30.00
CMZ = 0.8 FYLD = 40.0	5 +			L1	FTY = 30.00 fbz = 20.13
FYLD = 40.00 NSF = 1.00	D D ++	+++	-+++		fby = 0.00
NSF = 1.00 DFF = 0.00 dff = 0.00	0 13.8				Fey = 6.36 Fez = 94.46
dff = 0.00	J		TE MZ ENVELO TH LOAD NO.		Fez = 94.46 FV = 16.00
					fv = 0.17
		SHEAR-Y		MOMENT-Y	MOMENT-Z
VALUE	4.7	24.3	0.0	0.0	70.5
LOCATION	0.0	19.0	0.0	0.0	9.5
LOADING	3	1	0	0	1
***********	*******				***********
* *		DESIGN SUM	MARY (KIP-1	FEET)	
*		/			
* RESULT, FX		FICAL COND/ MY	RATIO/ MZ	LOADIN LOCATIC	
PASS 3.99		ISC- H1-3 0.00	0.999 -70.55	1 9.50)
* *********					

1-100 Tutorial 1

STAAD.PRO MEMBER SELECTION - (AISC 9TH EDITION) - 1 PROPERTIES | Y ******** IN INCH UNIT | ----- | = 1 = IMEMBER 3 * Т 1 1 AX = 10.00 --Z | ST W14X34 AY = 3.61 * 1 |DESIGN CODE * | AZ = 4.10 1 -- 1 | AISC-1989 * == SY = 6.91 ___ ===|=== SZ = 48.64* |<---LENGTH (FT)= 15.00 --->| RY = 1.53 RZ = 5.83 ***** 71.0 (KIP-FEET) | PARAMETER STRESSES |L3 | L3 IN KIP INCH IN KIP INCH | L3 L3 |----- + -----| KL/R-Y= 117.92 | LЗ FA = 10.72 | | KL/R-Z= 30.87 fa = 2.13 | + FCZ = 21.95 | UNL = 120.00 | L3 L3 | CB 1.00 FTZ = 24.00 | = + | CMY = 0.85 | L3 FCY = 30.00 | = 0.85 + L3 L3 FTY = 30.00 | I CMZ | FYLD = 40.00 | L0 fbz = 17.51 | | NSF = 1.00 +---+---+---+---+---+---+---+----| fby = 0.00 | | DFF = 0.00 -3.9 | dff = 0.00 Fey = 10.74 | ABSOLUTE MZ ENVELOPE Fez = 156.71 | FV = 16.00 | (WITH LOAD NO.) fv = 1.31 | MAX FORCE/ MOMENT SUMMARY (KIP-FEET) -----SHEAR-Y SHEAR-Z MOMENT-Y MOMENT-Z AXIAL 24.3 4.7 0.0 0.0 71.0 VALUE LOCATION 0.0 0.0 0.0 0.0 0.0 LOADING 1 3 0 0 3 1 1 | * * * ۱* *1 ۱* DESIGN SUMMARY (KIP-FEET) *1 ۱* _____ *1 ۱* *1 RESULT/ CRITICAL COND/ RATIO/ LOADING/ *1 FX MY MZ LOCATION 1 ____ ____ _____ 1 PASS AISC- H1-2 0.886 3 Т 1 21.32 C 0.00 70.97 0.00 1 1* *1 |-----54. PERFORM ANALYSIS 55. PARAMETER 56. CODE AISC 57. TRACK 1 MEMB 1 TO 3 58. CHECK CODE MEMB 1 TO 3

Tutorial 1 1-101

LL UNI	TS 2	ARE - KIP						
ember		TABLE	RESULT/ FX	CRITICA M		RATIO/ MZ	LOADING, LOCATIO	
1	ST	W12X35	PASS 23.02 C	AISC- 0	H1-1 .00	0.855 52.01	1 15.00	
KL/R-	• Y = 3	1, UNIT K 116.7 CB= 00 FCY= 3	IP-INCH, L= 1.00 YLD= 0.00 FTY= 3	180.0 40.00 0.00 FC	AX= 10.30 ALLOWABLE = 10.94	SZ= 45.6 STRESSES: FT= 24.00	SY= 7.5 FCZ= 18.15 FV= 16.00	9
2	ST	W14X30	PASS 5.16 C	AISC- 0	H1-3 .00	0.969 66.64	3 19.00	
KL/R-	-Y= 3	2, UNIT K 153.2 CB=	IP-INCH, L= 1.00 YLD=	228.0 40.00	AX= 8.85	SZ= 42.1 STRESSES:	SY= 5.3 FCZ= 21.6	
	24.0		0.00 FTY= 3		= 6.36			
 MEM=	ST	W14X34 3, UNIT K	PASS 21.45 C 	AISC- 0 180.0	H1-2 .00 AX= 10.00	0.959 77.36 SZ= 48.6	3 0.00 SY= 6.5	 9
MEM= KL/R- FTZ=	ST 	W14X34 3, UNIT K. 117.9 CB= 00 FCY= 3 ISH	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3	AISC- 0 180.0 j 40.00 j 0.00 FC-	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00	3 0.00 SY= 6.1 FCZ= 21.9 FV= 16.00	 9 5
MEM= KL/R- FTZ=	ST -Y= : 24.0 FIN:	W14X34 3, UNIT K 117.9 CB= 00 FCY= 3 ISH *********	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 	AISC- 0 180.0 2 40.00 7 0.00 FC E STAAD.1	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00	3 0.00 SY= 6.1 FCZ= 21.91 FV= 16.00	 9 5
MEM= KL/R- FTZ=	ST -Y= : 24.0 FIN:	W14X34 3, UNIT K. 117.9 CB= 00 FCY= 30 15H ********* **** D2 ***** Fo:	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 *** END OF TH ATE= *************	AISC-0 0 180.0 2 40.00 2 0.00 FC E STAAD.1	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 Pro RUN * TIME= **********	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00	3 0.00 SY= 6.1 FCZ= 21.9! FV= 16.00	 9 5
MEM= KL/R- FTZ=	ST 	W14X34 3, UNIT K. 117.9 CB= 00 FCY= 30 15H ********* **** D2 ***** Fo:	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 *** END OF TH ATE=	AISC-0 0 180.0 2 40.00 2 0.00 FC E STAAD.1	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 Pro RUN * TIME= **********	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00	3 0.00 SY= 6.1 FCZ= 21.9! FV= 16.00	 9 5
MEM= KL/R- FTZ=	ST 	W14X34 3, UNIT K 117.9 CB= 00 FCY= 30 ISH ********* **** D2 ********* Fo: Research	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 *** END OF TH ATE= ***************** r questions Engineers O Telephon	AISC- 0 180.0 2 40.00 5 0.00 FC: E STAAD.1 ********* on STAAD ffices at e	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 Pro RUN * TIME= ********** .Pro, plea t the fol:	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 SY= 6.3 FCZ= 21.99 FV= 16.00 ******** * tions * *	 9 5
MEM= KL/R- FTZ=	ST 	W14X34 3, UNIT K: 117.9 CB= 00 FCZ= 3 00 FCZ= 3 15H ********** Fo: Research USA:	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 	AISC- 0 180.0 ; 40.00 ; 0.00 FC- E STAAD.; ; ; ; ; ; ; ; ; ; ; ; ; ; ; ; ; ; ;	H1-2 .00 AL:00 Pro RUN * TIME= ********* .Pro, plea t the fol: suppo:	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 FT= 24.00 Contact ***********************************	3 0.00 SY= 6.1 FCZ= 21.99 FV= 16.00 ********* * tions * * * * *	 9 5
MEM= KL/R- FTZ=	ST -Y=: 24.1 FIN: *** * *	W14X34 3, UNIT K: 117.9 CB= 00 FCY= 30 ISH ********** Fo: Research USA: CANADA	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 *** END OF TH ATE= ***************** r questions Engineers O Telephon	AISC- 0 180.0 2 40.00 2 0.00 FC: E STAAD.1 5 ********* on STAAD ffices a e -2500 -4771	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 SY= 6.1 FCZ= 21.99 FV= 16.00 ********* * tions * * * * *	 9 5
MEM= KL/R- FTZ=	ST -Y= : 24.0 FIN: *** * * * * * *	W14X34 3, UNIT K: 117.9 CB= 00 FCZ= 3 00 FCZ= 3 CB SCC SCC Research USA: CANADA CANADA UK	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTF= 3 	AISC- 0 180.0 j 40.00 j 0.00 FC E STAAD.1 E STAAD.1 Fices a e -2500 -4771 6087 7-000	H1-2 .00 AL:00 Pro RUN * TIME= ********* .Pro, plea t the fol: suppo: detecl staad suppo:	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 FT= 24.00 Comparison of the second sec	3 0.00 SY= 6. FCZ= 21.9 FV= 16.00 ******** tions * * tions * * * th.com * * uk *	 9 5
MEM= KL/R- FTZ=	ST 24.0 FIN: ** * * * * * *	W14X34 3, UNIT K: 117.9 CB= 00 FCY= 3 15H *********** Fo: Research USA: CANADA CANADA UK FRANCE	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 	AISC- 0 180.0 j 40.00 j 0.00 FC 0.00 J 0.00 FC 0.00 J 0.00 J 0.00 J 0.00 J 0.00 J 0.00 J 0.00 FC 0.00 J 0.00 J	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 TIME= ********** .Pro, ple: t the fol: suppo: detecl staad: suppo: suppo:	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 FCZ= 21.9 FV= 16.00 ******** tions * * tions * * h.com * * uk *	 9 5
MEM= KL/R- FTZ=	ST -Y= : 24.0 FIN: *** * * * * * *	W14X34 3, UNIT K. 117.9 CB= 00 FCY= 30 15H ***** Dj ********** Fo: Research USA: CANADA CANADA UK FRANCE GERMANY	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 	AISC- 0 180.0 2 40.00 2 0.00 FC: 	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 SY= 6. FCZ= 21.9 FV= 16.00 ******** tions * * tions * * * th.com * * uk *	 9 5
MEM= KL/R- FTZ=	ST Y=: 24.(FIN: *** * * * * * * * * *	W14X34 3, UNIT K: 117.9 CB= 00 FCY= 30 ISH ***********************************	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 *** END OF TH ATE= ******************** r questions Engineers O Telephon +1 (714)974 +1 (905)632 +1 (604)629 +44 (1454)20 +33 (0)1 645 +49/931/404 +47 67 57 2	AISC- 0 180.0 2 40.00 2 0.00 FC E STAAD.1	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 FCZ= 21.9 FVZ= 16.00 ******** tions * * tions * * uk * uk * uk * uk * uk *	 9 5
MEM= KL/R- FTZ=	ST -Y= : 24.1 FIN: *** * * * * * * * * * * *	W14X34 3, UNIT K 117.9 CB= 00 FCY= 30 15H ***** Dj ********* Research USA: CANADA CANADA USA: CANADA UK FANCE GERMANY NORWAY SINGAPORE INDIA	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 0.00 FT	AISC- 0 180.0 2 40.00 2 0.00 FC ********* on STAAD. ffices a e -2500 -4771 6087 7-000 51084 68-71 1 30 15/16 7-3575 st	H1-2 .00 AXE 10.00 ALLOWABLE = 10.72 1 TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 FCZ= 21.9 FV= 16.00 ******** tions * tions * * tions * * uk * uk * uk * uk * sa.com *	 9 5
 MEM= KL/R- FTZ= 	ST -Y= : 24.0 FIN: *** * * * * * * * * * *	W14X34 3, UNIT K 117.9 CB= 00 FCZ= 3 00 FCZ= 3 CB S S S S S S S S S S S S S S S S S S	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTH= 3 Engineers O Telephon +1 (714)974 +1 (905)632 +1 (604)629 +44(1454)20 +33(0)1 645 +49/931/404 +47 67 57 2 +65 6225-60 +91(03)235 +81(03)5952	AISC- 0 180.0 ; 40.00 ; 0.00 FC- E STAAD.; ********* on STAAD ffices a: e -2500 -4771 6087 7-000 51084 68-71 30 15/16 7-3575 st -6500	H1-2 .00 AL:00 Pro RUN * TIME= ********** t the fol: suppo: detecl staad suppo: info@: staad suppo: upport@ca eng-e	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 	3 0.00 SY= 6. FCZ= 21.9 FV= 16.00 	 9 5
 MEM= KL/R- FTZ= 	ST -Y=: 24.(FIN: ** * * * * * * * *	W14X34 3, UNIT K 117.9 CB= 00 FCY= 30 15H ***** Dj ********* Research USA: CANADA CANADA USA: CANADA UK FANCE GERMANY NORWAY SINGAPORE INDIA	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTY= 3 0.00 FT	AISC- 0 180.0 ; 40.00 ; 0.00 FC- E STAAD.; ********* on STAAD ffices a: e -2500 -4771 6087 7-000 51084 68-71 30 15/16 7-3575 st -6500	H1-2 .00 AL:00 Pro RUN * TIME= ********** t the fol: suppo: detecl staad suppo: info@: staad suppo: upport@ca eng-e	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 SY= 6. FCZ= 21.9 FV= 16.00 	 9 5
 MEM= KL/R- FTZ= 	ST -Y= : 24.0 FIN: *** * * * * * * * * * * *	W14X34 3, UNIT K 117.9 CB= 00 FCZ= 3 00 FCZ= 3 CB S S S S S S S S S S S S S S S S S S	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTZ= 3 	AISC- 0 180.0 ; 40.00 ; 0.00 FC- E STAAD.; ********* on STAAD ffices a: e -2500 -4771 6087 7-000 51084 68-71 30 15/16 7-3575 st -6500	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 1 TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 	3 0.00 SY= 6. FCZ= 21.00 FVZ= 16.00 	 9 5
 MEM= KL/R- FTZ= 	ST -Y=: 24.0 FIN: ** * * * * * * * * * * *	W14X34 3, UNIT K. 117.9 CB= 00 FCY= 3 00 FCY= 3 CB 15H ********** For Canada CANADA CANADA CANADA CANADA CANADA CANADA CANADA CANADA CANADA CHINA	PASS 21.45 C IP-INCH, L= 1.00 YLD= 0.00 FTZ= 3 	AISC- 0 180.0 ; 40.00 ; 0.00 FC- E STAAD.; ********* on STAAD ffices a: e -2500 -4771 6087 7-000 51084 68-71 30 15/16 7-3575 st -6500	H1-2 .00 AX= 10.00 ALLOWABLE = 10.72 = = 10.72 = TIME= ************************************	0.959 77.36 SZ= 48.6 STRESSES: FT= 24.00 ***********************************	3 0.00 SY= 6.1 FCZ= 21.9 FCZ= 21.9 FCZ= 16.00 ********* * tions * * tions * * * uk * uk * uk * uk * uk * uk * uk	 9 5

1.10 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the *Post Processing* Mode. The *Post Processing* mode is used to verify the analysis and design results and generate reports.

For this tutorial problem, we shall perform the following tasks:

- Display deflection diagrams
- Annotate Displacements
- Display the force and moment diagrams.
- Change the degree of freedom for which the force/moment diagram is plotted
- Annotate the force diagram
- Displaying dimensions of members

1.10.1 Going to the post-processing mode

Steps:

1. At the end of section 1.8, we saw how one could go directly from the Analysis window to the post-processing screen. However, the formal method of accessing the *Post Processing* mode is either by clicking on the *Post-Processing* icon from the top toolbar or from the **Mode** menu as shown in the figures below.

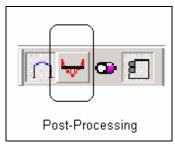


Figure 1.91

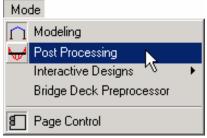


Figure 1.92

1-104 Tutorial 1

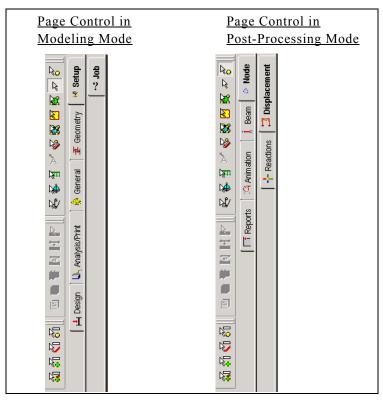
2. The *Results Setup* dialog box appears as shown below. Select the load cases for which to display the results. For this tutorial, let us **select all the load cases**. Then click on the **OK** button.

Results Setup	×
Loads Range Result View Options	,
Load Cases	
Available:	Selected:
	> 1 DEAD + LIVE 2 WIND FROM LEFT 3 75 PERCENT OF (DL+LL+WL)
	<
	OK Apply Help

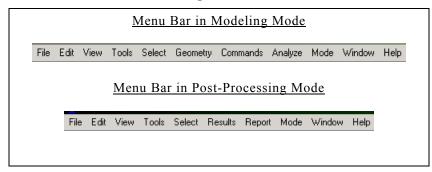
Figure 1.93

Tutorial 1 1-105

Notice that in the Post-Processing mode, the tabbed Page Control bar and the menu bar change to offer the post processing functions.







1.10.2 Annotating the displacements

Steps:

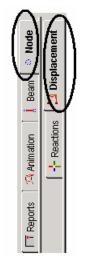
The screen will now look like the figure shown below.

	PORTAL.std Tools Select Resul	ts Report Mode Wi	ndow Help									_ 8
🖹 🚅 🖬 🖡	₽ X @ X ≤	2 ± 🗠 ± 候 🖬 🛛	1 🔝 🥕	8). 🗀 🛥 🗖	a 🎒 🗋	🗖 📥 🎒	∺ <u>*</u> †1	HH 😼 🎉	? 👱 🥂	🔄 🖉 🐔	i 🖉
ខែ៩៩ ៩	ദൈഹം	→			Q Q @ @					• 8		<u>_</u>
											<u>]], i</u>]	<u>* 2</u>
4 ᢣ 🎋 🙎	i 🗱 🎿 😰 🖪 🛙	🖥 🎝 🍜 🍜 📲	i 🛛 🛋 👌	(👭 G	► F ₂ 4 F9	🕇 🕅 🎢	🍃 🎐 🐔	9481		- 💷 🎅		
물물	PORTAL.std - W	/hole Structure	_ 🗆 X	🔳 POF	RTAL.std - No	ode Displace	ments:					
Node						ummary /						
o la la						Horizontal	Vertical	Horizontal	Resultant	Rotational		
aam o Node Displacement				Node	L/C	X	Y ft	Z	ft	rX rad	r¥ rad	rZ rad
<u> </u>				1	1 DEAD + LIV	0.000	0.000	0.000	0.000	0.000	0.000	0.000
					2 WIND FROM	0.000	0.000	0.000	0.000	0.000	0.000	0.000
					3 75 PERCEN	0.000	0.000	0.000	0.000	0.000	0.000	0.000
Animation					1 DEAD + LIV	0.013	-0.001	0.000	0.013	0.000	0.000	-0.005
					2 WIND FROM	0.061	0.000	0.000	0.061	0.000	0.000	-0.003
			-11		3 75 PERCEN	0.056	-0.001	0.000	0.056	0.000	0.000	-0.006
					1 DEAD + LIV 2 WIND FROM	0.013	-0.001	0.000	0.013	0.000	0.000	0.004
					3 75 PERCEN	0.061	-0.000	0.000	0.061	0.000	0.000	-0.001
					1 DEAD + LIV	0.000	0.001	0.000	0.000	0.000	0.000	-0.002
Reports					2 WIND FROM	0.000	0.000	0.000	0.000	0.000	0.000	-0.005
æ					3 75 PERCEN	0.000	0.000	0.000	0.000	0.000	0.000	-0.005
										A		
				🗰 PORTAL.std - Beam Relative Displacement Detail:								_0
				All Relative Displacement / Max Relative Displacements /								
						Diet			z	Resultant		
			6	Beam	L/C	ft	ft	у ft	ft	ft		
			-	1	1 DEAD + LIV	0.000	0.000	0.000	0.000	0.000		
						3.750	-0.005	-0.000	0.000	0.005		
						7.500	-0.009	-0.000	0.000	0.009		
						11.250	-0.009	0.000	0.000	0.009		
						15.000	0.000	0.000	0.000	0.000		
					2 WIND FROM	0.000	0.000	0.000	0.000	0.000		
						3.750	-0.008	0.000	0.000	0.008		
						7.500	-0.005	0.000	0.000	0.005		
						11.250	-0.001	0.000	0.000	0.001		
					3 75 PERCEN	15.000	0.000	0.000	0.000	0.000		
	Y				575 FERCEN	3.750	-0.009	-0.000	0.000	0.000		
						7.500	-0.003	-0.000	0.000	0.003		
	5.											
	'z	Load 1 : Displa	cement			11 250	-0.007	-0.000	0.000	0.007		

Figure 1.96

The diagram currently on display is the node deflection diagram for load case 1 (DEAD + LIVE). The title at the bottom of the diagram is indicative of that aspect. If you, say, wandered off into any other result diagram, and wanted to get back to the deflection diagram, just select the *Node* | *Displacement* tab along the page control area on the left side.

Tutorial 1 1-107





Annotation is the process of displaying the displacement values on the screen. Select the **View Value** option from the **Results** menu.



Figure 1.98

1-108 Tutorial 1

The following dialog box comes up. From the **Ranges** tab, select **All** nodes. If you wish to annotate deflection for just a few nodes, specify the node numbers in the node list.

88 Annotation	- PORTAL.std - Whole Structure
Ranges Bea	m Results Node Reactions
O None	
• All	
C View	V
C Group	
C Property	<u>v</u>
C Ranges	Nodes:
	Beams:
	Remove All Annotate Close

Figure 1.99

We will annotate the results for all the nodes. So, keep the button on **All**.

From the **Node** tab, check the **Resultant** option. Resultant stands for the square root of sum of squares of values of X,Y and Z displacements. Click the **Annotate** button and notice that the values appear on the structure. Click the **Close** button to close the dialog box.

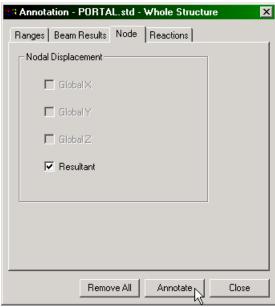


Figure 1. 100

The following figure shows the annotated deflection diagram for load case 1.

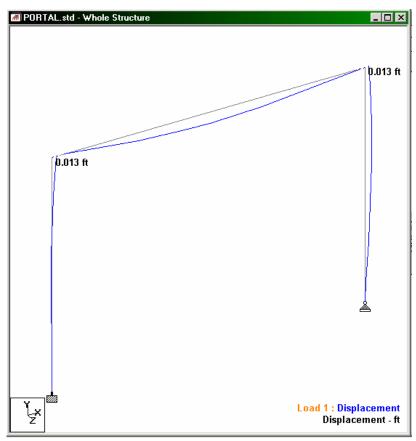


Figure 1. 101

1.10.3 Displaying force/moment diagrams

Steps:

The simplest method to access the facilities for displaying force/moment diagrams is from the **Beam** | **Forces** page along the page control area on the left side of the screen. The bending moment MZ will be plotted by default, evidence of which can be found in the form of the Mz icon show in the diagram below which becomes active.

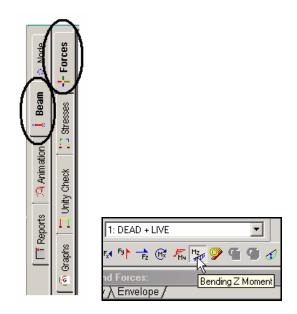


Figure 1. 102

1-112 Tutorial 1

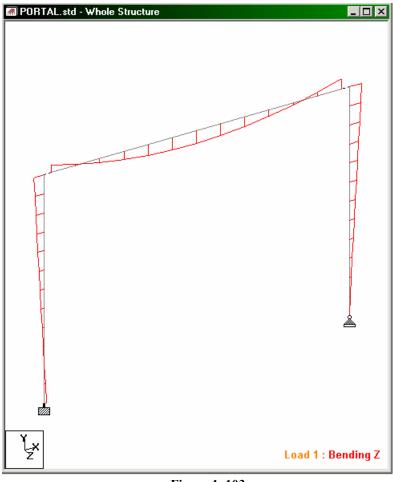


Figure 1. 103

The option for selecting the forces/moment diagram is available from another facility also - the *Results* | *Bending Moment* menu option.



Figure 1. 104

1.10.4 Annotating the force/moment diagram

Steps:

Annotation is the process of displaying the force/moment values on the screen. Select the **View Value** option from the **Results** menu.



In the dialog box that comes up, click on the **Ranges** tab and select **All** members. If you wish to annotate the force/moment for just a few members, specify the beam numbers in the beam list.

	- PORTAL.std - Whole Structure
nariyes Bea	m Results Node Reactions
C None	
• All	
C View	
C Group	
C Property	
C Ranges	Nodes:
	Beams:
	,
	Remove All Annotate Close

Tutorial 1 1-115

Figure 1. 106

We will annotate the results for all members. So, keep the button on **All**.

From the **Beam Results** tab, check the **Ends** and **Mid Point** options under the **Bending** section. Click the **Annotate** button and notice that the values appear on the structure. Click the **Close** button to close the dialog box.

Structure - PORTAL.std - Whole Structure	×
Ranges Beam Results Node Reactions	
Bending F Ends F Maximum F Mid point	
Shear Ends Maximum Mid point	
Axial]
Displacement	
Max Resultant	
Combined Bending and Axial Stress	
Ends Maximum Mid point	
Remove All Annotate Clo	se

Figure 1. 107

1-116 Tutorial 1

The following figure shows the annotated MZ diagram for load case 2.

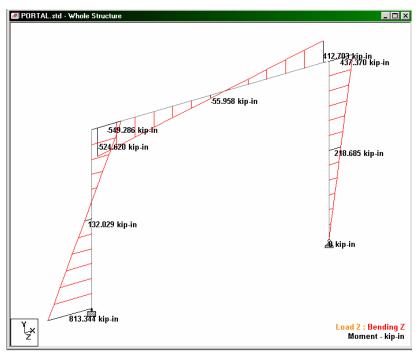


Figure 1. 108

1.10.5 Changing the degree of freedom for which forces diagram is plotted

Force and moment diagrams can be plotted for 6 degrees of freedom – Axial, Shear-Y, Shear-Z, Torsion, Moment-Y, Moment-Z. One may select or de-select one of more of these degrees of freedom from View | Structure Diagrams | Loads and Results. Let us select Shear yy and select load case 3 (75 PERCENT OF [DL+LL+WL] as shown below.

iagram s					
Force Limits	Animation		Design Results		
Structure	Loads and Results		Scales	Lab	els
Load Case: 3: 75 PEF	CENT OF [DL + LL +)	wL]			
Loads Deflection Beam Forces Diagram Image: Loads One Image: Loads Moment Image: Construction Image: Construction Image: Show Load Arrow One Outline Mode Shape Image: Construction Image: Construction					
Mode Shape: Non	e 🗾 🗖	Mode S	Shape 📃		
Beam Forces	т		From	То	
	' 		< -24.4848	-24.4848	-
C Diagram C Colo	or C/T 🗧	-	24.4848	-21.4242	
Intensity		-	21.4242	-18.3636	
Minimum: -24.48	40 4	-	18.3636	-15.303	
Minimum. J ^{24,40}	40 5	-	15.303	-12.2424	
Maximum: 24.484	18 6	-	12.2424	-9.18181	
	7	-	9.18181	-6.12121	
No of values: 16			6.12121	-3.0606	
	9	-	3.0606	0	
)	3.0606	
🔽 Shearyy 📃 🗌	Bending yy 📘 1		3.0606	6.12121	
📃 🗖 Shear zz 🛛 📕 🗖	Bending zz 📕 🧕		5.12121	9.18181	
Torsion	- 1	-	9.18181	12.2424	
		4 1	2.2424	15.303	
	OK Ca	ncel	Apply	+	łelp

Figure 1. 109

The resulting figure is shown below.

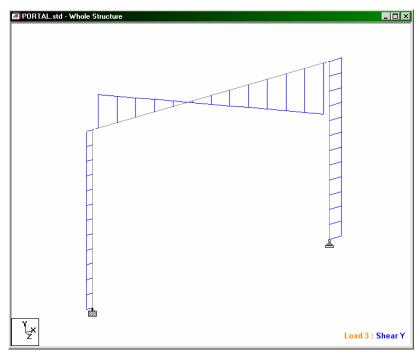


Figure 1. 110

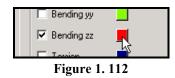
All degrees of freedom currently plotted will be indicated with a tick mark in the *Diagrams* dialog box.

The icons of the Results toolbar may also be used to turn on/off specific degrees of freedom.



Figure 1. 111

For the sake of easy identification, each degree of freedom (d.o.f) has been assigned a different color (see Diagrams dialog box shown above). One may change the color for that d.o.f. by clicking on the color button alongside the d.o.f, and make a new choice from the color palette.



The appearance of the diagram may also be set to one of the 3 - Hatch, Fill or Outline by turning on the relevant option in the dialog box shown earlier.

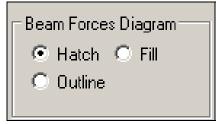


Figure 1. 113

1.10.6 Displaying the dimensions of the members

To display the dimension of the members, click on the **Dimension** icon. Alternatively, one may select the *Dimension Beams* option from the *Tools* menu. In the dialog box that opens, the option *Dimension to View* is active. Click on the **Display** button followed by the **Close** button, and the dimensions of the members will appear alongside the members.

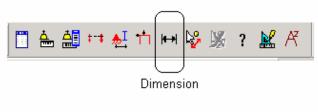


Figure 1. 114

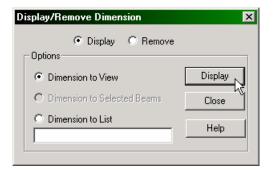


Figure 1. 115

The diagram will look like the one shown below.

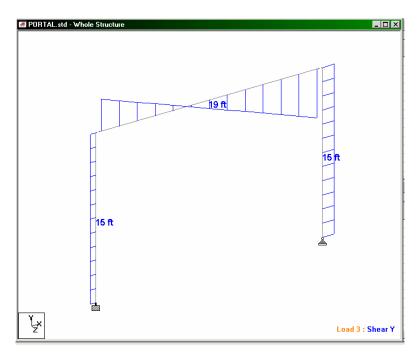


Figure 1. 116

We can opt to save the screenshot by clicking on the **Take Picture** icon (shown below). This picture may be included in custom reports. See Chapter 2 for a tutorial on taking pictures as well as generating custom reports.



Figure 1. 117

1-122 Tutorial 1

For obtaining a quick print of the plot on the screen, select the **Print Current View** icon as shown below.



Figure 1. 118

For detailed information on the Post Processing features, please refer to the *Post Processing* section in the STAAD.Pro Graphical Environment manual.

N o t e s 1-124

N o t e s

Tutorial Problem 2: RC Framed Structure

This tutorial provides step-by-step instructions for creating the model of a reinforced concrete framed structure using STAAD.Pro. The following topics are covered :

- Starting the Program
- Creating Joints and Members
- Switching On Node and Beam Labels
- Specifying Member Properties and Material Constants
- Specifying Member Orientation using Beta Angle
- Specifying Supports
- Specifying Loads
- Specifying the Analysis Type
- Specifying the parameters and instructions for Concrete Design
- Performing Analysis and Design
- Viewing results using the Output File
- Viewing results on screen both graphically and numerically
- Producing customized reports

Section 2

2.1 Methods of creating the model

As explained in Section 1.1 of tutorial problem 1, there are two methods of creating the structure data:

- a) using the graphical model generation mode, or graphical user interface (GUI) as it is usually referred to.
- b) using the command file.

Both methods are explained in this tutorial also. The graphical method is explained first, from Section 2.2 onwards. Section 2.8 describes the process of creating the model using the command file method and the STAAD.Pro text editor.

2.2 Description of the tutorial problem

The structure for this project is a 2 bay, 2 story reinforced concrete frame. The figure below shows the structure. Our goal is to create the model, assign all required input, and perform the analysis and concrete design.

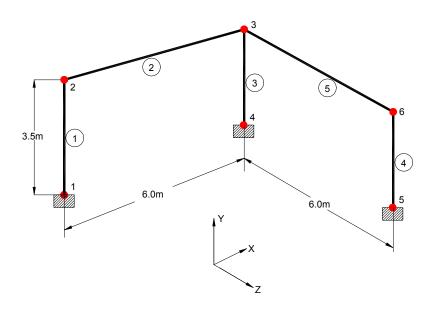


Figure 2.1

BASIC DATA FOR THE STRUCTURE

ATTRIBUTE	DATA	
Member properties	Beams 2 & 5 : Rectangular, 275 mm width X 350 mm depth	
	Columns 1 & 4 : Rectangular, 275 mm width X 300 mm depth	
	Column 3 : Circular, 350 mm diameter	
Member Orientation	All members except column 4 : Default	
	Column 4 : Rotated by 90 degrees with respect to default condition	
Material Constants	Modulus of Elasticity : 22 KN/sq.mm	
	Density : 25 kn/cu.m	
	Poisson's Ratio : 0.17	
Supports	Base of all columns : Fixed	
Loads	Load case 1 : Dead Load Selfweight of the structure. Beams 2 & 5 : 400 kg/m in global Y downward	
	Load case 2 : Live Load Beams 2 & 5 : 600 kg/m in global Y downward	
	Load case 3 : Wind Load Beam 1 : 300 kg/m along positive global X Beam 4 : 500 kg/m along positive global X	
	Load Case 4 : DEAD + LIVE L1 X 1.2 + L2 X 1.5 (Use REPEAT LOAD, not Load Combination)	
	Load Case 5 : DEAD + WIND L1 X 1.1 + L2 X 1.3 (Use REPEAT LOAD, not Load Combination)	

ATTRIBUTE	DATA
Analysis Type	PDELTA
Concrete Design	Consider load cases 4 and 5 only. Parameters: Ultimate Strength of Steel : 415 N/sq.mm Concrete Strength : 25 N/sq.mm
	Clear cover for top : 25 mm Clear cover for bottom : 30 mm Clear cover for side : 25 mm
	Design beams 2 and 5 Design columns 1, 3 and 4

2.3 Starting the program

Select the **STAAD.Pro** icon from the *STAAD.Pro* 2005 program group.

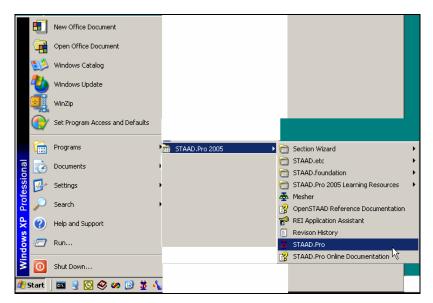


Figure 2.2

The STAAD.Pro Graphical Environment will be invoked and the following screen comes up.

STAAD.Pro				_ 8 ×
"∎ ≇ ≝ ≣ № % ඬ X Ω ± Ω ± ๙ ≣ ≣ ≸		日本部中和中国家	· M 저 [] 관광영관 [] 이 국 [] 프 []	11
₽₽₽₽₽₽₽₽ ₽₽₽₽₽			• ?	
	かたまの 一人 開きません	6 4 3 9 4 4 0 8 🗌		
Rea .				
			×	
	New File Recent Files			
14	Space	File Name :		
λ Ign	Plane Floor Truss	Structure1		
100 No. 100 No	-	C:\test\		
*		C. 464.1		
	A SPACE structure, which is a three-dimension any plane, is the most general.	nal framed structure with loads applied in		
E Contraction of the second se	1			
	C Inch C Decimeter	Force Units C Pound C Newton		
	Foot C Meter Milimeter C Kilometer	KiloPound C DecaNewton Kilogram C KiloNewton		
	C Centimeter	C Metric Ton C MegaNewton		
· · · · · · · · · · · · · · · · · · ·				
	Display this dialog box at the Startup			
	< Back	Next > Cancel Help		
			_	

Figure 2.3

This *New* dialog box will come up every time we start the program. To turn this feature off, simply uncheck the *Display this dialog box* at *Startup* box at the lower left hand corner. This feature can be turned on again at a later time when $File \mid New$ is invoked from the main menu.

Note about the unit system :

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, Aluminum – selected from the program's library (Please refer to Section 5 of the STAAD.Pro 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.

The place from where we can change this setting is under the *File* | *Configure* menu. To get to that option, first close down the dialog box shown in the earlier figure by clicking on **Cancel**. Then, click on the **File** | **Configure** menu option (see figure below) and choose the appropriate unit system you want. For this tutorial, let us choose the **Metric** units (KN, Meter, etc.).

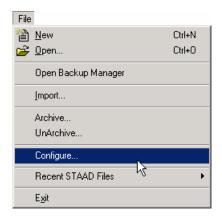


Figure 2.4

					Tutorial 2	1
onfigure P	rogram			×	1	
	e Options nput File Format		AD Default Des Error File	5		
Base Unit	Background			Analysis Engine		
Current De		Metric				
- Select E	lase Unit:					
	1etric		T			
<u> </u>						
	Accept	Cancel	1 Applu	Hala		
	Accept	Lancel	Apply	Help		

Figure 2.5

Click on the Accept button to close the above dialog box.

Following this, select File | New once again.

File	
웥 <u>N</u> ew	Ctrl+N
궏 <u>O</u> pen 场	Ctrl+O
Open Backup Manager	
Import	
Archive	
UnArchive	
Configure	
Recent STAAD Files	•
E <u>x</u> it	

Figure 2.6

The dialog box shown in Figure 2.3 will re-appear.

2.4 Creating a new structure

1. In the *New* dialog box, we provide some crucial initial data necessary for building the model.

The structure type is to be defined by choosing from among *Space*, *Plane*, *Floor* and *Truss*. A *Space* type is one where the structure, the loading or both, cause the structure to deform in all 3 global axes (X, Y and Z). In a *Plane* type, the geometry, loading and deformation are restricted to the global X-Y plane only. A *Floor* type is a structure whose geometry is confined to the X-Z plane. A *Truss* type of structure carries loading by pure axial action. Truss members are deemed incapable of carrying shear, bending and torsion. For our model, let us choose **Space**.

We choose **Meter** as the length unit and **Kilo Newton** as the force unit in which we will start to build the model. The units can be changed later if necessary, at any stage of the model creation.

We also need to provide a name in the *File Name* edit box. This is the name under which the structure data will be saved on the computer hard disk. The name "Structure?" (? will be a number) is recommended by the program by default, but we can change it to any name we want. Let us choose the name **rcframe**.

A default path name - the location on the computer drive where the file will be saved – is provided by the program under *Location*. If you wish to save the file in a different location, type in the name, or click the $\boxed{}$ button and specify the desired path.

After specifying the above input, click on the Next button.

New	×
✓ Space File Name : □ Plane	
Floor rcframe	
Location :	
C:\test\	
A SPACE structure, which is a three-dimensional framed structure with load any plane, is the most general.	ls applied in
any plane, is the most general.	
Length Units	
Ô Inch O Decimeter O Pound O Ne	
	caNewton Newton
	gaNewton
	gaivewion
Display this dialog box at the Startup	
< Back Next > Cancel	Help
<u>~</u>	

Figure 2.7

2. In the next dialog box, we choose the tools to be used to initially construct the model. Add Beams, Add Plates or Add Solids are, respectively, the starting points for constructing beams, plates or solids. Open Structure Wizard provides access to a library of structural templates which the program comes equipped with. Those template models can be extracted and modified parametrically to arrive at our model geometry or some of its parts. If the model is to be created initially using the STAAD command language, the Open STAAD Editor box can take us to the STAAD editor. Please remember that all these options are also available from the menus and dialog boxes of the GUI, even after we dismiss this dialog box.

Note: If you wish to use the Editor to create the model, choose Open STAAD Editor, click Finish, and proceed to Section 2.8.

For our model, let us check the **Add Beam** option. Click on the **Finish** button. The dialog box will be dismissed and the STAAD.Pro graphical environment will be displayed.

Where do you want to go ?		×
Add Beam Add Plate Add Solid Open Structure Wizard Open STAAD Editor Edit Job Information		
Begin building your model by creating new join drawing tools and spreadsheets.	nts and beams using the construction grid,	
Display this dialog box at the Startup < Back	Finish Cancel He	elp

Figure 2.8

2.5 Elements of the STAAD.Pro screen

The STAAD.Pro main window is the primary screen from where the model generation process takes place. It is important to familiarize ourselves with the components of that window before we embark on creating the RC Frame. Section 1.5 in tutorial problem 1 of this manual explains the components of that window in detail.

2.6 Building the STAAD.Pro model

We are now ready to start building the model geometry. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

2.6.1 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc. From the standpoint of the STAAD command file, the commands to be generated for the structure shown in section 2.2 are :

```
JOINT COORDINATES
1 0.0 0.0 0.0 ; 2 0.0 3.5 0.0
3 6.0 3.5 0.0 ; 4 6.0 0.0 0.0
5 6.0 0.0 6.0 ; 6 6.0 3.5 6.0
MEMBER INCIDENCE
1 1 2 ; 2 2 3 ; 3 3 4 ; 4 5 6 ; 5 3 6
```

Steps:

 We selected the *Add Beam* option earlier to enable us to add beams and columns to create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area.

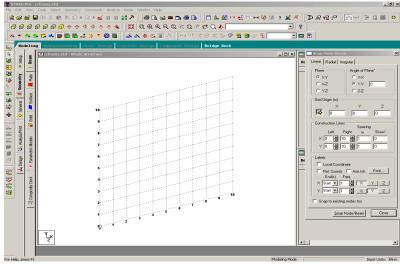


Figure 2.9

2. A Snap Node/Beam dialog box also appears in the data area on the right side of the screen. In our structure, the segment consisting of members 1 to 3, and nodes 1 to 4, happens to lie in the X-Y plane. So, in this dialog box, let us keep X-Y as the Plane of the grid. The size of the model that can be drawn at any time is controlled by the number of Construction Lines to the left and right of the origin of axes, and the Spacing between adjacent construction lines. By setting 12 as the number of lines to the right of the origin along X, 7 above the origin along Y, and a spacing of 0.5 meter between lines along both X and Y (see figure below) we can draw a frame 6m X 3.5m, adequate for our segment. Please note that these settings are only a starting grid setting, to enable us to start drawing the structure, and they do not restrict our overall model to those limits.

🗱 Snap Node/Bear	n 🛛 🗙			
Linear Radial Irregular				
Plane	Angle of Plane*			
⊙ X·Y	C XX			
C x-z	• Y-Y 0			
C Y-Z	O Z-Z			
Grid Origin (m)				
× –	Y Z			
<u>№</u> <u>0</u>	0 0			
- Construction Lines-				
Left Ri	Spacing Left Right m Skew*			
× 0 4 12 0.5 0				
Y 0 🗣 7 🗣 0.5 0				
Labels	-1-			
	Axis Ids Font			
End(s) Freq.				
X Start 💌 1 🚔 X Y Z				
Y Start 💌 1 🗮 X Y Z				
Snap to existing	nodes too			
<u>S</u> nap N	Node/Beam Close			

Figure 2.10

2-18 Tutorial 2

3. To start creating the nodes, let us first activate the **Snap Node/Beam** button by clicking on it. Then, with the help of the mouse, click at the origin (0, 0) to create the first node.

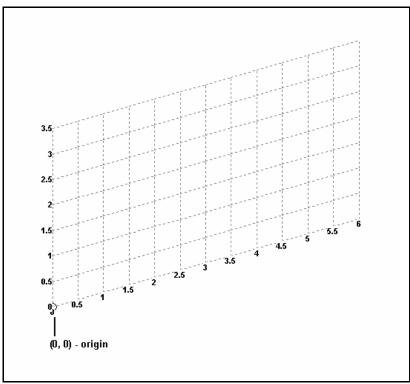


Figure 2.11

4. In a similar fashion, click on the following points to create nodes and automatically join successive nodes by beam members.

(0, 3.5), (6, 3.5) and (6, 0)

The exact location of the mouse arrow can be monitored on the status bar located at the bottom of the window where the X, Y, and Z coordinates of the current cursor position are continuously updated.

When steps 1 to 4 are completed, the frame will be displayed in the drawing area as shown below.

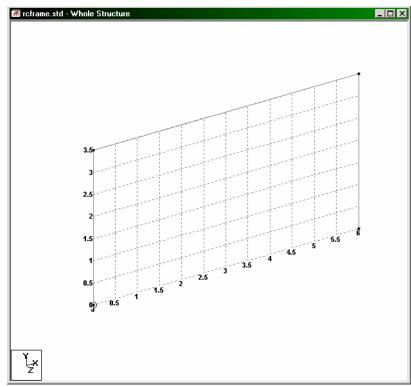


Figure 2.12

2-20 Tutorial 2

5. At this point, let us remove the grid display from the structure. To do that, click on the **Close** button in the *Snap Node/Beam* dialog box.

📲 Snap Node/Beam 🛛 🗙	1
Linear Radial Irregular	
Plane Angle of Plane*	
© XY © XX	
O X-Z O Y-Y D	
O Y-Z O Z-Z	
Grid Origin (m)	
X Y Z	
Construction Lines	
Spacing Left Right m Skew°	
× 0 🖨 12 🖨 0.5 0	
Y 0 🖨 7 🖨 0.5 0	
End(s) Freq.	
X Start 🔻 1 📮 X Y Z	
Y Start 💌 1 🗣 X Y Z	
Snap to existing nodes too	
Snap Node/Beam Close	D
	1

Figure 2.13

The grid will now be removed and the structure in the main window should resemble the figure shown below.

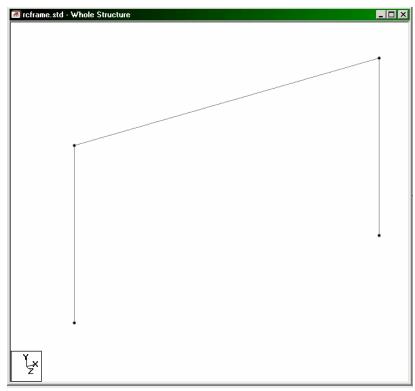


Figure 2.14

6. It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events. To save the file, pull down the **File** menu and select the **Save** command.

Switching on node and beam labels

7. Node and beam labels are a way of identifying the entities we have drawn on the screen. In order to display the *node* and *beam numbers*, right click anywhere in the drawing area. In the pop-up menu that appears, choose **Labels**. Alternatively, one may access this option by selecting the *View* menu followed by the *Structure Diagrams* option from the top menu bar, and the *Labels* tab of the dialog box that comes up.

View				
Zoom	•			
🛫 Pan				
View Selected Objects Only				
Always <u>F</u> it in Current Window				
Tables		×	Cuţ	Ctrl+X
🚼 Whole Structure		Be	Copy	Ctrl+C
Orientation	F4	e	Paste	Ctrl+V
			Select Cursor	+
Structure Diagrams			Selection Mode	+
Open View		-	Ta <u>k</u> e Picture	
New View		1	Add <u>B</u> eam	
View <u>M</u> anagement	•		Tables	
Toolbars				
Option <u>s</u>			<u>O</u> pen View	
Set Structure <u>C</u> olors			Labels	
			Orientation	F4
Structural Tool Tip Options			Structure <u>D</u> iagrams	
👃 3D Rendering		ę	Model View Details	
Refresh	F5	1	3D Rendering	

Figure 2.15

8. In the *Diagrams* dialog box that appears, turn the **Node Numbers** and **Beam Numbers** on and then click on **OK**.

Diagrams		×
Force Limits Structure	Animation	Design Results Scales Labels
Nodes Node Numbers (N) Node Points (K) Supports (S) Dimension (D)	Properties C References (J) C Sections (X) None	² hysical Members Member Number (^M) Design Group (^G) Design Envelope (^E) Design Brief (^B)
Beams Beam Numbers (B) Beam Orientation (O) Beam Spec (A) Releases (R)	Loading Display Options Load Values (V) Display Floor Loading (f Display Floor Load Distr Display Wind Load Cor Display Wind Load (W)	ibution (Y)
Beam Ends (E) Start Color End Color Plates	General Show Axes Window (^A) Show Axes At Org (I) Material (M) Show Diagram Info (G)	Master Slave (L)
 Plate Numbers (P) Plate Orientation (T) 	Surfaces Surface Numbers (^P)	Surface Orientation (^T)
Solids Solid Numbers (C)	Always Use STAAD.Pro Always Use Current Lat	bel Settings.
For quick ac	ode (Use labeling cursors to 0) ccess to the labels using keybo s Shift + the letter shown in bra	ard hotkeys,
	OK Cancel	Apply Help

Figure 2.16

The following figure illustrates the node and beam numbers displayed on the structure. The structure in the main window should resemble the figure shown below.

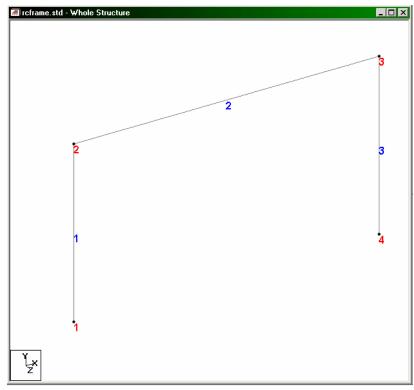


Figure 2.17

If you are feeling adventurous, here is a small exercise for you. Change the font of the node/beam labels by going to the *View* menu and selecting the *Options* command, and then selecting the appropriate tab (Node Labels / Beam labels) from the *Options* dialog box. 9. Examining the structure shown in section 2.2 of this tutorial, it can be seen that members 4 and 5 can be easily generated if we could first create a copy of members 1 and 2 and then rotate those copied units about a vertical line passing through the point (6, 0, 0, that is, node 4) by 90 degrees. Fortunately, such a facility does exist which can be executed in a single step. It is called *Circular Repeat* and is available under the *Geometry* menu.

First, select members 1 and 2 using the *Beams Cursor* . (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting members.)

10. Then, either click on the *Circular Repeat* icon from the appropriate toolbar, or, go to the **Geometry** | **Circular Repeat** menu option as shown below.



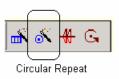


Figure 2. 18

2-26 Tutorial 2

11. In the 3D Circular dialog box that comes up, specify the Axis of Rotation as Y, Total Angle as 90 degrees, No. of Steps as 1 and the vertical line as passing through Node 4. Instead of specifying as passing through Node 4, one may also specify the X and Z co-ordinates as 6 and 0 respectively. Leave the Link Steps box unchecked and click on the OK button.

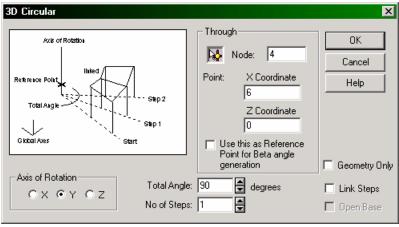


Figure 2.19

Tutorial 2 2-27

After completing the circular repeat procedure, the model will look as shown below.

Before

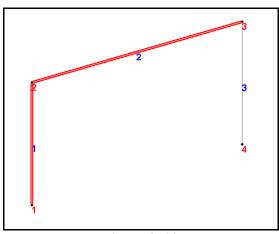


Figure 2.20



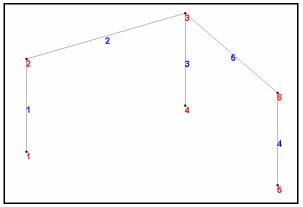


Figure 2.21

If any of the members are presently selected, let us unselect them by clicking anywhere else in the drawing area. Then, let us once again save the work by pulling down the **File** menu and selecting the **Save** command.

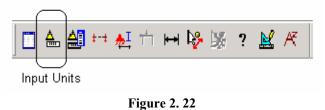
2.6.2 Changing the input units of length

As a matter of convenience, for specifying member properties for our structure, it is simpler if our length units are *millimeter* instead of *meter*. This will require changing the current length units of input. The commands to be generated are:

UNIT MMS KN

Steps:

1. Click on the **Input Units** icon from the appropriate toolbar.



Alternatively, one may select the *Tools* | *Set Current Input Unit* menu option as shown in the next figure.

Tutorial 2 2-29

Too	ls Select	Geometry	Commands	Analyze	Mode	Window
	Check Mul	tiple Structu	res			
	Check Dup	olicate				
	Orphan No	ides				+
	Check for V	Warped Plat	es			
	Check Zer	o Length Me	embers			
	Check Ove	erlapping Col	llinear Membe	rs		
	Check Ove	erlapping Pla	tes			
	Check Bea	am Plate Cor	nectivity			
	Check for S	Bolids with N	legative Volur	ne (Jacobi	an)	
	Check for V	Warped Soli	ds			
	Redefine I	ncidence				
	Calculator.					
	Unit Conve	erter				
↦	Dimension	Beams				
12	Display No	de to Node I	Distance			
1	Remove N	ode Dimensi	ion			
슲	Set Curren	t Input Unit				
<u> </u>	Set Curren	t Display Uni	it 115			
* -+	Cut Section	n				
	Query					•
		F				

Figure 2.23

2. In either case, the following dialog box comes up. Set the *Length Units* to **Millimeter** and click on the **OK** button.

S	et Current Inpu	t Units			×
	- Length Units			- Force Units	
	C Inch	C Decimeter		C Pound	C Newton
	C Foot	C Meter		C KiloPound	C DecaNewton
	 Millimeter 	C Kilometer		O Metric Ton	KiloNewton
	C Centimeter			🔿 Kilogram	C MegaNewton
		ОК	Ź	Cancel	

Figure 2.24

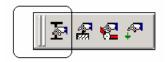
2.6.3 Specifying member properties

Our next task is to assign cross section properties for the beams and columns (see section 2.2). For those of us curious to know the equivalent commands in the STAAD command file, they are :

> MEMB PROP 1 4 PRIS YD 300 ZD 275 2 5 PRIS YD 350 ZD 275 3 PRIS YD 350

Steps:

1. Click on the **Property Page** icon located on the Structure Tools toolbar.



Property Page

Figure 2.25

Tutorial 2-31

Alternatively, one may go to the *General* | *Property* page from the left side of the screen as shown below.

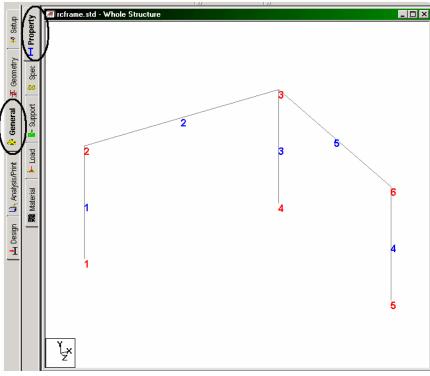


Figure 2.26

2-32 Tutorial 2

2. In either case, the *Properties* dialog box comes up. The property type we wish to assign is called PRISMATIC, and is available under the **Define** button in the *Properties* dialog box as shown below.

Properties - Wh	ole Structure	×
Section Beta Angle		
Ref Section	Material	
I I Highlight Assigne	d Goometru	
I♥ Highlight Assigne	Edit	Delete
Values	Section Database	Define
Materials	Thickness	User Table
Assignment Method		
 C Assign To Select C Assign To Edit L 		se Cursor To Assign ssign To View
	Assign Close	Help
	Assign Close	

Figure 2. 27

3. In the dialog box that comes up, select the **Rectangle** tab. Notice that the field called *Material* is presently on the <u>checked</u> mode. If we keep it that way, the material properties of concrete (E, Poisson, Density, Alpha, etc.) will be assigned along with the cross-section name. The material property values so assigned will be the program defaults. We do not want default values, instead we will assign our own values later on. Consequently, let us <u>uncheck</u> the **Material** box. Then, enter the following values:

$$YD = 300 mm$$
 $ZD = 275 mm$

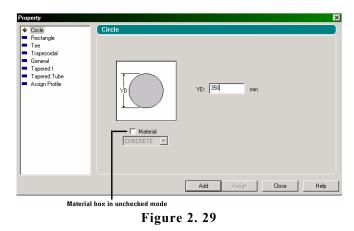
Finally, click on the Add button as shown below.

Property	x	I
Circle Rectangle Tee Trapezoidal General Tapered I Tapered Tube Assign Profile	YD: 300 mm YD: 2D: 275 mm Material Uncheck this Uncheck this	
	Add Assign Close Help	

Figure 2.28

4. To create the second member property (PRIS YD 350 ZD 275), provide **350** for **YD** and **275** for **ZD** (instead of 300 and 275) and click on the **Add** button.

To create the third member property, in the *Property* dialog box, select the **Circle** option. Specify the diameter (YD) as **350 mm**. Once again, <u>uncheck</u> the **Material** box and click on the **Add** button.



Now that we have finished creating the member properties, let us **Close** this dialog box.

2-34 Tutorial 2

The next step is to assign these member properties in the following manner:

```
Rect 0.30x0.28 – members 1 and 4
Rect 0.35x0.28 – members 2 and 5
Cir 0.35 – member 3
```

To assign the member properties, follow these steps:

- a. Select the first property reference in the *Properties* dialog box (Rect 0.30x28).
- b. Make sure that the "Use Cursor to Assign" button is selected under the Assignment Method box.
- c. Click on the Assign button. The cursor changes to
- d. Using the cursor, click on members 1 and 4.
- e. Finally, click on the Assign button again, or type the 'Esc' button on your keyboard to stop the assignment process.

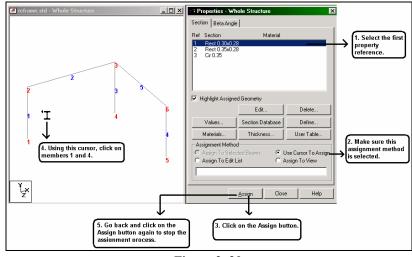


Figure 2.30

In a similar fashion, assign the remaining properties.

Tutorial 2-35

After all the member properties have been assigned, the model will look as shown below.

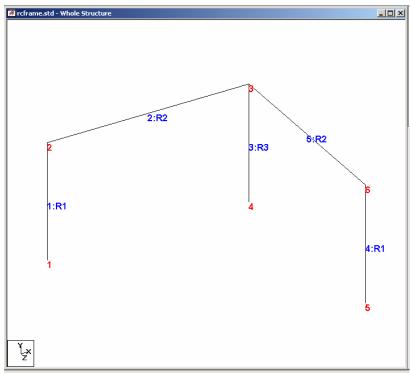


Figure 2.31

We ought to save the model once again by pulling down the **File** menu and selecting the **Save** command.

2.6.4 Specifying geometric constants

In the absence of any explicit instructions, STAAD will orient the beams and columns of the structure in a pre-defined way. Orientation refers to the directions along which the width and depth of the cross section are aligned with respect to the global axis system. The rules which dictate this default orientation are explained in Section 1 of the STAAD.Pro Technical Reference Manual.

We wish to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis. This requires applying a beta angle of 90 degrees. The command which needs to be generated is :

BETA 90 MEMB 4

Steps:

- 1. Select the **Beta Angle** tab in the *Properties* dialog box.
- In the dialog box, specify the angle as 90 degrees. Select member
 4 using the Beams Cursor

Notice that as we select the member, the Assignment Method automatically sets to Assign to Selected Beams. Click on the Assign button.

Click anywhere in the drawing area to un-highlight the member.

Properties - Whole Structure	×
Section Beta Angle	
Beta Angle: 30	
Assignment Method • Assign To Selected Beams • Assign To Edit List 3	ŋn
Assign Close Help	

Tutorial 2-37

Figure 2.32

(An alternative method to assign beta angles is the following. First select the member for which you wish to assign the beta angle. Then, go to the *Commands* | *Geometric Constants* | *Beta Angle* menu option. Specify the *Angle in Degrees* to be 90, ensure that the assignment method is "*To Selection*" and click on *OK*.)

		beta Angle	<u> </u>
Commands Analyze Mode Wind Plate Thickness Surface Thickness Member Property	dow Help	Angle Angle in Degrees 90 Angle RAngle	
Material Constants	🕨 a 511 🕂 🔊 🖉 🖉 🖉 🚱 🚱 🚱		
Geometric Constants	 Beta Angle 	Assign	
Support Specifications	Member Reference Point Plate Reference Point	C To View C To Selection	
Member Specfications	Flate meterense Folit		
Plate Element Specifications	 Clear Above Commands 	OK Cancel I	Help
Master/Slave Specification	▶ <u></u>		

Figure 2.33

One may view the orientation of the member local axes by going to the *View* | *Structure diagrams* | *Labels* menu option and switching on *Beam Orientation*.

2.6.5 Specifying material constants

At the time of assigning member properties, we deliberately chose not to assign the material constants simultaneously, since we wanted to specify values which are different from the built-in defaults. The desired values are listed at the beginning of this tutorial. The corresponding commands we wish to generate in the STAAD input file are:

> CONSTANTS E 22 ALL UNIT METER DENSITY 25.0 ALL POISSON 0.17 ALL

Steps:

1. From the **Commands** menu, select **Material Constants**. To define the Modulus of Elasticity, select the **Elasticity** option as shown below.

Commands Analyze Mode Wind	dow_Help
Plate Thickness Surface Thickness	🖻 🏟 📑 🏐 🗋 📗 🗂 📥
Member Property) ବ୍ର୍ଷ୍ <i>ଷ୍</i> ପ୍ 🖻 📌 Q
Material Constants	Material Table
Geometric Constants	► Density
Support Specifications	Elasticity
Member Specfications Plate Element Specifications	 Poisson's Ratio Alpha
Master/Slave Specification	Damping Ratio
Pre Analysis Print	 G (Shear Modulus)
№ Define Damping for Dynamics	Clear Above Commands
Loading	•
Analysis	•
Post-Analysis Print	•
Design	•
Miscellaneous	•

Figure 2.34

2. In the *Material Constant* dialog box that appears, enter **22** in the *Enter Value* box. Since the value has to be assigned to all the members of the structure, the current setting of the assignment method, namely, **To View**, allows us to achieve this easily. Then, click on **OK**.

Material Constant - Elasticity	×
Material Constant	
C Aluminum	
C Concrete	
C Steel	
Enter Value 22 kN/mm2	
Assign C To Selection	
OK Cancel Help	

Figure 2.35

3. For specifying the DENSITY constant, it will be convenient if we change our length units to meters. To change the length units, as

before, click on the **Input Units** icon from the Structure toolbar, or select the *Tools* | *Set Current Input Unit* menu option from the top menu bar. In the *Set Current Input Units* dialog box that comes up, specify the length units as **Meter**.

Set Current Input Units		×
Length Units	- Force Units	
C Inch C Decime	ter C Pound	C Newton
C Foot C Meter	KiloPou	und C DecaNewton
C Millimeter C Kilomet	er C Metric	Ton 💿 KiloNewton
C Centimeter	C Kilogra	m C MegaNewton
	K Cancel	

Figure 2.36

2-40 Tutorial 2

- 4. Following the steps 1 and 2 above, we choose **Commands** | **Material Constants** | **Density**, specify the value as **25KN/m³**, and assign **To View**.
- 5. To define the **POISSON'S RATIO**, using the similar procedure as described above, provide the value **0.17** to all members in the **View**.

2.6.6 Specifying Supports

The base nodes of all the columns are restrained against translation and rotation about all the 3 global axes (see section 2.2). In other words, fixed supports are to be specified at those nodes. The commands to be generated are :

SUPPORTS 1 4 5 FIXED

Steps:

1. To create supports, click on the **Support Page** icon located in the Structure Tools toolbar as shown below.

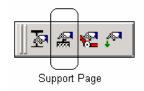


Figure 2.37

2-42 Tutorial 2

Alternatively, one may go to the *General* | *Support* Page from the left side of the screen.

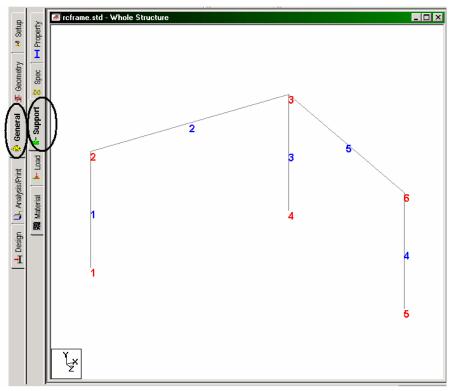


Figure 2.38

- In either case, the Supports dialog box comes up. Since we already know that nodes 1, 4 and 5 are to be associated with the Fixed support, using the Nodes Cursor bo, select these nodes.
- 3. Then, click on the **Create** button in the *Supports* dialog box as shown below.

88 Supports - Whole Structure	×					
Ref Description						
S1 No support						
Edit Create Delete						
Assignment Method						
 Assign To Selected Nodes 						
C Assign To View						
O Use Cursor To Assign						
C Assign To Edit List						
Assign Close Help						

Figure 2.39

2-44 Tutorial 2

4. The *Create Support* dialog box comes up. In the dialog box, the **Fixed** tab happens to be the default which is convenient for this case. Click on the **Assign** button as shown below.

Create	e Support							х
	iforced But Fixed		Multiline Pinned	ar Spring	_	Foundation	Inclined Enforced	
	Restraint						-	
		2	FX		V	М×		
			FY		V	MY		
		V	FZ		V	ΜZ		
							_	
_		Add		Cancel		Assign 💦	Help	

Figure 2.40

After the supports have been assigned, the structure will look like the one shown below.

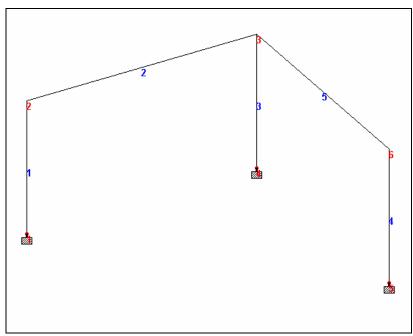


Figure 2.41

Click anywhere in the drawing area to un-select all selected nodes and prevent accidental assignment of unwanted data to those nodes.

As earlier, save the work completed so far by going to the **File** menu and clicking on the **Save** command.

2.6.7 Specifying Loads

5 load cases are to be created for this structure. Details of the individual cases are explained at the beginning of this tutorial. The corresponding commands to be generated are listed below. Notice that cases 4 and 5 are to be generated not as the standard combination type, but using a combination load type called REPEAT LOAD. The instructions at the beginning of this tutorial require us to analyze this structure using an analysis type called PDelta. A Pdelta analysis is a non-linear type of analysis. In STAAD, to accurately account for the PDelta effects arising from the simultaneous action of previously defined horizontal and vertical loads, those previous cases must be included as components of the combination case using the REPEAT LOAD type.

UNIT METER KG

LOAD 1 DEAD LOAD SELFWEIGHT Y -1 MEMBER LOAD 2 5 UNI GY -400

LOAD 2 LIVE LOAD MEMBER LOAD 2 5 UNI GY -600

LOAD 3 WIND LOAD MEMBER LOAD 1 UNI GX 300 4 UNI GX 500

LOAD 4 DEAD + LIVE REPEAT LOAD 1 1.2 2 1.5

LOAD 5 DEAD + WIND REPEAT LOAD 1 1.1 3 1.3

Steps:

LOAD CASE 1

1. To create loads, click on the **Load Page** icon located on the *Structure Tools* tool bar.

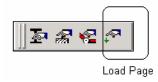


Figure 2.42

Alternatively, one may go to the *General* | *Load* Page from the left side of the screen.

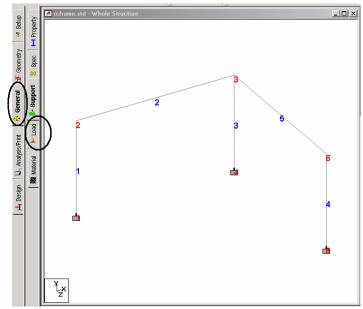


Figure 2.43

2-48 Tutorial 2

2. A window titled "Load" appears on the right-hand side of the screen. To initiate the first load case, highlight the Load Cases **Details** option and click on the Add button.

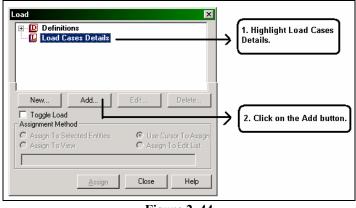


Figure 2.44

3. The *Add New Load Cases* dialog box comes up. The drop-down list box against *Loading Type* is available in case we wish to associate the load case we are creating with any of the ACI, AISC or IBC definitions of Dead, Live, Ice, etc. This type of association needs to be done if we intend to use the program's facility for automatically generating load combinations in accordance with those codes.

Notice that there is a check box called *Reducible per UBC/IBC*. This feature becomes active only when the load case is assigned a Loading Type called *Live* at the time of creation of that case. Please refer to STAAD.Pro 2004 Release Report for further details.

As we do not intend to use the automatic load combination generation option, we will leave the *Loading Type* as **None**. Enter **DEAD LOAD** as the *Title* for *Load Case 1* and click on **Add**.

Tutorial 2 2-49

Add	New Load Cases	X
	Primary Load Generation Define Combinations Auto Load Combination	Primary
		Number 1 Loading Type : None
		Title DEAD LOAD
		Add Close Help

Figure 2.45

The newly created load case will now appear under the *Load Cases Details* option.

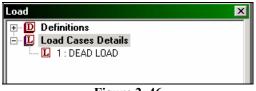


Figure 2.46

2-50 Tutorial 2

4. To generate and assign the selfweight load type, first **highlight** DEAD LOAD. You will notice that the *Add New Load Items* dialog box shows more options now.

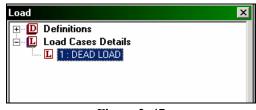


Figure 2.47

5. In the Add New Load Items dialog box, select the Selfweight Load option under the Selfweight item. Specify the Direction as Y, and the Factor as -1.0. The negative number signifies that the selfweight load acts opposite to the positive direction of the global axis (Y in this case) along which it is applied. Click on the Add button. The selfweight load is applicable to every member of the structure, and cannot be applied on a selected list of members.

Add New : Load Items		×
Selfweight Selfweight Selfweight Area Load Area Load Area Load Floot Load Sidu Loads Soid Loads Soid Loads Sidu Loads Sidu Loads Time History Wind Load Response Spectra Response Spectra Response Spectra Frequency	Selfweight Load	
	Add Close Help	

Figure 2.48

6. Load 1 contains an additional load component, the member loads on members 2 and 5. However, notice that the load values are listed in the beginning of this tutorial in *kg* and *meter* units. Rather than convert those values to the current input units, we will conform to those units. The current input units, which we last set while specifying Density, are KN and METER. We have to change the force unit to Kilogram.

STAAD has a limitation in that one cannot change the units while editing load cases. If we attempt to, the following message will be displayed.



Figure 2.49

Click on *OK*. Close the *Add New Load Items* dialog box. Then, go to the *Setup* Page as shown below (or any other page).



Figure 2.50

As before, click on the **Input Units** icon from the top toolbar, or select the *Tools* | *Set Current Input Unit* menu option from the top menu bar. In the *Set Current Input Units* dialog box that comes up, specify the force units as **Kilogram**. Again, click on *General* | *Load* page to resume creating the load cases.

 To create the member load, first, click on the expression DEAD LOAD followed by the Add button. Then, click on the Member Load item in the Add New Load Items dialog box.

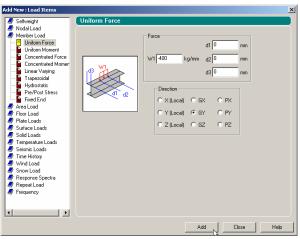


Figure 2.51

- 8. Select the **Uniform Force** option and specify **GY** as the *Direction* and **-400** as the *Force*. For these members, since the local Y axis coincides with the global Y axis, one may choose the direction of the load as either "Y" or "GY", they will both have the same effect. (One may view the orientation of the member local axes by going to *View* | *Structure Diagrams* | *Labels* | *Beam Orientation*.) The negative value signifies that the load acts along the negative GY direction. Then, click on the **Add** button followed by the **Close** button.
- 9. The member load we just created has to be assigned to members 2 and 5. First, make sure that the expression UNI GY -400Kg/m is selected in the *Load* dialog box as shown below.

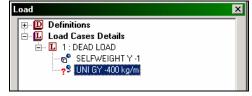


Figure 2. 52

10. Next, select members 2 and 5 using the *Beams Cursor* .
(Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting members.)

Then, click on *Assign to Selected Beams* followed by the **Assign** button.

Assignment Method	C Use Cursor To Assign		
	C Assign To Edit List		
<u>A</u> s:	sign	Close	Help

Figure 2.53

As we click on the *Assign* button, the following dialog box appears. This message box appears just to confirm that we indeed wish to associate the loadcase with the selected beams. So, let us choose **Yes**.



Figure 2. 54

2-54 Tutorial 2

After the load has been assigned, the structure will look as shown below:

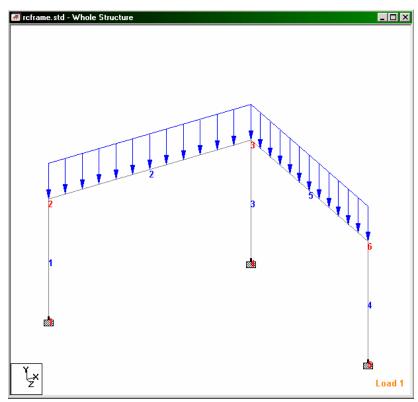


Figure 2.55

LOAD CASE 2

11. The next step is to initiate the second load case which again contains MEMBER LOADs. Highlight Load Cases Details and click on the Add button. Once again, the Add New Load Cases dialog box comes up.

Load X Definitions Load Cases Details Definitions Load Cases Details Definitions Definitio	— (1. Highlight this.)
New Add Edit Delete Toggle Load Assignment Method C Assign To Selected Beams C Use Cursor To Assign	2. Click on Add.
C Assign To View C Assign To Edit List Assign Close Help	

Figure 2.56

In this dialog box, once again, we are not associating the load case we are about to create with any code based Loading Type and so, we will leave that box as **None**. Specify the *Title* of the second load case as **LIVE LOAD** and click on the **Add** button.

Ac	d New Load Cases		<
	Primary Load Generation Define Combinations Auto Load Combination	Primary	
		Number 2 Loading Type: None	
		Title LUVE LOAD	
		Add Close Help]

Figure 2. 57

2-56 Tutorial 2

12. To create the member load, highlight LIVE LOAD as shown below.

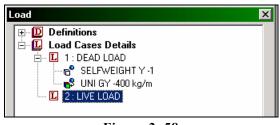


Figure 2.58

13. Follow steps 7 to 10 to create and assign a uniformly distributed force of -600Kg/m on members 2 and 5.

After the second load case has been assigned, the structure will look as shown below:

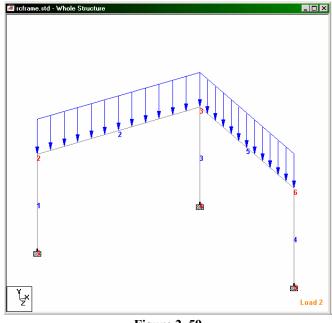


Figure 2. 59

Click anywhere in the drawing area to un-highlight the members.

LOAD CASE 3

- 14. Creating the third load case, which again has MEMBER LOADs, involves the same procedure as that for load case 2. As before, first highlight **Load Cases Details** in the *Load* dialog box to initiate the third load case. Enter **WIND LOAD** as the *Title* for Load Case 3.
- 15. To apply the load on member 1, follow the procedure similar to that in steps 7 to 10. The only differences are, the member which receives this load is 1, the *Direction* is **GX** and the *Force* is +300Kg/m.
- 16. Similarly, for member 4 and the third load case, specify the *Force* as **500Kg/m** and the *Direction* as **GX**.

After the third load case has been assigned, the structure will look as shown below:

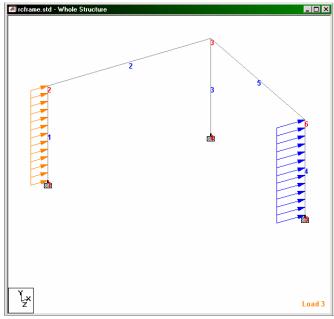


Figure 2.60

LOAD CASE 4

- 17. We now come to the point where we have to create load case 4 as (1.2 x Load 1) + (1.5 x Load 2). We saw in the beginning of this section that we should be creating a "REPEAT LOAD" type of combination, and not the "LOAD COMBINATION" type. To initiate load case 4, highlight Load Cases Details in the Load dialog box and specify the title as DEAD + LIVE.
- Then, click on DEAD + LIVE in the Load dialog box as shown below.

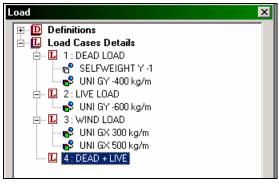


Figure 2. 61

- 19. In the Add New Load Items dialog box, click on the Repeat Load option. Then, select Load Case 1 (DEAD LOAD), click on the button and enter the Factor as 1.2. (This indicates that the load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4.)
- 20. Similarly, select Load Case 2 (LIVE LOAD), click on the button and enter the *Factor* as **1.5**.

The *Add New Load Items* dialog box will now look as shown below. Click on the **Add** button.

Add New : Load Items					×
Selfweight	Repeat Load				
🦪 Nodal Load	Available Load Cases:		Repeated Load Definiti		
🦪 Member Load 🍠 Area Load			· · · · · · · · · · · · · · · · · · ·	on.	
🦪 Area Load	3: WIND LOAD		Load Cases	Factor	
Plate Loads			Load Case 1	1.2	
Surface Loads		>	Load Case 2	1.5	
I Solid Loads		>>			
Temperature Loads					
Seismic Loads					
🛷 Time History 🛷 Wind Load		<<			
Snow Load		<			
Sesponse Spectra					
I Repeat Load					
Repeat Load	1		,		
I Frequency					
,					
			Add	Close	Help
			Add	Close	Help

Figure 2. 62

No further operation is required for load case 4. The recipients (members) of the loads in load case 4 are automatically chosen to be the very same ones to which the components of the REPEAT LOAD cases (loads 1 and 2) were assigned.

The structure will now look similar to the one shown below.

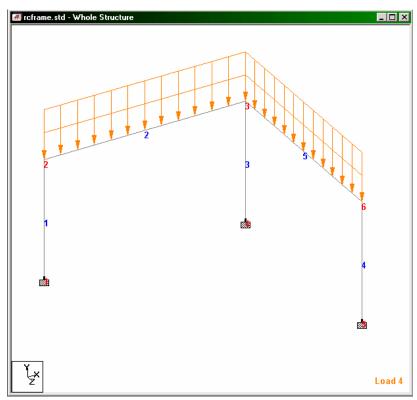


Figure 2.63

LOAD CASE 5

- 21. Since load cases 4 and 5 are near identical in nature, the same procedure used in creating load case 4 is applicable for case 5 also. Let us highlight Load Cases Details in the Load dialog box to initiate the fifth load case. Enter DEAD + WIND as the Title for Load Case 5.
- 22. Follow steps 18 to 20 except for associating a *Factor* of **1.1** with the first load case and a *Factor* of **1.3** with the third load case.

The *Add New Load Items* dialog box will now look as shown below. Click on the **Add** button.

Add New : Load Items					×
I Selfweight	Repeat Load				
I Nodal Load	Available Load Cases:		Repeated Load Definiti		
🦉 Member Load			nepeated Load Dennit	ion:	_
🦪 Area Load	2: LIVE LOAD		Load Cases	Factor	
Schoor Load	4: DEAD + LIVE		Load Case 1	1.1	
Surface Loads		>	Load Case 3	1.3	
Solid Loads		>>			
Temperature Loads					
Seismic Loads					
Ime History					
I Wind Load					
I Snow Load		<			
I Response Spectra					
I Repeat Load					
🔤 Repeat Load	1		,		
I Frequency					
				- 1	
			Add	Close	Help
	E *	. 2 (4			

Figure 2.64

Since we have completed creating all the load cases, we may now click on the **Close** button to dismiss the *Add New Load Items* dialog box.

The structure will now look similar to the one shown below.

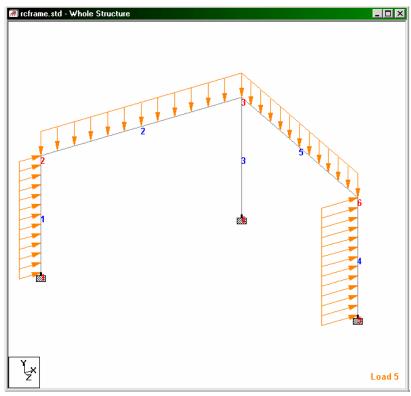


Figure 2.65

Let us save the work completed so far by going to the **File** menu and selecting the **Save** command or by holding the 'Ctrl' key and pressing the 'S' key.

2.6.8 Specifying the analysis type

The analysis type for this structure is called P-Delta. Since this problem involves concrete beam and column design per the ACI code, second-order analysis is required and has to be done on factored loads acting simultaneously. The factored loads have been created earlier as cases 4 and 5. Now is the time to specify the analysis type.

The command for a pdelta analysis will appear in the STAAD file as:

PDELTA ANALYSIS

Steps:

	ame.std - Whole Structure	
8 문 🛛	Analysis/Print Commands	×
 Analysis/Print & General	Perform Cable Analysis PDelta Analysis Nonlinear Analysis Perform Analysis Perform Imperfection Ana Print Option © No Print © Load Data © Statics Load © Mode Shapes © Both	Change
H Design	C All After Add Assign Close	Help

1. Go to Analysis/Print Page on the left side of the screen.

Figure 2.66

2-64 Tutorial 2

2. In the *Analysis/Print Commands* dialog box that appears, select the **PDelta Analysis** tab. Then, click on the **Add** button followed by the **Close** button.

Analysis/Print Commands		×
Perform Analysis	Perform Imperfection Analy	vsis
Perform Cable Analysis PDelta A	nalysis Nonlinear Analysis	Change
Number of Iterations	Print Option No Print Load Data Statics Load Mode Shapes Both All	
After Add	Assign Close	Help

Figure 2.67

Save the work again using the Save option of the File menu.

2.6.9 Short-listing the load cases to be used in concrete design

The concrete design has to be performed for load cases 4 and 5 only since only those are the factored cases. To instruct the program to use just these cases, and ignore the remaining, we have to use the LOAD LIST command.

The command will appear in the STAAD file as :

LOAD LIST 4 5

Steps:

In the menus on the top of the screen, go to Commands | Loading
 | Load List option as shown below.

Commands Analyze Mode Window	Help
Plate Thickness Surface Thickness	۵ 🛥 🛋 🍓 🗅 📗 🗂 📥 🟥 🕶
Member Property	, 🔍 🔍 📿 🔍 🖻 🛒 🔍 📣 1: DE
Material Constants	→ M 和 🕆 🖗 🛣 🐎 9 省 9
Geometric Constants	
Support Specifications	•
Member Specifications	•
Plate Element Specifications	•
Master/Slave Specification	•
Pre Analysis Print	•
№ Define Damping for Dynamics	
Loading	Definitions
Analysis	Primary Load
Post-Analysis Print	Moving Load Generation
Design	Load Combination Automatic Load Combination
Miscellaneous	Load Commands
	Define Primary Load Types
	Load List
	Edit Auto Load Rules

Figure 2.68

2-66 Tutorial 2

2. In the Load List dialog box that comes up, select load cases 4 (DEAD + LIVE) and 5 (DEAD + WIND) by holding the 'Ctrl' key down. Then, click on the button. Load cases 4 and 5 will be selected and placed in the Load List selection box as shown below. Click on the OK button.

Load List		×
Load Cases 1: DEAD LOAD 2: LIVE LOAD 3: WIND LOAD	 Load List 4: DEAD + LIVE 5: DEAD + WIND 	OK Cancel Help
Use the > button to transfer selected load cases to the load list. Use >> to transfer all.	Use the < button to remove selected load cases from the load list. Use << to remove all.	

Figure 2.69

2.6.10 Specifying concrete design parameters

Among the various terms which appear in the equations for design of concrete beams and columns, some of them can be user controlled, such as, the grade of concrete, or the maximum size of reinforcing bar one may wish to use. Such terms are called concrete design parameters. For the ACI code, a list of these parameters is available in Section 3 of the STAAD.Pro Technical Reference Manual. The parameters we wish to use, and the corresponding command which ought to appear in the STAAD input file are :

> UNIT MMS NEWTON CODE ACI CLT 25 ALL CLB 30 ALL CLS 25 ALL FC 25 ALL FYMAIN 415 ALL TRACK 1 ALL

Steps:

- 1. Before we can start assigning the parameters, we want our force units to be Newton and our length units to be millimeter. We last set the units during load specification as Kg and Meter. To change the units, as before, click on the **Input Units** icon from the appropriate toolbar, or select the *Tools* | *Set Current Input Unit* menu option from the top menu bar. In the *Set Current Input Units* dialog box that comes up, specify the force units as **Newton** and the length units as **Millimeter**.
- Next, go to Design | Concrete Page from the left side of the screen. Make sure that under the *Current Code* selections on the top right hand side, ACI is selected. Then, click on the Define Parameters button in the *Concrete Design* dialog box.

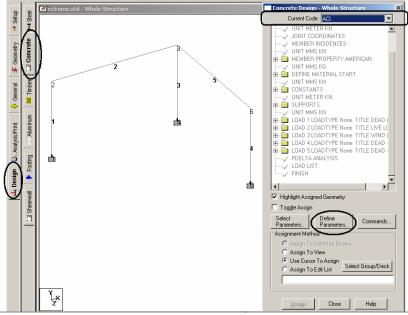


Figure 2.70

3. In the *Design Parameters* dialog box that opens, select the **Clt** (Clear Cover for top) tab. Then, provide the value as **25mm** and click on the **Add** button as shown below.

Design Parameters		x
CLB CLS CLS DEPTH EFACE FC FC MAXMAIN MINMAIN MINMAIN MINSEC MAMAG NSECTION REINF RHOMN SFACE TRACK WIDTH	CLT Clear cover for outermost top reinforcement: 25 mm	
	Close Help	

Figure 2.71

4. To define the remaining parameters, follow the above procedure and provide the following values.

Parameter	Value
Clb	30
Cls	25
Fc	25
Fymain	415
Track	1.0

When all the above parameters have been assigned, click on the **Close** button in the *Design Parameters* dialog box.

After all the design parameters have been assigned, the *Concrete Design* dialog box will look as shown below.

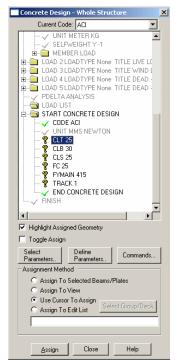


Figure 2.72

The next step is to assign these parameters to all the members in our model. The easiest way to do that is to use the **Assign To View** method. Follow the steps as shown in the figure below.

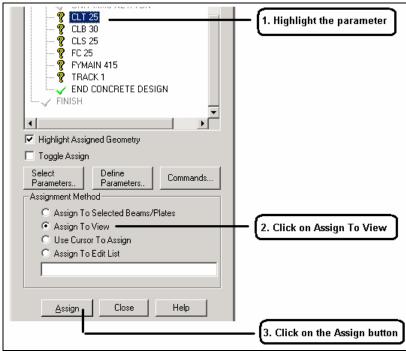


Figure 2.73

Let us save our structure once again using the **Save** option of the **File** menu.

2.6.11 Specifying design commands

Design commands are the actual instructions for the design of beams and columns. We intend to design beams 2 and 5 and columns 1, 3 and 4. The commands to be generated are :

DESIGN BEAM 2 5 DESIGN COLUMN 1 3 4

Steps:

1. Design commands are generated through the dialog boxes available under the *Commands* button in the *Concrete Design* dialog box. So, let us click on the **Commands** button as shown below.

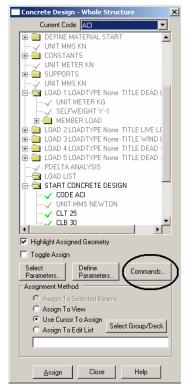


Figure 2.74

2. In the *Design Commands* dialog box that comes up, select the **DESIGN BEAM** option and click on the **Add** button.

Design Commands		×
DESIGN BEAM	DESIGN BEAM	
DESIGN COLUMN DESIGN SLAB/ELEMEN ⁻ TAKE OFF	Design beams for flexure, shear and torsion.	
	This command has no additional parameters.	
	After Add Assign Close Help	

Figure 2.75

3. We also need to add a command for designing columns. So, select the **DESIGN COLUMN** option and click on **Add button**.

After steps 2 and 3 are completed, let us Close this dialog box.

4. The next step is to associate the Design Beam command with members 2 and 5 and the Design Column command with members 1, 3 and 4.

To do this, as before, first highlight the expression DESIGN BEAM. Then, **select members 2 and 5** using the *Beams Cursor* Click on *Assign to Selected Beams* followed by the **Assign** button.

Assignment Method	
Assign To Selected Beams	
🔿 Assign To View	
Use Cursor To Assign Select Group/Deck	
Assign Close Help	

Figure 2.76

As we click on the *Assign* button, the following dialog box appears. This message box appears just to confirm that we indeed wish to associate the design command with the selected beams. So, let us say **Yes**.

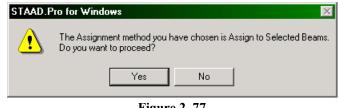


Figure 2.77

Similarly, assign the **Design Column** command to members 1, 3 and 4

This concludes the task of assigning all the input for our model. Let us **Save** the file one final time.

2-74 Tutorial 22.7 Viewing the input command file

Let us now take a look at the data that has been written into the file that we just saved above. The contents of the file can be viewed either by clicking on the **STAAD Editor** icon or, by going to the *Edit* menu and choosing *Edit Input Command File* as shown below.

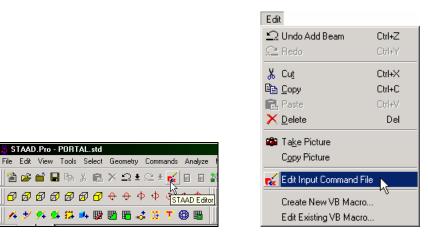


Figure 2.78

Tutorial 2-75

A new window will open up with the data listed as shown here:

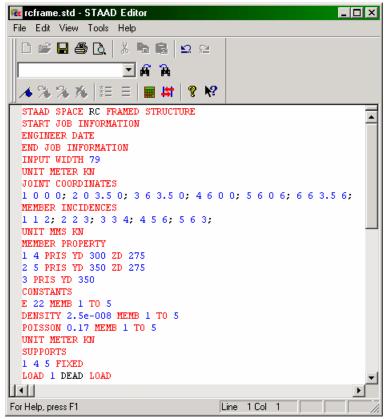


Figure 2.79

This window and the facilities it contains is known as the *STAAD Editor*.

We could make modifications to the data of our structure in this *Editor* if we wish to do so. Let us **Exit** the *Editor* without doing so by selecting the *File* | *Exit* menu option of the editor window (not the File | Exit menu of the main window behind the editor window).

As we saw in Section 2.1, we could also have created the same model by typing the relevant STAAD commands into a text file using either the STAAD editor, or by using any external editor of our choice. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to section 2.9 where we perform the analysis and design on this model.

2.8 Creating the model using the command file

Let us now use the command file method to create the model for the above structure. The commands used in the command file are described later in this section.

The STAAD.Pro command file may be created using the built-in editor, the procedure for which is explained further below in this section. 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. A typical editor screen is shown below to illustrate its general appearance.

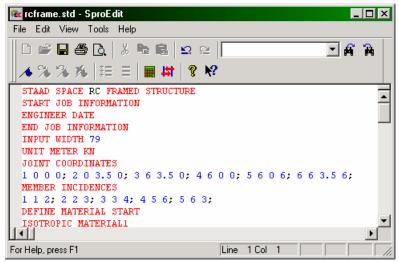


Figure 2.80

To access the built-in editor, first start the program using the procedure explained in Section 2.2. Next, follow step 1 of Section 2.4.

Plane Floor Truss		rcframe	
-		C:\test\].
C Millimeter	C Decimeter Meter C Kilometer	Force Units C Pound C Newton C KiloPound C DecaNewtor C Kilogram (© KiloNewton	
C Centimeter		C Metric Ton C MegaNewton)

Figure 2.81

You will then encounter the dialog box shown below. In this dialog box, choose *Open STAAD Editor*.

Where do you want to go ?	×
Add Beam Add Plate Add Sold Dpen Structure Wizard Dipen STADE Editor Edit Job Information	
Begin building your model using STAAD syntax commands (non-graphical interface) through the STAAD editor.	
✓ Display this dialog box at the Startup < Back	

Figure 2.82

Tutorial 2-79

At this point, the editor screen will open as shown below.

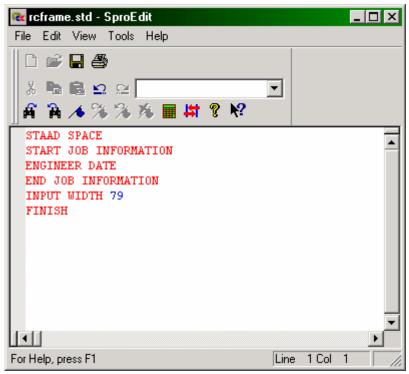


Figure 2.83

Delete all the command lines displayed in the editor window and type the lines shown in bold below (You don't have to delete the lines if you know which to keep and where to fill in the rest of the commands). The commands may be typed in upper or lower case letters. Usually the first three letters of a keyword are all that are needed -- the rest of the letters of the word are not required. The required letters are underlined. ("SPACE" = "SPA" = "space" = "spa")

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE RC FRAMED STRUCTURE

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

UNIT METER KN

Specifies the unit to be used.

<u>JOI</u>NT <u>COO</u>RDINATES 1 0 0 0 ; 2 0 3.5 0 ; 3 6 3.5 0 4 6 0 0 ; 5 6 0 6 ; 6 6 3.5 6

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables us to provide multiple sets of data on one line.

<u>MEMBER INCIDENCES</u> 1 1 2 ; 2 2 3 ; 3 3 4 4 5 6 ; 5 6 3

Defines the members by the joints they are connected to.

```
<u>UNIT MMS KN</u>
<u>MEMBER PRO</u>PERTY <u>AME</u>RICAN
1 4 <u>PRIS YD</u> 300 <u>ZD</u> 275
2 5 <u>PRIS YD</u> 350 <u>ZD</u> 275
3 <u>PRIS YD</u> 350
```

Member properties have been defined above using the PRISMATIC attribute for which YD (depth) and ZD (width) values are provided in MM unit. When YD and ZD are provided together, STAAD considers the section to be rectangular. When YD alone is specified, the section is considered to be circular. Details are available in Section 5 of the Technical Reference Manual.

<u>CON</u>STANTS <u>E</u> 22 <u>MEM</u>B 1 TO 5

Material constant E (modulus of elasticity) is specified as 22KN/sq.mm following the command CONSTANTS.

<u>UNIT METER KN</u> <u>CON</u>STANTS <u>DEN</u>SITY 25.0 <u>ALL</u> <u>POI</u>SSON 0.17 <u>ALL</u>

Length unit is changed from MMS to METER to facilitate the input of Density. Next, the Poisson's Ratio is specified.

<u>BET</u>A 90 <u>MEM</u>B 4

In the absence of any explicit instructions, STAAD will orient the beams and columns of the structure in a pre-defined way (see Section 1 of the Technical Reference Manual for details.) In order to orient member 4 so that its longer edges (sides parallel to local Y axis) are parallel to the global Z axis, we need to apply a beta angle of 90 degrees.

<u>SUP</u>PORT 1 4 5 <u>FIX</u>ED

Joints 1, 4 and 5 are defined as fixed supported.

<u>UNI</u>T <u>MET</u>ER <u>KG</u> <u>LOA</u>D 1 DEAD LOAD

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is initiated along with an accompanying title.

SELFWEIGHT Y -1

One of the components of load case 1 is the selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

<u>MEM</u>BER <u>LOA</u>D 2 5 <u>UNI GY</u> -400

Load 1 contains member loads also. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

LOAD 2 LIVE LOAD

Load case 2 is initiated along with an accompanying title.

<u>MEM</u>BER <u>LOA</u>D 2 5 <u>UNI GY</u> -600

Load 2 also contains member loads. GY indicates that the load is in the global Y direction. The word UNI stands for uniformly distributed load. Loads are applied on members 2 and 5.

LOAD 3 WIND LOAD

Load case 3 is initiated along with an accompanying title.

<u>MEM</u>BER <u>LOA</u>D 1 <u>UNI GX</u> 300 4 <u>UNI GX</u> 500

Load 3 also contains member loads. GX indicates that the load is in the global X direction. The word UNI stands for uniformly distributed load. Loads are applied on members 1 and 4.

LOAD 4 DEAD + LIVE

Load case 4 is initiated along with an accompanying title.

<u>REP</u>EAT <u>LOA</u>D 1 1.2 2 1.5

Load case 4 illustrates the technique employed to instruct STAAD to create a load case which consists of data to be assembled from other load cases specified earlier. We are instructing the program to analyze the structure for loads from cases 1 and 2 acting simultaneously. The load data values from load case 1 are multiplied by a factor of 1.2, and the resulting values are utilized in load case 4. Similarly, the load data values from load case 2 are multiplied by a factor of 1.5, and the resulting values too are utilized in load case 4.

LOAD 5 DEAD + WIND

Load case 5 is initiated along with an accompanying title.

<u>REP</u>EAT <u>LOA</u>D 1 1.1 3 1.3

We are instructing the program to analyze the structure for loads from cases 1 and 3 acting simultaneously.

<u>PDE</u>LTA <u>ANA</u>LYSIS

The PDELTA ANALYSIS command is an instruction to the program to execute a second-order analysis and account for P-delta effects.

<u>LOA</u>D LIST 4 5

The above LOAD LIST command is a means of stating that all further calculations should be based on the results of load cases 4 and 5 only. The intent here is to restrict concrete design calculations to that for load cases 4 and 5 only.

2-84 Tutorial 2

START CONCRETE DESIGN CODE ACI UNIT MMS NEWTON CLT 25 ALL CLB 30 ALL CLS 25 ALL FC 25 ALL FYMAIN 415 ALL TRACK 1 ALL

We first line is the command that initiates the concrete design operation. The values for the concrete design parameters are defined in the above commands. Design is performed per the ACI Code. The length units are changed from METER to MMS to facilitate the input of the design parameters. Similarly, force units are changed from KG to NEWTON. The TRACK value dictates the extent of design related information which should be produced by the program in the output. The parameters specified include CLT(Clear cover for top surface), CLB (Clear cover for bottom surface), CLS (Clear cover for sides), FC(Strength of concrete), and FYMAIN(Ultimate strength of steel). These parameters are described in Section 3 of the Technical Reference Manual.

DESIGN BEAM 2 5 DESIGN COLUMN 1 3 4

The above commands instruct the program to design beams 2 and 5 for flexure, shear and torsion, and to design columns 1, 3 and 4 for axial load and biaxial bending.

END CONCRETE DESIGN

This command terminates the concrete design operation.

<u>FIN</u>ISH

This command terminates the STAAD run.

Let us save the file and exit the editor.

2.9 Performing the analysis and design

STAAD.Pro performs Analysis and Design simultaneously. In order to perform Analysis and Design, select the **Run Analysis** option from the **Analyze** menu.



If the structure has not been saved after the last change was made, you should save the structure first by using the *Save* command from the *File* menu.

When you select the *Run Analysis* option from the *Analyze* menu, the following dialog box appears:

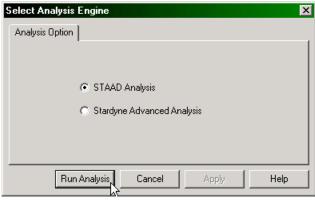


Figure 2.85

We are presented with the choice of 2 engines : the STAAD engine and the STARDYNE Advanced Analysis engine. The STARDYNE Analysis engine is suitable for advanced problems such as Buckling Analysis, Modal Extraction using various methods, etc. However, if the calculations call for steel or concrete design, UBC load generation, etc., we have to select the STAAD engine. So, let us ensure that the radio button is on the **STAAD** engine. 2-86 Tutorial 2

Click on the Run Analysis button.

As the Analysis progresses, several messages appear on the screen as shown in the next figure.

STAAD Analysis and Design	
++ Read/Check Data in Load Cases	18:19:49 🔺
++ Processing and setting up Load Vector.	18:19:49
++ Processing Element Stiffness Matrix.	18:19:49
++ Processing Global Stiffness Matrix.	18:19:49
++ Finished Processing Global Stiffness Matrix.	0 sec
++ Processing Triangular Factorization.	18:19:49
++ Finished Triangular Factorization.	0 sec
++ Calculating Joint Displacements.	18:19:49
++ Finished Joint Displacement Calculation.	0 sec
++ Adjusting Displacements	18:19:49
++ Calculating Member Forces.	18:19:49
++ Analysis Successfully Completed ++	
++ Performing Concrete Design	18:19:49
++ Calculating Section Forces	
++ Creating Displacement File (DSP)	18:19:50
++ Creating Reaction File (REA)	
++ Calculating Section Forces	
++ Creating Section Force File (BMD)	
++ Creating Section Displace File (SCN)	
++ Creating Design information File (DGN)	
++ Done.	18:19:50
0 Error(s), 0 Warning(s)	
** End STAAD.Pro Run Elapsed Time = 1 Secs	
** Output Written to File:	
rcframe.anl	-
C View Output File	
Go to Post Processing Mode	
Stay in Modelling Mode	Done
S - Stay in modeling mode	

Figure 2.86

Notice that we can choose from the three options available in the above dialog box :

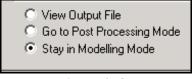


Figure 2.87

These options are indicative of what will happen after we click on the **Done** button.

The *View Output File* option allows us to view the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands we specified during the model generation process. It also tells us whether any errors were encountered, and if so, whether the analysis and design was successfully completed or not. Section 2.10 offers additional details on viewing and understanding the contents of the output file.

The *Go to Post Processing Mode* option allows us to go to graphical part of the program known as the Post-processor. This is where one can extensively verify the results, view the results graphically, plot result diagrams, produce reports, etc. Section 2.11 explains the post processing mode in greater detail.

The *Stay in Modelling Mode* lets us continue to be in the Model generation mode of the program (the one we current are in) in case we wish to make further changes to our model.

2.10 Viewing the output file

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly. For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

We can access the output file using the method explained at the end of the previous section. Alternatively, we can select the **File** | **View** | **Output File** | **STAAD Output** option from the top menu. The STAAD.Pro output file for the problem we just ran is shown in the next few pages.

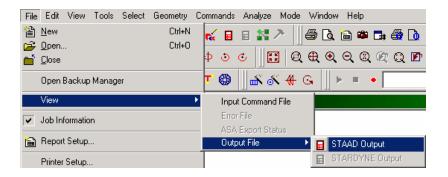


Figure 2.88

The STAAD.Pro output file is displayed through a file viewer called *SproView*. This viewer allows us to set the text font for the entire file and print the output file to a printer. Use the appropriate *File* menu option from the menu bar.

🗐 rcframe.anl - SproVie w				
File Edit Vie	w Help			
Open	Ctrl+0	N?		
Print	Ctrl+P			
Print Preview	N			
Print Setup				
Page Setup				
Font Setup				
🗸 Top Most				
Recent File				
Exit				

Figure 2.89

By default, the output file contains a listing of the entire input also. You may choose not to print the echo of the input commands in the output file. Please select *Commands* | *Miscellaneous* | *Set Echo* option from the menu bar and select the *Echo Off* button.

It is quite important that we browse through the entire output file and make sure that the results look reasonable, that there are no error messages or warnings reported, etc. Errors encountered during the analysis & design can disable access to the postprocessing mode – the graphical screens where results can be viewed graphically. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
STAAD.Pro
        *
                  Version
                                Bld
        *
                  Proprietary Program of
                                                      *
                  Research Engineers, Intl.
        *
                  Date=
                  Time=
        *
             USER ID:
        *
        1. STAAD SPACE RC FRAMED STRUCTURE
2. START JOB INFORMATION
3. ENGINEER DATE
4. END JOB INFORMATION
5. INPUT WIDTH 79
6 UNIT METER KN
7. JOINT COORDINATES
8. 1 0 0 0; 2 0 3.5 0; 3 6 3.5 0; 4 6 0 0; 5 6 0 6; 6 6 3.5 6
9. MEMBER INCIDENCES
10. 1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 3
11. UNIT MMS KN
12. MEMBER PROPERTY AMERICAN
13. 1 4 PRIS YD 300 ZD 275
14. 2 5 PRIS YD 350 ZD 275
15. 3 PRIS YD 350
16. CONSTANTS
17. E 22 MEMB 1 TO 5
18. UNIT METER KN
19. CONSTANTS
20. DENSITY 25.0 ALL
21. POISSON 0.17 ALL
22. BETA 90 MEMB 4
23. SUPPORTS
24. 1 4 5 FIXED
25. UNIT METER KG
26. LOAD 1 DEAD LOAD
27. SELFWEIGHT Y -1
28. MEMBER LOAD
29. 2 5 UNI GY -400
30. LOAD 2 LIVE LOAD
31. MEMBER LOAD
32. 2 5 UNI GY -600
33. LOAD 3 WIND LOAD
34. MEMBER LOAD
35. 1 UNI GX 300
36. 4 UNI GX 500
37. LOAD 4 DEAD + LIVE
38. REPEAT LOAD
39. 1 1.2 2 1.5
40. LOAD 5 DEAD + WIND
41. REPEAT LOAD
42. 1 1.1 3 1.3
43. PDELTA ANALYSIS
       PROBLEM STATISTICS
       _____
NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS = 6/
                                                  5/ 3
ORIGINAL/FINAL BAND-WIDTH= 3/ 3/ 12 DOF
                          5, TOTAL DEGREES OF FREEDOM =
TOTAL PRIMARY LOAD CASES =
                                                        18
SIZE OF STIFFNESS MATRIX =
                           1 DOUBLE KILO-WORDS
SIZE OF STIFFNESS MATRIX = 1 DOUBLE KILO
REQRD/AVAIL. DISK SPACE = 12.0/ 43064.0 MB
```

Tutorial 2 2-91

44. LOAD LIST 4 5 45. START CONCRETE DESIGN 46. CODE ACI 47. UNIT MMS NEWTON 48. CLT 25 MEMB 1 TO 5 49. CLB 30 MEMB 1 TO 5 50. CLS 25 ALL 51. FC 25 MEMB 1 TO 5 52. FYMAIN 415 MEMB 1 TO 5 53. TRACK 1 ALL 54. DESIGN BEAM 2 5 2 DESIGN RESULTS - FLEXURE PER CODE ACI 318-02 BEAM NO LEN - 6000. MM FY - 415. FC - 25. MPA, SIZE - 275. X 350. MMS LEVEL HEIGHT BAR INFO FROM то ANCHOR (MM) (MM) (MM) STA END 1 51 2 - 16MM 313. 5437. NO NO |-----| CRITICAL POS MOMENT= 39.16 KN-MET AT 3000.MM, LOAD 4| 1 REQD STEEL= 366.MM2, ROW=0.0044, ROWMX=0.0193 ROWMN=0.0033 | | MAX/MIN/ACTUAL BAR SPACING= 273./ 41./ 184. MMS | | REQD. DEVELOPMENT LENGTH = 480. MMS 1 |------| Cracked Moment of Inertia Iz at above location = 20964.0 cm⁴ 860. YES NO 2 306. 3 - 12MM Ο. |-----| CRITICAL NEG MOMENT= 33.29 KN-MET AT 0.MM, LOAD 4 REQD STEEL= 301.MM2, ROW=0.0036, ROWMX=0.0193 ROWMN=0.0033 | MAX/MIN/ACTUAL BAR SPACING= 273./ 37./ 94. MMS REQD. DEVELOPMENT LENGTH = 360. MMS - 1 -----| 1 -Cracked Moment of Inertia Iz at above location = 19148.5 cm⁴ 3 306. 3 - 12MM 4390. 6000. NO YES |-----| CRITICAL NEG MOMENT= 36.18 KN-MET AT 6000.MM, LOAD 4 | REQD STEEL= 329.MM2, ROW=0.0039, ROWMX=0.0193 ROWMN=0.0033 | MAX/MIN/ACTUAL BAR SPACING= 273./ 37./ 94. MMS 1 REQD. DEVELOPMENT LENGTH = 360. MMS 1 1 |-----| Cracked Moment of Inertia Iz at above location = 19148.5 cm^4 BEAM NO. 2 D E S I G N R E S U L T S - SHEAR AT START SUPPORT - Vu= 43.94 KNS Vc= 72.44 KNS Vs= 0.00 KNS Tu= 2.55 KN-MET Tc= 2.4 KN-MET Ts= 3.4 KN-MET LOAD 4 3.4 KN-MET LOAD STIRRUPS ARE REQUIRED FOR TORSION. REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1. PROVIDE 12 MM 2-LEGGED STIRRUPS AT 130. MM C/C FOR 2705. MM ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE = 0.75 SQ.CM. AT END SUPPORT - Vu= 44.91 KNS Vc= 71.94 KNS Vs= 0.00 KNS Tu= 2.55 KN-MET Tc= 2.4 KN-MET Ts= 3.4 KN-MET LOAD 4 STIRRUPS ARE REQUIRED FOR TORSION. REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1. PROVIDE 12 MM 2-LEGGED STIRRUPS AT 130. MM C/C FOR 2705. MM ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE = 0.75 SQ.CM.

```
2-92 Tutorial 2
```

2J_ _____6000x 275x 350____ _____ 3J____ _____| | | === | 3No12 H 306. 0.TO 860 3No12|H|306.4390.TO 6000 | 22*12c/c130 | | |22*12c/c130 2No16 H 51. 313.TO 5437 000 1 Т 1 1 1 000 1 - 1 000 1 | 3#12 1 3#12 1 3#12 1 1 1 т | 2#16 | 2#16 | 2#16 - 1 1 1 Т 00 Т 1 00 1 00 BEAM NO. 5 DESIGN RESULTS - FLEXURE PER CODE ACI 318-02 LEN - 6000. MM FY - 415. FC - 25. MPA, SIZE - 275. X 350. MMS BAR INFO LEVEL HEIGHT FROM то ANCHOR (MM) (MM) (MM) STA END 1 51. 2 - 16MM 313. 5437. NO NO |-----| | CRITICAL POS MOMENT= 39.16 KN-MET AT 3000.MM, LOAD 4| REQD STEEL= 366.MM2, ROW=0.0044, ROWMX=0.0193 ROWMN=0.0033 | MAX/MIN/ACTUAL BAR SPACING= 273./ 41./ 184. MMS - 1 | REQD. DEVELOPMENT LENGTH = 480. MMS 1 ------Cracked Moment of Inertia Iz at above location = 20964.0 cm^4 3 - 12MM 860. 2 306. 0. YES NO |-----| CRITICAL NEG MOMENT= 33.29 KN-MET AT 0.MM, LOAD 4 REQD STEEL= 301.MM2, ROW=0.0036, ROWMX=0.0193 ROWMN=0.0033 | 1 MAX/MIN/ACTUAL BAR SPACING= 273./ 37./ 94. MMS | REQD. DEVELOPMENT LENGTH = 360. MMS | 1 -----| Cracked Moment of Inertia Iz at above location = 19148.5 cm⁴ 306. 3 3 - 12MM 4890. 6000. NO YES -----| 1---------CRITICAL NEG MOMENT= 36.18 KN-MET AT 6000.MM, LOAD 4 | REQD STEEL= 329.MM2, ROW=0.0039, ROWMX=0.0193 ROWMN=0.0033 | MAX/MIN/ACTUAL BAR SPACING= 273./ 37./ 94. MMS | REOD. DEVELOPMENT LENGTH = 360. MMS | . Т -----| Cracked Moment of Inertia Iz at above location = 19148.5 cm⁴ BEAM NO. 5 DESIGN RESULTS - SHEAR AT START SUPPORT - Vu= 18.56 KNS Vc= 72.56 KNS Vs= 0.00 Tu= 4.73 KN-MET Tc= 2.3 KN-MET Ts= 6.3 KN-MET LOAD 0 00 KNS 5 STIRRUPS ARE REQUIRED FOR TORSION. REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1. PROVIDE 12 MM 2-LEGGED STIRRUPS AT 130. MM C/C FOR 2705. MM ADDITIONAL LONGITUDINAL STEEL REOD. FOR TORSIONAL RESISTANCE = 1.40 SQ.CM.

Tutorial 2-93

```
AT END SUPPORT - Vu= 19.11 KNS Vc= 71.87 KNS Vs= 0.00 KNS
Tu= 4.73 KN-MET Tc= 2.3 KN-MET Ts= 6.3 KN-MET LOAD 5
  STIRRUPS ARE REQUIRED FOR TORSION.
  REINFORCEMENT FOR SHEAR IS PER CL.11.5.5.1.
  PROVIDE 12 MM 2-LEGGED STIRRUPS AT 130. MM C/C FOR 2705. MM
  ADDITIONAL LONGITUDINAL STEEL REQD. FOR TORSIONAL RESISTANCE = 1.40 SQ.CM.
                                                   3J_
 _____6J__
             _____ 6000x 275x 350____
||=========
                                              _____|
                                      ЗNo12 Н 306.4890.ТО 6000 |
3No12 H 306. 0.TO 860
| 22*12c/c130
                                              22*12c/c130
  2No16 H 51. 313.TO 5437
                                               1 1 1 1
                                                   ___
   000
                                    1
                                                1
                                                    000
         1
                        1
| 3#12
                                                | 3#12
                                    1
         1
                                    1
                                                - 1
т
         1
            - 1
                     .
                                 .
                                             | 2#16
                                    | 2#16
            | 2#16
                     1
                                 1
                                              Т
                                                 Т
                   1
              00
                        1 00
                                 1
                                    1
                                      00
                                              Т
                                                 Т
         1
            1
                     Т
                                 Т
                                    1
                                                 1
                                    Т
 55. DESIGN COLUMN 1 3 4
 _____
                _____
     COLUMN NO.
                1 DESIGN PER ACI 318-02 - AXIAL + BENDING
 FY - 415.0 FC - 25.0 MPA, RECT SIZE - 275.0 X 300.0 MMS, TIED
       AREA OF STEEL REQUIRED = 882.8 SQ. MM
 BAR CONFIGURATION
                  REINF PCT. LOAD LOCATION PHI
 _____
                                           ____
  8 - 12 MM
                    1.097
                           4
                                   END 0.650
 (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE)
 TIE BAR NUMBER 8 SPACING 192.00 MM
    COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET)
  _____
   PO Pn max P-bal. M-bal. e-bal. (MM)
  2109.38 1687.50 785.43
                         97.75
                                 124.5
   M0
          P-tens. Des.Pn Des.Mn
                                  e/h
   47.51 -375.48
                          51.21 0.19497
                  75.04
  _____
    COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET)
  PO Pn max P-bal. M-bal. e-bal. (MM)
2109.38 1687.50 775.80 88.78 114.4
   M0
         P-tens. Des.Pn Des.Mn
                                e/h
   43.01 -375.48 75.04
                          3.92
                                0.01492
      _____
                                       _____
```

COLUMN NO. 3 DESIGN PER ACI 318-02 - AXIAL + BENDING FY - 415.0 FC - 25.0 MPA, CIRC SIZE 350.0 MMS DIAMETER TIED AREA OF STEEL REQUIRED = 1096.8 SO. MM BAR CONFIGURATION REINF PCT. LOAD LOCATION PHI -----10 - 12 MM 1.176 4 STA 0.650 (EQUALLY SPACED) TIE BAR NUMBER 8 SPACING 192.00 MM COLUMN INTERACTION: MOMENT ABOUT Z/Y -AXIS (KN-MET) _____ P0 Pn max P-bal. M-bal. e-bal. (MM) 2489.81 1991.85 939.46 109.84 116.9 M0 P-tens. Des.Pn Des.Mn e/h 62.23 -469.35 153.06 73.18 0.09660 _____ COLUMN NO. 4 DESIGN PER ACI 318-02 - AXIAL + BENDING FY - 415.0 FC - 25.0 MPA, RECT SIZE - 275.0 X 300.0 MMS, TIED AREA OF STEEL REQUIRED = 1056.0 SQ. MM BAR CONFIGURATION REINF PCT. LOAD LOCATION PHI -----5 4 - 20 MM 1.523 STA 0.650 (PROVIDE EQUAL NUMBER OF BARS ON EACH FACE) TIE BAR NUMBER 8 SPACING 192.00 MM COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KN-MET) _____ P0 Pn max P-bal. M-bal. e-bal. (MM) e-bal 161.3 2247.93 1798.34 764.68 123.35 161.3 M0 P-tens. Des.Pn Des.Mn e/h 63.35 -521.51 43.92 10.50 0.06829 COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KN-MET) _____ Pn max P-bal. M-bal. e-bal. (MM) P0 2247.93 1798.34 755.33 111.44 147.5 M0 P-tens. Des.Pn Des.Mn e/h 57.35 -521.51 43.92 47.50 0.30898 _____ 56. END CONCRETE DESIGN

Tutorial 2 **2-95**

	******	*** END OF THE	STAAD.Pro	RUN *********	
	**** D2	ATE=	TIME=	****	
**	*****	******	*******	*****	• *
*				, please contact	*
*	Research	Engineers Off	ices at th	e following locations	*
*		malanhana		Res 41	*
*	USA:	Telephone +1 (714)974-2	500	Email support@reiusa.com	*
*	CANADA	+1 (905)632-4	771	detech@odandetech.com	*
*	CANADA	+1 (604)629 6	087	staad@dowco.com	*
*	UK	+44 (1454) 207-		support@reel.co.uk	*
*		+33(0)1 64551 +49/931/40468		<pre>support@reel.co.uk info@reig.de</pre>	*
*		+49/931/40468		staad@edr.no	*
*	SINGAPORE	+65 6225-6015	/16	support@reiasia.net	*
*	INDIA	+91 (033) 2357-	3575 suppo	ort@calcutta.reiusa.com	*
*	JAPAN	+81 (03) 5952-6	500	eng-eye@crc.co.jp	*
*	CHINA	+86(411)363-1	983	support@reiasia.net	*
*	North Ame	rica		support@reiusa.com	*
*	Europe			support@reel.co.uk	*
*	Asia			upport@reiasia.net	*

2.11 Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the *Post Processing* Mode. The *Post Processing* mode is used to verify the analysis and design results and generate reports.

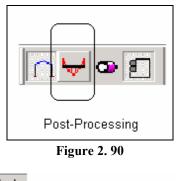
For this tutorial problem, we shall perform the following tasks:

- Display deflection diagrams
- Annotate Displacements
- Change the Display units for displacement values shown in the tables.
- Switching between load cases for viewing deflection diagrams.
- Display the force and moment diagrams.
- Changing the degree of freedom for which the force/moment diagram is plotted
- Annotating the force diagram
- Changing the Display units for the force and moment values shown in the tables.
- Restricting the load cases for which results are viewed
- Using Member Query
- Viewing Concrete Design results using Query.
- Producing an on-screen report
- Taking pictures
- Creating Customized Reports

2.11.1 Going to the post-processing mode

Steps:

1. At the end of section 2.9, we saw how one could go directly from the Analysis window to the post-processing screen. However, the formal method of accessing the *Post Processing* mode is either by clicking on the **Post-Processing** icon from the top toolbar or from the *Mode* menu as shown in the figures below.



g
ocessing 📐
ve Designs 👘 🕨 🕨
eck Preprocessor
ontrol

Figure 2.91

2-98 Tutorial 2

2. The *Results Setup* dialog box appears as shown below. Select the load cases for which to display the results. For our case, let us select all the load cases. Then click on the **OK** button.

Results Setup	x
Loads Range Result View Options	
Load Cases	
Available:	Selected:
	1 DEAD LOAD 2 LIVE LOAD
	>> 3 WIND LOAD 4 DEAD + LIVE
	5 DEAD + WIND
	<
	OK Apply Help

Figure 2.92

2.11.2 Viewing the deflection diagram

<u> </u>		reframe.st Tools Sek		Report	Mode	Window H	elp									×
1	-	₽a % I	a X 🗅	± <u>C</u> ±	n (> ∥⊜	🗟 📾 🕮 🗖	¥ 🚳 🗋	🗖 📥 🏭	i + + 🛃 🗂	H 🗞 🖗	? 👱 🥂] 🔄 🔗 📢	2 🕫	
60 60	Ø Ø	1 🗗 🗇 I	7	$\Phi \Phi$	0 C	, 🔶 🛛 🖽	@, €,	Q Q (Q (R 🔍 🖻 g	🕈 🔾 📣 🛙	: DEAD LOAD		• ?	•		
14 4	6 6	111 M. I	2 P B	2 °C	TR	• 📖 🗍 🍝	× 44	G		Hz_ 🕞 🖀					1	
<u> </u>	і — Іс					11		1.11				1]] · · [• [⊂	-	
Vode	Ħ	A reframe	e.std - Wha	le Struct	ure			rame.std - No		ents:						<u> </u>
NC NC	Displacement								Horizontal	Vertical	Horizontal	Resultant		Rotational		
🐹 —— I	Ba						Node	L/C	X	Y	Z	noounun	rX	rY	rZ	П
Beam									mm	mm	mm	mm	rad	rad	rad	
	M						1	1 DEAD LOA 2 LIVE LOAD	0.000	0.000	0.000	0.000	0.000	0.000	0.000	
* –	-							3 WIND LOAD	0.000	0.000	0.000	0.000	0.000	0.000	0.000	
	- <mark> </mark> - Reactions							4 DEAD + LIV	0.000	0.000	0.000	0.000	0.000	0.000	0.000	
Animation	act							5 DEAD + WIN	0.000	0.000	0.000	0.000	0.000	0.000	0.000	j
m Ĕ	2						2	1 DEAD LOA	-0.069	-0.043	0.406	0.414	0.000	0.000	-0.001	
🌢 🥈								2 LIVE LOAD	-0.064	-0.034	0.377	0.384	0.000	0.000	-0.001	
								3 WIND LOAD	1.109	0.003	0.325	1.155	0.000	-0.000	-0.000	
\$/ <u> </u>					n .			4 DEAD + LIV	-0.179	-0.102	1.061	1.081	0.001	0.000	-0.002	
불								5 DEAD + WIN	1.373	-0.044	0.874	1.628	0.000	-0.000	-0.001	
Reports			1			2	3	1 DEAD LOA	-0.084	-0.070	0.084	0.138	0.001	-0.000	0.001	
- L		2			3	×		TOTIMETON	0.078	n neg	0.078	. 1195.	11.11.04 :	0.000:	<u>•</u>	
AF.						6	TTT ref	rame.std - Bea	am Relative D)isplacemen	t Detail:				_ 0	×
		1		E	<u> *</u>		ৰিৰ	All Re	lative Displac	ement / Ma	ax Relative D	Displacemen	ts/			
							Bear	LC	Dist	×	У	z	Resultant			
		1					Веап		mm	mm	mm	mm	mm			
9							1	1 DEAD LOA	0.000	0.000	0.000	0.000	0.000			-
						2			875.000	-0.129	-0.000	-0.079	0.151			
50									1750.000	-0.353	-0.000	-0.109	0.370			
5									2625.000	-0.401	-0.000	-0.085	0.410			
<u> </u>								2 LIVE LOAD	3500.000	0.000	0.000	0.000	0.000			
6								2 LIVE LUAD	875.000	-0.120	-0.000	-0.073	0.000			
3									1750.000	-0.120	-0.000	-0.101	0.140			
									2625.000	-0.373	-0.000	-0.079	0.344			
									3500.000	0.000	0.000	0.000	0.000			
								3 WIND LOAD	0.000	0.000	0.000	0.000	0.000			
		Ĭ _z x							875.000	-0.067	-0.000	-0.050	0.083			
									1750.000	0.055	-0.000	-0.052	0.076			
		Ž.		Load	1 : Dis	placement										
		Σ		Load	1 : Dis	placemen			000.000	0.000	0.000	0.002	0.010			-

The screen will now look like the figure shown below.

Figure 2.93

The diagram currently on display is the node deflection diagram for load case 1 (DEAD LOAD). The title at the bottom of the diagram is indicative of that aspect. If you, say, wandered off into any other result diagram, and wanted to get back to the deflection diagram, just select the *Node* | *Displacement* tab along the page control area on the left side.

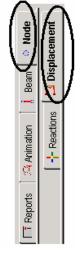


Figure 2.94

The option for selecting the deflection diagram is available from another facility also - the *Results* | *Deflection* menu option - as shown below.

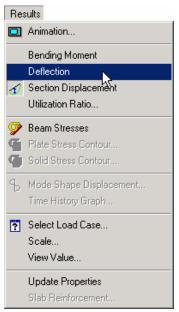


Figure 2.95

2.11.3 Switching between load cases for viewing the deflection diagram

Steps:

1. To change the load case for which to view the deflection diagram, you may click in the list box called **Active Load** and choose the one you want.



Figure 2.96

2. Alternatively, either click on the *Symbols and Labels* icon or, go to *View* | *Structure Diagrams* menu option as shown below.

	View	
	Zoom	•
	🛫 Pan	
	View Selected Objects Only	
	✓ Always <u>Fit</u> in Current Window	
	Tables	
	🔀 Whole Structure	
	Ogientation	F4
	🔁 Structure Diagrams	
	Open View	
	<u>N</u> ew View	
	View <u>M</u> anagement	•
	Tool <u>b</u> ars	
	Option <u>s</u>	
\bigcirc	Set Structure <u>C</u> olors	
🛅 📥 🏭 🕶 🛃 📩 🛏 🧏 💹 ? 鯹 🥂	Structural Tool Tip Options	
🗖 📥 🏭 🕶 🛃 🕇 🛏 🧏 🚿 ? 🕍 🥂	📣 3D Rendering	
Symbols and Labels	Refresh	F5

Figure 2.97

2-102 Tutorial 2

3. In either case, the *Diagrams* dialog box comes up. Select the **Loads and Results** tab and choose the desired load case from the *Load Case* list box. Then, click on **OK**.

Force Limits Structure Loads an	Animation d Results	De Scales	sign Results Labels
oad Case: 3: WIND LOAD			
Show Load Arrow		Hati	rces Diagram — ch C Fill ine
Mode Shape Mode Shape: None 💌	🗌 Mod	le Shape 📃	
Beam Forces			
🗆 Axial 🛛 C 📕 T 🔜		From	To 🔺
O Diagram O Color C/T	1	< -11.9982	-11.9982
Intensity		-11.9982	-10.4985
	4	-10.4985 -8.99868	-8.99868
Minimum: -11.9982	5	-7.4989	-5.99912
11,0000	6	-5.99912	-4,49934
Maximum: 11.9982	7	-4.49934	-2.99956
No of values: 16	8	-2.99956	-1.49978
	9	-1.49978	0
	10	0	1.49978
🔲 Shearyy 📃 🗖 Bending	עע אַ 📕 🚺	1.49978	2.99956
		2.99956	4.49934
🔲 🔚 Shearzz 🛛 📕 🗖 Bending	13	4.49934	5.99912
		5.99912	7.4989 🗣
Torsion	14		
	<u>14</u> ◀		•

Figure 2.98

Tutorial 2-103

The diagram below shows the deflected shape of the structure for load case 3.

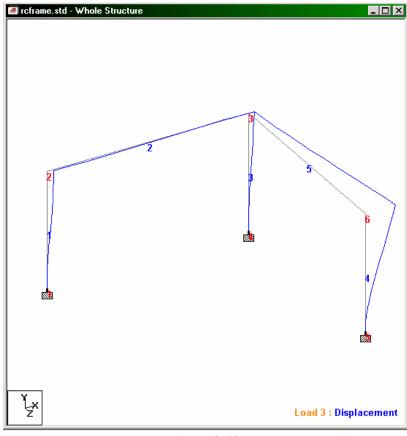


Figure 2.99

4. To display the deflection for say, load case 5 (DEAD + WIND), follow step 1 or 2 and select load case 5.

The deflection of Load Case 5 will now displayed on the model as shown below.

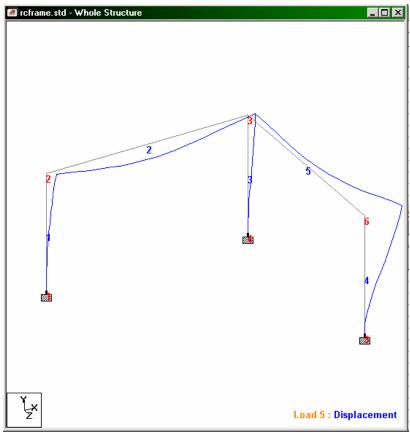


Figure 2. 100

2.11.4 Changing the size of the deflection diagram

Steps:

If the diagram appears too imperceptible, it may be because it may be drawn to too small a scale. To change the scale of the deflection plot, you may

a) click on the Scale icon



Figure 2. 101

b) choose Scale from the Results menu



Figure 2. 102

c) go to View | Structure Diagrams | Scales menu option.

- I.	ς Γ	A 1 11	1 5	
Force Lim	··· , 1	Animation	Des Scales	ign Results
Structure	Loads a	nd Results	Scales	Labels
Load Scales				pply Immediately
Point Force:	17.5127	Npermm		
Dist. Force	0.409931	lbf/ft per mm		
Point.Moment:	0.00393701	kip⁻in permm		
Dist. Moment:	196.85	lb ⁻ in/in per mm		
Pressure:	1.9685	psipermm		
Result Scales				
Bending Y :	7.87402	kip⁻in permm		
Bending Z :	7.87402	kip⁻in permm		
Shear Y:	175.127	N permm		
Shear Z:	175.127	Nipermm		
Axial:	875.634	N permm		
Torsion:	0.0393701	kip⁻in permm		
Displacement:	0.5	mm per mm		
Beam Stress:	196.85	psipermm	S	ave As Default
Mode Shape:	0.00393701	1	B	eset To Default
Icon Size				
Support I con :	3	🖌 % Master	Block : 3	× %

All of the above will bring up the following dialog box.

Figure 2. 103

In the *Displacement* field, specify a smaller number than what is currently listed, and click on **OK**. The deflection diagram should now be larger.

In the above dialog box, if you switch on the check box *Apply Immediately*, pressing the up or down arrow keys alongside the number will produce immediate results in terms of a smaller or a larger diagram depending on whether you click the up or the down arrow keys.

Di	iagram s						×
	Force Limi	its	Animation		7	Results	
	Structure	Loads ar	nd Results	Scale	<u></u>	Labels	
	Load Scales				🔽 App	ly Immediately	
	Point Force:	17.5127	N per mm				
	Dist. Force	0.409931 🚔	lbf/ft per mm				
	Point.Moment:	0.00393701 🚔	kip⁻in permm				
	Dist. Moment:	196.85	lb⁻in∕in per mr	n			
	Pressure:	1.9685	psi per mm				
	Bending Y:	7.87402	kip⁻in permm				
	Bending Z :	7.87402	kip⁻in permm				
	Shear Y:	175.127	N per mm				
	Shear Z:	175.127	N per mm				
	Axial:	875.634	N per mm				
	Torsion:	0.0393701	kip⁻in permm				
C	Displacement:	0.5	mm per mm				.
	Beam Stress:	196.85 🚔	psi per mm		Save	e As Default	
	Mode Shape:	0.00393701			Rese	t To Default	1
	Cloon Size						
	Support Icon :	3	Na: ▼% Ma:	ster Block :	3	▲ %	
		OK	Canc	el /	Apply	Help	

Figure 2. 104

2.11.5 Annotating displacements

Annotation is the process of displaying the displacement values on the screen.

Steps:

1. Select the View Value option from the *Results* menu.



Figure 2. 105

2. The following dialog box comes up. From the *Ranges* tab, select **All** nodes. If you wish to annotate deflection for just a few nodes, specify the node numbers in the node list.

XII Annotation	- rcframe	.std - V	√hole Sti
Ranges Bear	m Results	Node	Reaction
C None			
O All			
C View			
C Group			
C Property			
Ranges	Nodes:	23	
	Beams:		

2-109

Tutorial 2

Figure 2. 106

We will annotate the results for all nodes. So, keep the button on **All**.

From the *Node* tab, check the **Resultant** option. Resultant stands for the square root of sum of squares of values of X,Y and Z displacements. Click the **Annotate** button and notice that the values appear on the structure. Click the **Close** button to close the dialog box.



Figure 2. 107

The following figure shows the annotated deflection diagram for load case 2.

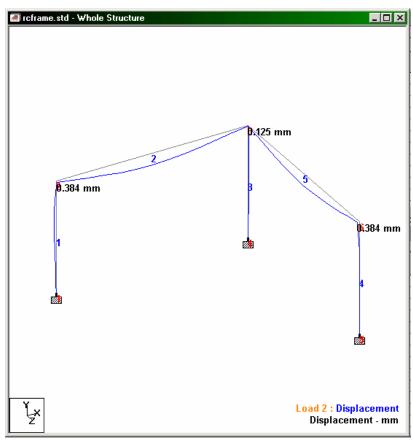


Figure 2. 108

2.11.6 Changing the units in which displacement values are annotated

The units in which displacement values are displayed in the postprocessing mode are referred to as the display units.

Steps:

- 1. Display units may be modified by using any one of the following methods:
 - a. by clicking on the Change Graphical Display Unit icon,

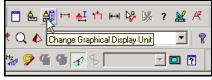


Figure 2. 109

b. by going to Tools | Set Current Display Unit menu option

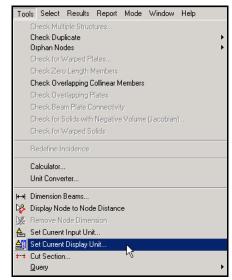


Figure 2. 110

c. by selecting the *View* | *Options* menu option.

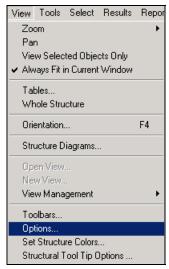


Figure 2. 111

2. In the *Options* dialog box that comes up, select the **Structure Units** tab. Change the *Dimensions* of **Displacement** from *Millimeter* to say, **cm** or *inches* or anything else you desire, and click on **OK**.

Options		×
View Highlight	Structure Units	
Tolerance Tables	Dimensions	
Assign Dlgs Node Labels	Dimension: mm 💌 Show 3 曼 dec places	
Beam Labels	Displacement: cm Show 3 🖨 dec places	
Solid Labels	Springs	
Load Labels	Linear kN/mm Show 3 🖨 dec places	
Structure Units Section Units	Rotational kN⁻m/de Show 3 🖨 dec places	
Force Units Dimension	Other	
	Temperature C Show 3 dec places	
	Weight kg Show 3 🖨 dec places	
	OK Apply Cancel Help	2

Figure 2. 112

Tutorial 2-113

🛲 rcframe.std - Whole Structure _ 🗆 × **0.**013 cm 0.038 cm **6**038 cm 124 动 Load 2 : Displacement Displacement - cm

The diagram will be updated to reflect the new units.

Figure 2. 113

2.11.7 The Node Displacement Table

Upon entering the Post-Processing mode, the first screen that we came across is shown below.

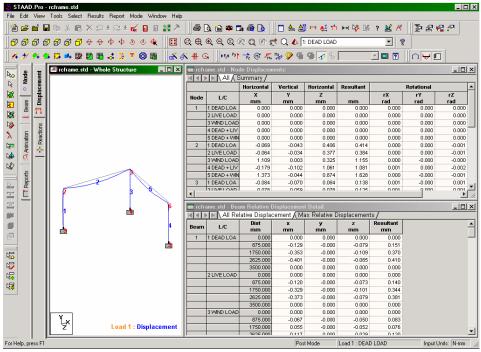


Figure 2. 114

For the *Node* | *Displacement* page on the left side, notice that there are 2 tables displayed along the right side. The upper table, called the Node Displacements table, lists the displacement values for every node for every selected load case. Load cases may be selected or de-selected for the purpose of this table from the *Results* | *Select Load Case* menu. (See section 2.11.16 for details) The lower table is called the Beam relative displacement table.

If you happen to close down any of these tables, you can restore them from the *View* | *Tables* menu.

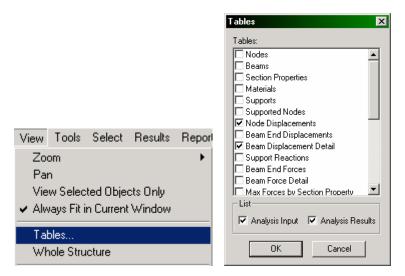


Figure 2. 115

The *Node Displacement* table window has two tabs: *All* and *Summary* (see figure below).

ĺ	III rcframe.std - Node Displacements:							
Horizontal Vertical Horizontal Result								
I	Node	L/C	X	Y	Z			
Ш			сm	CM	cm	cm		
I	1	1 DEAD LOA	0.000	0.000	0.000	0.000		
Ш		2 LIVE LOAD	0.000	0.000	0.000	0.000		

Figure 2. 116

All - This tab presents all nodal displacements in tabular form for all load cases and all degrees of freedom.

ncfr	ame.std - No		ents:					_ [
All & Summary / Horizontal Vertical Horizontal Resultant Rotational											
Node	L/C	X cm	Y cm	Z cm	cm	rX rad	r¥ rad	rZ rad			
1	1 DEAD LOA	0.000	0.000	0.000	0.000	0.000000	0.000000	0.000000			
	2 LIVE LOAD	0.000	0.000	0.000	0.000	0.000000	0.000000	0.000000			
	3 WIND LOAD	0.000	0.000	0.000	0.000	0.000000	0.000000	0.000000			
	4 DEAD + LIV	0.000	0.000	0.000	0.000	0.000000	0.000000	0.000000			
	5 DEAD + WIN	0.000	0.000	0.000	0.000	0.000000	0.000000	0.000000			
2	1 DEAD LOA	-0.007	-0.004	0.041	0.041	0.000249	0.000062	-0.000808			
	2 LIVE LOAD	-0.006	-0.003	0.038	0.038	0.000232	0.000058	-0.000751			
	3 WIND LOAD	0.111	0.000	0.032	0.116	0.000120	-0.000063	-0.000063			
	4 DEAD + LIV	-0.018	-0.010	0.106	0.108	0.000650	0.000163	-0.002095			
	5 DEAD + WIN	0.137	-0.004	0.087	0.163	0.000432	-0.000013	-0.000971			
3	1 DEAD LOA	-0.008	-0.007	0.008	0.014	0.000746	0.000000	0.000746			
	2 LIVE LOAD	-0.008	-0.006	0.008	0.013	0.000694	-0.000000	0.000694			
	3 WIND LOAD	0.110	-0.000	-0.006	0.111	-0.000004	0.000355	-0.000336			

Figure 2. 117

Summary - This tab, shown in the figure below, presents the maximum and minimum nodal displacements (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

			Horizontal	Vertical	Horizontal	Resultant		Rotational	
	Node	L/C	X cm	Y cm	Z cm	cm	rX rad	rY rad	rZ rad
Max X	6	5 DEAD + WIN	0.675	-0.005	-0.000	0.675	-0.000904	0.000866	-0.002009
Min X	6	4 DEAD + LIV	-0.106	-0.010	0.018	0.108	-0.002095	-0.000163	0.000650
Max Y	2	3 WIND LOAD	0.111	0.000	0.032	0.116	0.000120	-0.000063	-0.000063
Min Y	3	4 DEAD + LIV	-0.022	-0.017	0.022	0.036	0.001937	0.000000	0.001937
Max Z	2	4 DEAD + LIV	-0.018	-0.010	0.106	0.108	0.000650	0.000163	-0.002095
Min Z	3	3 WIND LOAD	0.110	-0.000	-0.006	0.111	-0.000004	0.000355	-0.000336
Max rX	3	4 DEAD + LIV	-0.022	-0.017	0.022	0.036	0.001937	0.000000	0.001937
Min rX	6	4 DEAD + LIV	-0.106	-0.010	0.018	0.108	-0.002095	-0.000163	0.000650
MaxrY	6	5 DEAD + WIN	0.675	-0.005	-0.000	0.675	-0.000904	0.000866	-0.002009
Min rY	6	4 DEAD + LIV	-0.106	-0.010	0.018	0.108	-0.002095	-0.000163	0.000650
MaxrZ	3	4 DEAD + LIV	-0.022	-0.017	0.022	0.036	0.001937	0.000000	0.001937
Min rZ	2	4 DEAD + LIV	-0.018	-0.010	0.106	0.108	0.000650	0.000163	-0.002095
Max Rst	6	5 DEAD + WIN	0.675	-0.005	-0.000	0.675	-0.000904	0.000866	-0.002009

Figure 2. 118

For the Beam Relative Displacement table, the details are as follows :

All

The *All* tab presents the displacements of members at intermediate section points. All specified members and all specified load cases are included. The table shows displacements along the local axes of the members, as well as their resultants.

Max Displacements

The *Max Displacements* tab presents the summary of maximum sectional displacements (see figure below). This table includes the maximum displacement values and location of its occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the member to the resultant maximum section displacement of the member.

					isplacement					_
Beam	L/C	Length m	Max x ft	Dist m	Maxy ft	Length m	Max z ft	Dist m	Max ft	
	1 DEAD LOA	3.500	-0.001	2.333	-0.000	3.208	-0.000	0.500	0.001	
	2 LIVE LOAD	3.500	-0.001	2.333	0.000	2.625	-0.000	0.500	0.001	
	3 WIND LOAD	3.500	0.000	2.625	-0.000	3.208	-0.000	0.417	0.000	
	4 DEAD + LIV	3.500	-0.004	2.333	-0.000	3.208	-0.001	0.500	0.004	
	5 DEAD + WIN	3.500	-0.001	2.042	0.000	3.208	-0.001	0.500	0.001	
2	1 DEAD LOA	6.000	-0.000	4.000	-0.007	3.000	-0.000	0.667	0.007	
	2 LIVE LOAD	6.000	0.000	5.000	-0.006	3.000	-0.000	0.667	0.006	
	3 WIND LOAD	6.000	0.000	5.500	0.001	4.000	0.001	0.583	0.001	
	4 DEAD + LIV	6.000	0.000	5.500	-0.018	3.000	-0.000	0.667	0.018	
	5 DEAD + WIN	6.000	0.000	5.500	-0.007	3.000	0.001	0.583	0.007	
3	1 DEAD LOA	3.500	0.001	1.167	-0.000	0.292	-0.001	0.333	0.002	
	2 LIVE LOAD	3.500	0.001	1.167	0.000	2.625	-0.001	0.333	0.002	
	STAIND LOAD	3 500	-0.001	2 333	-0.000	1 167	0.000	0.750	0.001	

Figure 2. 119

2-118 Tutorial 2

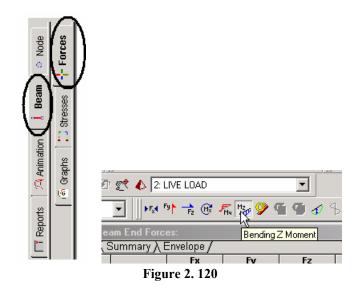
The sub-pages under the Node page are described below in brief.

Page	Sub-Page	Purpose
Node	Displacement	Displays nodal displacements along with tabular results for Node-Displacements and sectional Beam displacements.
	Reactions	Displays support reactions on the drawing as well as in a tabular form.
	Modes	Displays mode shapes for the selected Mode shape number. The eigenvectors are simultaneously displayed in tabular form. This Page appears only for dynamic analyses cases, namely, response spectrum, time history, and if modal calculations are requested.
	Time History	Displays Time history plots, for time history analysis. This sub-page too will appear only if time history analysis is performed.

2.11.8 Displaying force/moment diagrams

Steps:

1. The simplest method to access the facilities for displaying force/moment diagrams is from the *Beam* | *Forces* page along the page control area on the left side of the screen. The bending moment MZ will be plotted by default, evidence of which can be found in the form of the Mz icon show in the diagram below which becomes active.



2-120 Tutorial 2

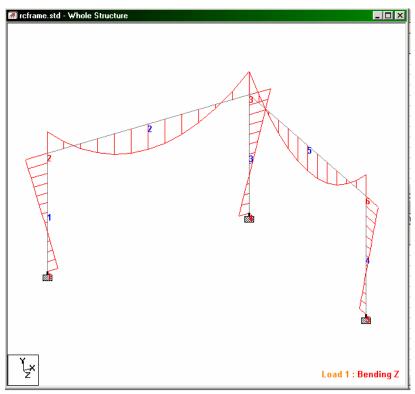


Figure 2. 121

2. The option for selecting the force/moment diagram is available from another facility also - the *Results* | *Bending Moment* menu option - as shown below.

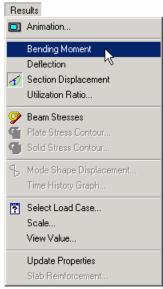


Figure 2. 122

2.11.9 Switching between load cases for viewing the force/moment diagram

Steps:

1. To change the load case for which to view the force/moment diagram, you may click in the list box called Active Load and choose the one you want.



Figure 2. 123

2. Alternatively, either click on the Symbols and Labels icon or, go to *View* | *Structure Diagrams* menu option as shown below.

	View Tools Select Results	Repor				
	Zoom Pan View Selected Objects Only ✔ Always Fit in Current Window	•				
	Tables Whole Structure					
	Orientation	F4				
	Structure Diagrams					
	Open View New View View Management	•				
\square	Toolbars Options Set Structure Colors					
📋 🏯 🏭 🕶 🛃 🕇 🛏 🧏 🚿 ? 🕍 🥂	Structural Tool Tip Options					
Symbols and Labels	Refresh					

Figure 2. 124

3. In either case, the *Diagrams* dialog box comes up. Select the **Loads** and **Results** tab and choose the second load case (LIVE LOAD) from the *Load Case* list box. Also, let us check the **Shear yy** check box. Then, click on **OK**.

Force Limits Anima Structure Loads and Resul	ation Its Si	Des cales	ign Results Lab	els
Load Case: 2: LIVE LOAD			J	
Loads: Direct Deflection			ces Diagran h O Fill ne	n
Mode Shape: None 🔽	🔲 Mode Sha	ape 📕		
Beam Forces				
🗖 Axial C 📕 T 📕		From	То	<u> </u>
C Diagram C Color C/T		11.9982	-11.9982	
Intensity		.9982	-10.4985	
		.4985 19868	-8.99868 -7.4989	
Minimum: -11.9982		989	-7.4989 -5.99912	
		909	-3.99912	
Maximum: 11.9982		9934	-4.43334	
	-	19956	-1.49978	
No of values: 16		9978	0	
	10 0		1.49978	
🔽 Shearyy 📃 🗖 Bendingyy 👱	11 1.4	9978	2.99956	
🔲 Shearzz 📕 🗖 Bending zz 📕	12 2.9	9956	4.49934	
	13 4.4	9934	5.99912	
	14 <mark>5.9</mark> 9	9912	7.4989	-
🗖 Stress 🛛 📕 T 📕	•		•	
				_

Figure 2. 125

2-124 Tutorial 2

4. The figure below shows the shear force diagram for load case 2.

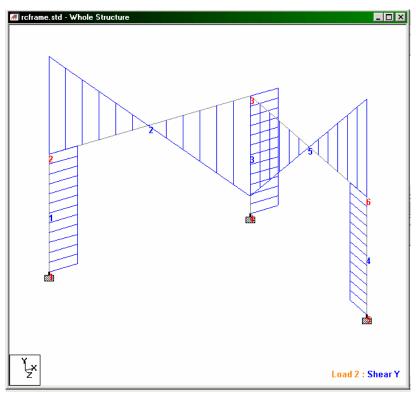


Figure 2. 126

5. To display the bending moment diagram for say, load case 4 (DEAD + LIVE), follow steps 1 to 3 above and select load case 4.

Load 4 : Bending Z

The following diagram should appear in the drawing area:

Figure 2. 127

2.11.10 Changing the size of the force/ moment diagram

Steps:

If the diagram appears too imperceptible, it may be because it may be drawn to too small a scale. To change the scale of the moment plot, you may

a) click on the Scale icon



Figure 2. 128

b) choose Scale from the Results menu

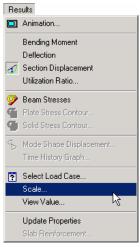


Figure 2. 129

c) go to *View* | *Structure Diagrams* | *Scales* menu option. All of the above will bring up the following dialog box.

Dia	grams				×
	Force Limi	its Loads and	Animation	Design I	Results
	Structure	Loads an			Labeis
	Load Scales			🗌 🗌 Apply	Immediately
	Point Force:	17.5127	N per mm		
	Dist. Force	0.409931	lbf/ft per mm		
	Point.Moment:	0.00393701 🚔	kip⁻in per mm		
	Dist. Moment:	196.85	lb⁻in/in per mm		
	Pressure:	1.9685	psi per mm		
	Result Scales				
Π	Bending Y :	7.87402	kip ⁻ in per mm		
	Bending Z :	7.87402	kip⁻in permm		
	Shear Y:	175.127	N per mm		
	Shear Z:	175.127	N per mm		
	Axial:	875.634	N per mm		
	Torsion:	0.0393701	kip⁻in per mm		
	Displacement:	0.5	mm per mm		
	Beam Stress:	196.85 🚔	psi per mm	Save	As Default
	Mode Shape:	0.00393701		Reset	To Default
	lcon Size				
	Support Icon :	3	% Master Bloc	к: <mark>3</mark>	▲ %
		OK	Cancel	Apply	Help

Figure 2. 130

In the *Bending* field, specify a smaller number than what is currently listed, and click on *OK*. The moment diagram should now be larger.

In the above dialog box, if you switch on the check box *Apply Immediately*, pressing the up or down arrow keys alongside the number will produce immediate results in terms of a smaller or a 2-128 Tutorial 2

larger diagram depending on whether you click the up or the down arrow keys.

Diagrams			×
Force Lim Structure	its Loads an	Animation di Results Scale	Design Results S Labels
Load Scales		(Apply Immediately
Point Force:	17.5127	N per mm	
Dist. Force	0.409931	lbf/ft per mm	
Point.Moment:	0.00393701 🚔	kip⁻in permm	
Dist. Moment:	196.85	lb⁻in/in per mm	
Pressure:	1.9685	psi per mm	
Result Scales			
Bending Y :	7.87402	kip⁻in permm	
Bending Z :	7.87402	kip⁻in permm	
Shear Y:	175.127	N per mm	
Shear Z:	175.127	N per mm	
Axial:	875.634	N per mm	
Torsion:	0.0393701	kip⁻in permm	
Displacement:	0.5	mm per mm	
Beam Stress:	0.5	psi per mm	Save As Default
Mode Shape:	0.00393701		Reset To Default
Licon Size			
Support Icon :	3	X Master Block :	3 🗸 %
,	ОК	Cancel	Apply Help

Figure 2. 131

2.11.11 Changing the degree of freedom for which forces diagram is plotted

Force and moment diagrams can be plotted for 6 degrees of freedom – Axial, Shear-Y, Shear-Z, Torsion, Moment-Y, Moment-Z. One may select or de-select one of more of these degrees of freedom from View | Structure Diagrams | Loads and Results.

agrams					
Force Limits	Anima	ation	De	sign Results	
Structure	Loads and Resul	is 🛛	Scales	Lab	els
Load Case: 2: LIVE	E LOAD			•	
Loads Loads: Direct Moment		· <u> </u>	Hati	orces Diagran ch © Fill line	n
· ,	lone 💌	□ Mod	e Shape 📃		
Beam Forces			From	То	
-		1	< -11.9982	-11.9982	
C Diagram C (Lolor L/ I	2	-11.9982	-10.4985	
Intensity		3	-10.4985	-8.99868	
Minimum: 11	.9982	4	-8.99868	-7.4989	
	.0002	5	-7.4989	-5.99912	
Maximum: 11.	9982	6	-5.99912	-4.49934	
Maximum. 1	0002	7	-4.49934	-2.99956	
No of values: 16		8	-2.99956	-1.49978	
I		9	-1.49978	0	
		10	0	1.49978	
🗌 Shearyy 📃	🗌 Bending yy 📘	11	1.49978	2.99956	
🗖 Shear zz 📕	💌 Bending zz 🛛	12	2.99956	4.49934	
		13	4.49934	5.99912	
Torsion	_	14	5.99912	7.4989	-
🗖 Stress 🛛 🗖	т 💻			<u> </u>	

Figure 2. 132

All degrees of freedom currently plotted will be indicated with a tick mark.

The icons of the Results toolbar may also be used to turn on/off specific degrees of freedom.



For the sake of easy identification, each degree of freedom (d.o.f) has been assigned a different color. One may change the color for that d.o.f. by clicking on the color button alongside the d.o.f, and make a new choice from the color palette.

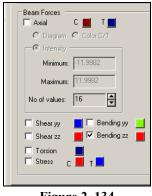


Figure 2. 134

The appearance of the diagram may also be set to one of the 3 - Hatch, Fill or Outline by turning on the relevant option in the dialog box shown earlier.

Beam Forces Diagram
💿 Hatch 🔿 Fill
C Outline

Figure 2. 135

2.11.12 Annotating the force/moment diagram

Steps:

1. Annotation is the process of displaying the force/moment values on the screen. Select the *View Value* option from the *Results* menu.



Figure 2. 136

2. The following dialog box comes up. From the *Ranges* tab, select **All** members. If you wish to annotate the force/moment for just a few members, specify the beam numbers in the beam list.

Ranges	Nodes: Beams: 14
	Remove All Annotate Close

Figure 2. 137

We will annotate the results for all members. So, keep the button on **All.**

2-132 Tutorial 2

From the *Beam Results* tab, check the **Bending - Maximum** option. Click the **Annotate** button and notice that the values appear on the structure. Click the **Close** button to close the dialog box.

📽 Annotation - rcframe.std - Whole Structure
Ranges Beam Results Node Reactions
Bending Ends 🔽 Maximum 🗖 Mid point
Shear Ends Maximum Mid point
Axial
Displacement Max Resultant
Combined Bending and Axial Stress
Ends Maximum Mid point
Remove All Annotate Close

Figure 2. 138

The following figure shows the annotated MZ diagram for load case 5.

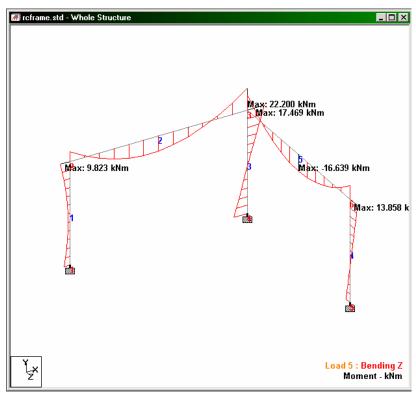


Figure 2. 139

2.11.13 Changing the units in which force/moment values are annotated

Steps:

1. The units in which force and moment values are displayed in the post-processing mode are referred to as the display units.

Display units may be modified by using any one of the following methods:

a. by clicking on the Change Graphical Display Unit icon

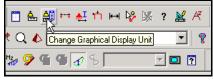


Figure 2. 140

b. by going to Tools | Set Current Display Unit menu option

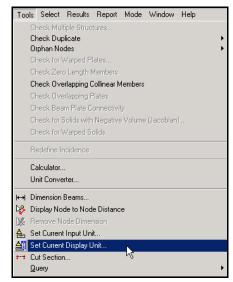


Figure 2. 141

c. by selecting the *View* | *Options* menu option.



Figure 2. 142

2. In the *Options* dialog box that comes up, select the **Force Units** tab. For bending moments, change the **Moment** unit from its current setting to one of the choices available, say, *Mton-m* or *kip-ft* or anything else you desire, and select **OK**.

Options		X
View Highlight	Force Units	
Tolerance	Force	
Assign DIgs	Force N 💌	Show 3 🚔 dec places
Beam Labels	Distr. Force kN/m	Show 3 🖨 dec places
Solid Labels	Moment kip-ft 🗸	Show 3 🖨 dec places
Annotation Structure Units	Distr. Moment kNm/m	Show 3 🖨 dec places
Section Units	Stress N/mm ² -	Show 🛛 🚔 dec places
	Acceleration m/sec ²	Show 3 🖨 dec places
	Rotation rad	Show 3 🚔 dec places
1		Cancel Help

Figure 2. 143

The diagram will be updated to reflect the new units.

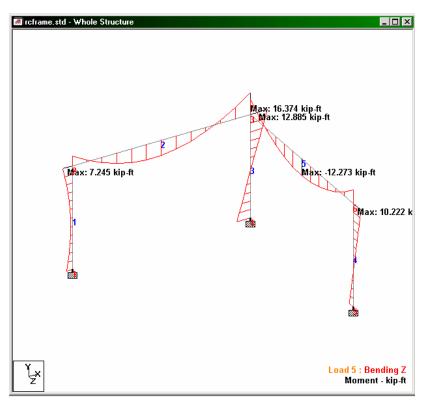


Figure 2. 144

2.11.14 Beam Forces Table

5 STAAD.Pro - File Edit View	rcframe.std Tools Select Results Report Mode Window) ×
12 🖻 😂 🗎] 🖬 ½ 🖻 X Ω ± Ω ± 🖬 🖥	1.99	d 📾 📽 🗖					? 🔟 🥂	🖾 🖉 🛛	S 🖉	
67 67 67 67	0660+++++++++++++++++++++++++++++++++++	🔛 Q 🤁	e , e, 🔍 🦉	2 🔍 🖻 g	🕈 🔍 📣 🛽	DEAD LOAD		• ?			
14 9 94 9	4 🗱 🍂 😰 🖻 📲 🎝 🔅 T 🕲 🗃	a (A PEA FA	🕇 🛞 🎢	🛸 🎐 🖷	9 🕢 🖇		- 💷 🏹		5	
8 8	🕷 rcframe.std - Whole Structure		ame.std - Bear								×
A ode Forces			All (Si	ummary λ E							
State Node		Beam	L/C	Node	Fx	Fy N	Fz N	Mx kip-ft	My kip-ft	Mz kip-ft	H
		1	1 DEAD LOA	1	26018.461	-5534.667	96.331	-0.117	0.477	-4.827	1
8 gg 8				2	-18799.713	5534.667	-96.331	0.117	-0.726	-9.461	
Aresses			2 LIVE LOAD	1	17477.987	-5145.387	90.001	-0.109	0.442	-4.487	-
a 50				2	-17477.987	5145.387	-90.001	0.109	-0.675	-8.796	
11 ₁₂			3 WIND LOAD	1	-1419.712	8876.163	-363.155	0.118	0.757	7.208	
Animation				2	1419.712	1420.820	363.155	-0.118	0.180	2.415	
Arima 🖉 🖉			4 DEAD + LIV	1	57441.387	-14363.648	241.543	-0.307	1.255	-12.529	
a 🕺 🖉				2	-48778.893	14363.648	-241.543	0.307	-1.879	-24.551	
- <u>M</u> (6	∧		5 DEAD + WIN	1	26763.197	5469.768	-372.356	0.025	1.522	4.088	
	1.17			2	-18822.571	7916.312	372.356	-0.025	-0.560	-7.245	
18		2	1 DEAD LOA	2	5534.230	18799.736	99.211	-0.726	-0.117	9.461	
				3	-5534.230	19173.725	-99.211	0.726	-0.322	-10.289	
Li Reports		1 💷		2	5145 070	17478.008	92.130	0.675	0.109	8 796	Ĺ
			ame.std - Bear								×
			M AIL M	ax Axial Fo	rces)∖ Max I	Bending Mo	ments ∖ Max	Shear Force	is/		
		Beam	L/C	Dist mm	Fx N	Fy N	Fz N	Mx kip-ft	My kip-ft	Mz kip-ft	-
1	6		1 DEAD LOA	0.000	26018.461	-5534.667	96.331	-0.117	0.477	-4.827	
	E	為 │		875.000	24213.774	-5534.667	96.331	-0.117	0.539	-1.255	
				1750.000	22409.086	-5534.667	96.331	-0.117	0.601	2.317	
				2625.000	20604.399	-5534.667	96.331	-0.117	0.663	5.889	
				3500.000	18799.713	-5534.667	96.331	-0.117	0.726	9.461	
			2 LIVE LOAD	0.000	17477.987	-5145.387	90.001	-0.109	0.442	-4.487	
				875.000	17477.987	-5145.387	90.001	-0.109	0.500	-1.167	
				1750.000	17477.987	-5145.387	90.001	-0.109	0.559	2.154	
				2625.000	17477.987	-5145.387	90.001	-0.109	0.617	5.475	
				3500.000	17477.987	-5145.387	90.001	-0.109	0.675	8.796	
			3 WIND LOAD	0.000	-1419.712	8876.163	-363.155	0.118	0.757	7.208	
	z Load 1 : Bendin	- 1		875.000	-1419.712	6301.917	-363.155	0.118	0.523	2.310	
	Load 1 : Bendin	ig Z		1750.000	1/10 710	3707 671	363 155	0.118	0.288	n 997 •	
r Help, press F1					Post	Mode	Load 1 : DEAD	LOAD	Input	Jnits: N-mm	

When we select the *Beam* | *Forces* page from the page control area on the left side, the screen that appears is shown below.

Figure 2. 145

The axial forces and shear forces, bending and torsional moments in all selected beams for all selected load cases are displayed in a tabular form along the right half of the screen. Load cases may be selected or de-selected for the purpose of this table from the *Results* | *Select Load Case* menu. (See section 2.11.16 for details) If you happen to close down any of these tables, you can restore them from the *View* | *Tables* menu.

		Tables X
		Tables: Section Properties Materials Supports Supported Nodes Node Displacements Beam End Displacements Beam Displacement Detail Support Reactions
View Tools Select Results R	lepoi	Beam End Forces
Zoom Pan View Selected Objects Only ✓ Always Fit in Current Window	•	Max Forces by Section Property Plate Centre Stress Solid Centre Stress List Analysis Input Analysis Results
Tables Whole Structure		OK Cancel

Figure 2. 146

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

Figure 2. 147

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

	All / St	nninary A L	.invelope /					
Bearn	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	22.948	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

Figure 2. 148

Summary - This tab, shown in the next figure, presents the maximum and minimum values (forces and moments) for each degree of freedom. All beams and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom are presented with the corresponding Node of occurrence and Load Case number (L/C).

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 M×	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	

Figure 2. 149

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

II reframe.std - Beam End Forces: □ □ D								
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
						3 WIND LOAD		
		-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			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

Figure 2. 150

2.11.15 Viewing the force/moment diagrams from the Beam | Graphs page

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 left side of the screen as shown below.

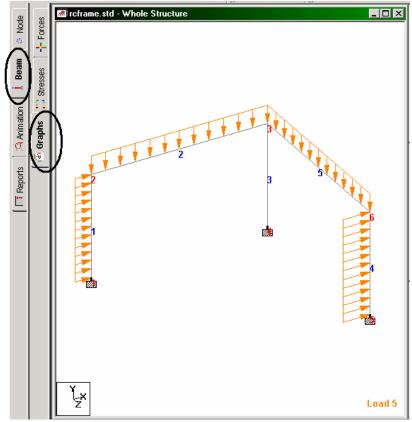


Figure 2. 151

The Main Window area of the screen shows the loading on the structure. On the right side of the screen, the force/moment diagrams appear (see figure below). When we highlight a member 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 1 for load case 4.

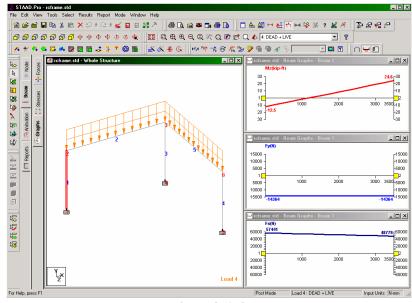


Figure 2. 152

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

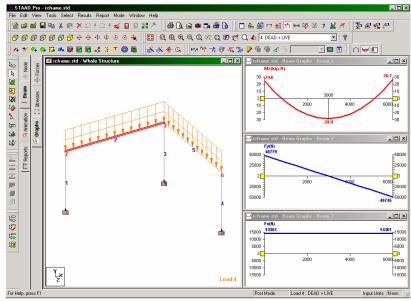


Figure 2. 153

We can change the degree of freedom for which results are plotted by doing the following. Go into one of the 3 drawing windows on the right side, and click the right mouse button. The following dialog box will appear.



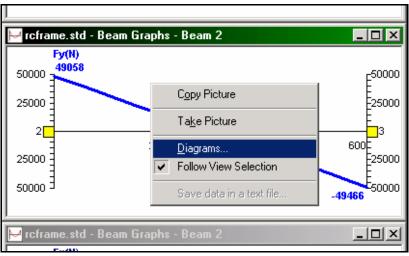


Figure 2. 154

Select **Diagrams**. In the dialog box which appears, switch on/switch off the degrees of freedom you desire.

Diagram 🗙
Forces
🗖 Bending yy 🗾 🗖 Shearzz 📕
🗖 Bending zz 📕 🗖 Shearyy 📃
🔽 Torsion 📃 🗖 Axial 📃
🗖 Stress 📃
OK Cancel Help
E' 0 155

Figure 2. 155

Click on **OK** and that degree of freedom will be plotted in that window.

2.11.16 Restricting the load cases for which results are viewed

Steps:

1. To restrict the load cases for which results are viewed, either click

on the **Results Setup** icon or, go to *Results* | *Select Load Case* menu option as shown below.



Figure 2. 156

2-146 Tutorial 2

2. In the *Results Setup* dialog box that comes up, let us first un-select the already selected load cases by clicking on the \checkmark button.

Results Setup		x
Loads Range Result View Options		_
Load Cases		
Available:	Selected:	
1 DEAD LOAD 2 LIVE LOAD 3 WIND LOAD 4 DEAD + LIVE 5 DEAD + WIND	> >>	
	<	
	OK Apply Help	

Figure 2. 157

3. Select load cases 1 (DEAD LOAD) and 3 (WIND LOAD) by holding the 'Ctrl' key down. Then, click on the → button. After the load cases have been selected, click on the **OK** button.

Results Setup	X
Loads Range Result View Options	
Load Cases	
Available: 2 LIVE LOAD 4 DEAD + LIVE 5 DEAD + WIND	Selected: > 1 DEAD LOAD 3 WIND LOAD <
	OK Apply Help

Figure 2. 158

2.11.17 Using Member Query

Member query is a facility where several results for specific members can be viewed at the same time from a single dialog box. It is also a place from where many of the member attributes such as the property definition, specifications (releases, truss, cable, etc.) and beta angle can be changed for input purposes.

Steps:

To access this facility, first select the member. Then, either go to **Tools** | **Query** | **Member** menu option or, double-click on the member. Let us try double-clicking on member 4.

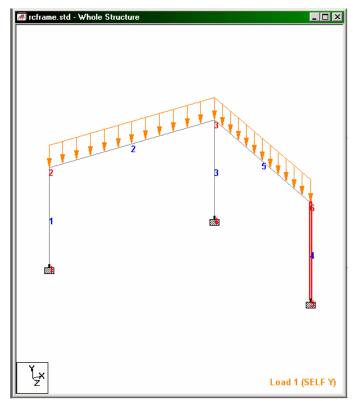


Figure 2. 159

As we double-click on member 4, the following dialog box comes up. Let us take a look at the *Property* tab.

rcframe	.std - Beam					×				
Geometry	Property L	oading	Shear Bending Deflec	tion Cor	icrete Design					
		Beam	n no. = 4. Section: Rect	0.30×0.28	1					
	300.000									
- Dhusian	l Properties (U	_	th = 3500		2	75.000				
Ax	182500	inic ning-	9.49335e+00							
Ay	82500	ly	5.19922e+00							
Az	82500	lz	6.1875e+008	Ass	ssign/Change Property					
D	300	W	275							
	Properties	2 .17	Density(kg/n3) 25 Alpha 0	19.29	MATE Assic	RIAL1 _				
					Print	Close				

Figure 2. 160

The figure above shows where the buttons are located on the member query box. If the member contains output result tabs (Shear/Bending, Deflection, Steel Design, etc.) in the query box, changing member attributes like releases will cause these result tabs to disappear. This is due to the fact that the current output no longer reflects the new input.

NOTE: If you assign or change property by clicking on the *Assign/Change Property* button in the above dialog box, ensure that you keep the check mark on "Apply to this Member only" in the ensuing dialog box. Else, changing the member attributes for one member will subsequently change the attributes of all other members belonging to the same attribute list. For example, if the current member's property is also assigned to other members, changing the property on the current member will change the property of all the members.

Let us click on the *Shear/Bending* tab. The following dialog box appears.

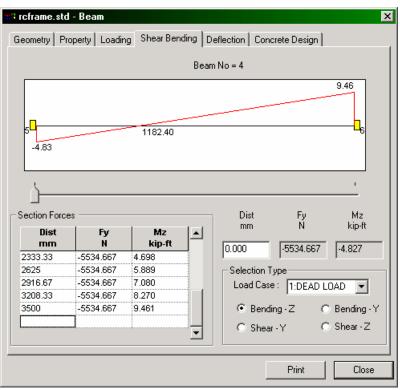


Figure 2. 161

The above page contains facilities for viewing values for shears and moments, selecting the load cases for which those results are presented, a slider bar (see next figure) for looking at the values at specific points along the member length, and a Print option for printing the items on display. Experiment with these options to see what sort of results you can get. Grab the slider bar using the mouse and move it to obtain the values at specific locations.

2-150 Tutorial 2

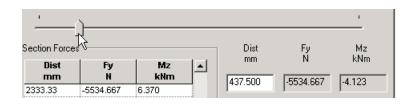


Figure 2. 162

Another page (Deflection) of the above dialog box is shown below.

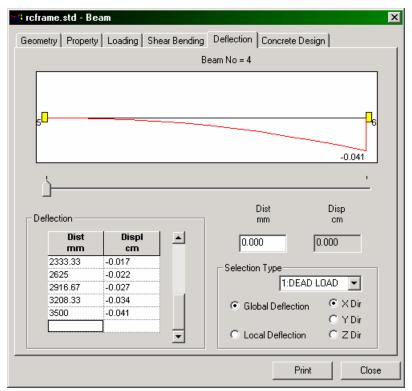


Figure 2. 163

The Concrete Design page of the above dialog box is shown below.

भन्न rcframe.std - Beam	×
Geometry Property Loading Shear Bending Deflection Co Beam no. = 4 Design code : ACI-0.	
300.000 Design Load 200.000 Load 5 Location STA Pu(Kns) 28.55 Mz(Kns-Mt) 6.82 My(Kns-Mt) 30.87 275.000 mm 275.000 mm State State State	Design ParameterFy(Mpa)415Fc(Mpa)25As Reqd(mm²)1056As (%)1.523Bar Size20Bar No4
	Print Close

Figure 2. 164

To look at the results of another member using this query facility, simply close down this query dialog box and repeat the steps outlined earlier in this section for the desired member.

2.11.18 Producing an on-screen report

Steps:

Occasionally, we will come across a need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form. The facility which enables us to obtain such customized on-screen results is the Report menu on top of the screen.

 Let us create one such report. We will create a table that shows the member major axis moment (MZ) values sorted in the order High to Low, for members 1 and 4 for all the load cases. The first step to do this is to select members 1 and 4 from the structure. With the beams cursor active, select members 1 and 4 using the mouse, or use the *Select* | *By List* | *Beams* option of the *Select* menu and type 1 and 4 as the member numbers. Next, go to the **Report** | **Beam** End Forces menu option as shown below.



Figure 2. 165

2. In the dialog box that appears, select the Sorting tab. Let us select Moment-Z as the End Force, set the Sorting Order to List from High to Low, and switch on Absolute values under If Sorting done. (If we wish to save this report for future use, we may select the Report tab, provide a title for the report, and turn on the Save ID option.) Under the Loading tab, make sure all the 5 load cases have been selected. Then, click on the OK button.

Beam End Force	×
Sorting Loading Report	
Sort By End Forces C Axial Force C Shear-Y C Shear-Z C Torsion C Moment-Y C Moment-Z	If Sorting Done ✓ Absolute Values Set Sorting Order ← List from High to Low ← List from Low to High
C List with no sort done	OK Cancel Help

Figure 2. 166

The next figure shows the member end forces sorted table with the MZ values sorted from High to Low based on Absolute numbers.

Beam	L/C	Node	Axial Force kN	Shear-Y kN	Shear-Z kN	Torsion kNm	Moment-Y kNm	Moment-2 kNm
4	4	6	-48.779	14.364	0.242	-0.416	2.547	-33.28
1	4	2	-48.779	14.364	-0.242	0.416	-2.547	-33.28
1	4	1	57.441	-14.364	0.242	-0.416	1.702	-16.98
4	4	5	57.441	-14.364	-0.242	0.416	-1.702	-16.98
4	5	6	-20.609	5.909	-0.982	2.210	4.731	-13.85
1	1	2	-18.800	5.535	-0.096	0.159	-0.984	-12.82
4	1	6	-18.800	5.535	0.096	-0.159	0.984	-12.82
1	2	2	-17.478	5.145	-0.090	0.148	-0.915	-11.92
4	2	6	-17.478	5.145	0.090	-0.148	0.915	-11.92
1	5	2	-18.823	7.916	0.372	-0.034	-0.760	-9.82
1	3	1	-1.420	8.876	-0.363	0.161	1.026	9.77
4	5	5	28.549	-5.909	-21.328	-2.210	30.874	-6.82
1	1	1	26.018	-5.535	0.096	-0.159	0.647	-6.54
4	1	5	26.018	-5.535	-0.096	0.159	-0.647	-6.54
1	2	1	17.478	-5.145	0.090	-0.148	0.600	-6.08
4	2	5	17.478	-5.145	-0.090	0.148	-0.600	-6.08
1	5	1	26.763	5.470	-0.372	0.034	2.063	5.54
1	3	2	1.420	1.421	0.363	-0.161	0.245	3.27
4	3	5	-0.054	0.136	-16.287	-1.822	24.188	0.28
4	3	6	0.054	-0.136	-0.874	1.822	2,785	0.19

Figure 2. 167

2-154 Tutorial 2

3. To print this table, click the right mouse button anywhere within the table. A list of options will appear.

X Cut	Ctrl+X
The Copy	Ctrl+C
💼 Baste	Ctrl+V
Delete	Del
<u>G</u> o To	F5
Change Unit for this column	•
📣 3D Rendering	
🛃 Pri <u>n</u> t	
D Print Pre <u>v</u> iew	
🝸 Results <u>S</u> etup	
Node <u>D</u> isplacements Table	
Beam End Displacements Table	
Beam Relative Dis <u>p</u> lacement De	tail Table
Support Reactions Table	
Beam End <u>F</u> orces Table	
Beam Forces Detail Table	
Plate Centre Stress Table	
Plate Corner Stress Table	
Solid Centre Stress Table	
Solid Corner Stress Table	
🔲 Other <u>T</u> ables	

Figure 2. 168

Select the print option to get a hardcopy of the report.

2.11.19 Taking Pictures

There are several options available for taking pictures. The simplest of these is in the edit menu and is called Copy Picture. It transfers the contents of the active drawing window to the windows clipboard. We can then go into any picture processing program like Microsoft Paint or Microsoft Word and paste the picture in that program for further processing.

Another more versatile option enables us to include any "snapshot" or picture of the drawing window into a report. It is called *Take Picture* and is under the *Edit* menu. Let us examine this feature.

Steps:

1. To take a picture, either click on the **Take Picture** icon or, go to *Tools* | *Take Picture* menu option.

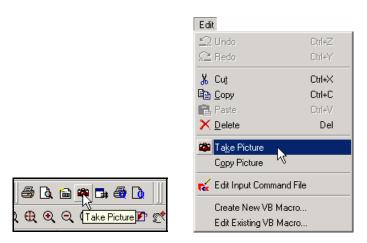


Figure 2. 169

2-156 Tutorial 2

2. The following dialog box comes up. Here, we may provide a caption for the picture so that it may be identified later on.

Picture 1	(K
ID:	Picture 1	
Caption:	Whole Structure	
	OK Help	
	Figure 2. 170	

This picture will be saved till we are ready to produce a customized report of results. Proceed to the next section for details.

2.11.20 Creating Customized Reports

STAAD.Pro offers extensive report generation facilities. Items which can be incorporated into such reports include input information, numerical results, steel design results, etc. One can choose from among a select set of load cases, mode shapes, structural elements, etc.. We may include any "snapshot" or picture of the screen taken using the *Take Picture* toolbar icon. Other customizable parameters include the font size, title block, headers, footers, etc.

The *Report Setup* utility may be accessed either by selecting the Reports Page or by clicking on the *Report Setup* icon.



Figure 2. 171

In either case, the following dialog box appears:

Picture Album Items Load Ca:	Options ses M	Name a lodes	and Logo Ranges	Load/Save Steel Design
Available		Selected	ł	
Input	•			
Nodes Beams	-	> Job Info	I	
Sections				
Materials		·>		
Supports				
Releases				+
Primary Load Cases		<		*
Combination Load Cases				
Loading Generators Loadings		··		
	Report Detail Ir	ncrements	10	
	riopon b ordini	101011101110		

Figure 2. 172

Different tabs of this dialog box offer different options. The *Items* tab lists all available data which may be included in the report. Note that the items under the *Selected* list are the ones which have been selected by default.

Available items are classified into seven categories: *Input*, *Output*, *Pictures*, *Reports*, *STAAD.etc output*, *Steel Design Output* and *Advanced Query Reports*.

Tutorial 2-159

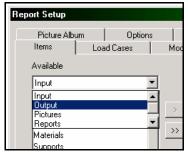


Figure 2. 173

2. In our report, we want to show Job Information, Node Displacement Summary, Beam Max Moments, and Picture 1.

Job Information is already selected by default.

From the Available list box, select Output. From the available output items, select Node Displacement Summary and Beam Max Moments.

Then select *Pictures* from the *Available* list box and select *Picture 1*.

When all the items have been selected, the *Report Setup* dialog box should appear as shown below.

```
2-160 Tutorial 2
```

Report Setup				×
Picture Album Items Lo. Available Pictures	Options ad Cases	Nam Modes Select	e and Logo Ranges ed	Load/Save Steel Design
		Beam Pictur	Displacement Sur Max Moments e 1	nmary
	Heport Deta	il Increments	10	
	СК	Car	icel Prin	nt Help

Figure 2. 174

The *Report Detail Increments* check box at the bottom indicates the number of segments into which a member would be divided for printing sectional displacements, forces, etc.

3. Click on the *Load Cases* tab to select the Load Cases to be included in the report. The *Grouping* buttons indicate whether the report data will be grouped by Node/Beam numbers or by Load Case number. In the first case, all Load Case results will appear under a particular Node or Beam. In the second case, results for all Nodes or Beams for a particular Load Case will appear together.

						Tutorial 2	2-161		
Report Setup						×			
Picture Album Dptions Items Load Cases			Nar	Load/Save	1				
Items	Load	Cases	Modes	Ranges	Steel Design				
Available				Selected					
			> >> < <	1 DEAD LOAD 2 LIVE LOAD 3 WIND LOAD 4 DEAD + LIVE 5 DEAD + WIN					
	Grouping	for Load Table	es — Grou	I Iping for Result Tab	oles—				
	O by Loa			y Node/Beam					
	• by Loa	ad Case	01	by Load Case					
OK Cancel Print Help									

Figure 2. 175

4. Click on the *Picture Album* tab to visually identify the pictures taken earlier. The following figure displays *Picture 1* as stored by the program.

Report Setup	×								
Items Load Cases Modes Ranges Picture Album Options Name and Logo	Steel Design Load/Save								
Name: Picture 1 Name: Picture 1 Pelete Picture Full Page Height: 174.7 mm Width: 83.63 mm Caption: Whole Structure Displacements 0.0003cm:1mm 1 DEAD LOAD									
OK Cancel Pri	int Help								

Figure 2. 176

The Options tab lets you include Header, Footer, Page Numbers, Table Grids, fonts for Column Heading and Table data, etc.

The *Name and Logo* tab allows you to enter the Company Name and Logo. Click on the blank area and type the name and address of the company. Click on the *Font* button in the *Text* group and adjust the font to be Arial 16 Pt Bold. Click on the *Right* radio button in the *Alignment* group under *Text* to right-align the company name.

Report Setup						x			
Items Picture Album	Load Case	es Options	Modes Name	Ranges and Logo	Steel Design Load/Save				
ABC CONSULTANTS									
Graphic									
File					Delete				
Paste	Position:	C Left	Centre	C Right					
Text						i I			
Font	Arial 16 pt								
	Alignment:	🖲 Left	C Centre	C Right					
		OK	Canc	el P	rint Help				

Figure 2. 177

5. Click *OK* to finish or click *Print* to print the report. However, it is always a good idea to first preview the report before printing it.

Tutorial 2 2-163

To preview the report just created, select the *Print Preview* icon from the Toolbar.



Figure 2. 178

The first and the last pages of the report are shown in the next two figures.

		ABC (CONSUL	TANTS				3 haad Ha		1-
			0011002						1	
3.8	-	i Harri Carpela								
-						-				
						۹r		Deb30-Hur-	de 10e	
1						the rest.		D	****	r-366+ I
неновото п Л Лановото п Татич	Norae Norae	S La Casea Rote Losa Case Note Servicion Mode Servicion	Hyberikere Hyberikere S er 6		<u>७</u> ४					
Prinury Prinury Prinury Prinury Prinury		1 2 1 •	DE60 L060 LNEL060 DE60 - LNE DE60 - LNE DE60 - WIND							
Prinury Prinury Prinury Prinury	Disp	s acemer	LINGLOOD WINDLOOD DECO - LING DECO - WIND INT SUMMA	<u> </u>						
Prinury Prinury Prinury Prinury		2 1 5	LMELOGO WINDLOGO DEGO - LME DEGO - WIND	arγ γ ιε ι	I I I I	E-Milleone 12M	rX E34	77 17.04	12	
Priteury Priteury Priteury Node	Disp	3 3 acemer Lc sono - v	INFLORD WINDLORD DERE - LAS DERE - WIND DERE - WIND DE	7 16 M -6566	14M		814) -6261	834) 6661	1734 -6262	
Priteury Priteury Priteury Node Hu> X	Disp hate	3 3 3 3 3 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5	LAFLORD WIND LORD DERO - LAF DERO - WIND TE Summa X ICH MINI 66075 AF -6160	7 16 M -6266 -6216	1694 -6366 6618	16M 6.675 6.164	F34) -6261 -6262	824) 6261 -6266	1734 -6.662 -6.663	
Proteury Proteury Proteury Proteury Node Hu> X He X Hu> V	Disp hate 6	3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3	LINELOSO WINDLOSO DESE - LINE DESE - LINE DESE - LINE DESE - LINE T SE - CATSO NE - CATSO NE - CATSO SE - CATSO	7 16 M -6565 -6516 -6666	1004 -6266 6418 -6452	16 M 6475 6.164 6.116	#24 -6261 -6262 -6260	(623 (623 (623 (623)	1734 -6363 -6366 -6366	
Priteury Priteury Priteury Node Hu> X	Disp hate	3 3 3 3 3 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5	LINE LOED WIND LOED DEED - LINE DEED - LINE DEED - LINE DEED - LINE DEED - LINE NIEL - COTS NIEL - COT	7 16 M -6266 -6216	1694 -6366 6618	16M 6.675 6.164	F34) -6261 -6262	824) 6261 -6266	1734 -6.662 -6.663	
Prinury Prinury Prinury Prinury Prinury Prinury Hu> X Hu> X Hu> X Hu> X Hu> T Hu> T	Disp hate	3 3 3 5 3 3 3 5 5 5 5 5 5 5 5 5 5 5 5 5	UNFLOED WIND LOED DEED - UNF DEED - UNF DEED - UNF DEED - UNF MINI GETS NT - 46622 NT - 46612 NT - 46612	7 12 M 48865 48818 6886 48818 48818 48866	1694 -6566 6618 -6613 -6653 -6555 -6566	1694 6675 6.164 6.116 6.164 6.164	#34 -6565 -6566 -6566 -6565 -6566	824) 6261 6266 6266 6266 6266	1011 (000 (000 (000 (000 (000 (000 (000	
Prinury Prinury Prinury Prinury Nocie Hu> X Hu> X Hu> X Hu> X Hu> X Hu> X Hu> X	Disp hate 6 2 3 3	2 3 5 5 300000 - W - 5000 - U 2000 - U 2000 - U 2000 - U 2000 - U 2000 - U	UNFLOED WINDLOED DEED-UNF DEED-UNF DEED-WIND XIII CONT XIII XIII XIII XIII XIII XIII XIII XI	7 16 m 48865 48816 68816 68816 48816 48866 48817	1494 -6466 6613 6633 6633 6466 -6466 6633	1204 6.6575 6.166 6.616 6.616 6.164 6.111 6.616	#24 -6260 -6260 -6260 -6260 -6260 -6260 -6260	F24 6561 -6566 -6566 -6566 6566 -6566	1124 5355 5355 5355 5355 5355 5355 5355	
Prinury Prinury Prinury Prinury Prinury Prinury Hu> X Hu> X Hu> X Hu> X Hu> T Hu> T	Disp hate	3 3 3 5 3 3 3 5 5 5 5 5 5 5 5 5 5 5 5 5	LINFLORD WIND LORD DEED - LINF DEED - WIND THE SUITTING WINI 6075 NE - 6012 NE - 6022 NE - 6022 NE - 6022 NE - 6022 NE - 6022 NE - 6022 NE - 6022	7 12 M 48865 48818 6886 48818 48818 48866	1694 -6566 6618 -6613 -6653 -6555 -6566	1694 6675 6.164 6.116 6.164 6.164	#34 -6565 -6566 -6566 -6565 -6566	824) 6261 6266 6266 6266 6266	1011 (000 (000 (000 (000 (000 (000 (000	
Prinury Prinury Prinury Prinury Node Hu> X Hu> X Hu> X Hu> X Hu> T Hu> T Hu> T Hu> T	Disp hate 6 3 3 3 4	2 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3	LINFLORD WIND LORD DEED - LINF DEED - WIND THE SUIMMEN NET SOLOTION NET SOLOTION NET SOLOTION NET SOLOTION NET SOLOTION NET SOLOTION	7 16 M 16865 16865 16865 16865 16865 16865 16865	1294 -62666 -6253 -6253 -62666 -62666 -62666 -62666 -62666 -62666 -62666 -62666	12M 6,675 6,164 6,164 6,164 6,164 6,111 6,636 6,164	#34 45551 45552 65552 65552 65551 45555 65552 65552 65552 65552	524 6266 6266 6266 6266 6266 6266 6266	4171 (3323- (3323- (3323- (3323- (3323- (3323- (3323- (3323- (3323- (3323-	
Prinary Prinary Prinary Prinary Prinary Prinary Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X	Disp hate 6 2 3 3 3 5 6 6 6 6	2 3 3 3 3 5 3 5 5 5 5 5 5 5 5 5 5 5 5 5	LINFLORD WIND LORD DEED - LINF DEED - WIND THE SUITTING WINT 6675 NF - 66322 NF - 66322 NF - 66326 NF - 66325 NF - 66326 NF - 66325 NF - 66325	7 12M 48865 4818 48865 48865 48865 48865 48865 48865 48865 48865 48865	1204 -6.666 6.612 6.632 6.632 6.666 6.667 6.667 6.651 6.651 6.651 6.651 6.651	12M 6.575 6.164 6.116 6.116 6.111 6.111 6.616 6.164 6.575 6.164 6.616	234 4555 4555 4555 4555 4555 4555 4555 4	141 6266 62666 62666 62666 62666 62666 62666 62666 62666	4111 (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333-))))))))))	
Prinury Prinury Prinury Prinury Prinury Prinury Prinury Hu> X Hu> X Hu Hu X Hu X Hu X Hu X Hu X Hu X Hu X	Disp hate 6 3 3 3 4 5 5 3 3 5 5 5 3 3 5 5 5 5 5 5 5	2 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3	UNET LOGO WIND LOGO DEGO - UNET DEGO - UNET DEGO - WIND TE SUMMING NEL 6675 NE 66622 NE 66622 NE 66622 NE 66625 NE 66625 NE 66626 NE 66625 NE 66616 NE 66616	7 16M 4886 4886 4886 4886 4886 4886 4886 48	12M -6.666 6.612 6.632 6.632 6.666 6.6518 -6.666 6.6518 6.6518 6.6522 6.166	1204 6.675 6.164 6.116 6.016 6.116 6.164 6.164 6.164 6.164	234 4555 4555 4555 4555 4555 4555 4555 4	1231 5255 5255 5255 5255 5255 5255 5255	1734 -6565 -6565 -6565 -6565 -6565 -6565 -6565 -6565 -6565 -6565 -6565 -6565	
Prinary Prinary Prinary Prinary Prinary Prinary Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X Hu>X	Disp hate 6 2 3 3 3 5 6 6 6 6	2 3 3 3 3 5 3 5 5 5 5 5 5 5 5 5 5 5 5 5	UNET LOGO WIND LOGO DEGO - UNET DEGO - UNET DEGO - WIND TE SUMMING NEL 6675 NE 66622 NE 66622 NE 66622 NE 66625 NE 66625 NE 66626 NE 66625 NE 66616 NE 66616	7 12M 48865 4818 48865 48865 48865 48865 48865 48865 48865 48865 48865	1204 -6.666 6.612 6.632 6.632 6.666 6.667 6.667 6.651 6.651 6.651 6.651 6.651	12M 6.575 6.164 6.116 6.116 6.111 6.111 6.616 6.164 6.575 6.164 6.616	234 4555 4555 4555 4555 4555 4555 4555 4	141 6266 62666 62666 62666 62666 62666 62666 62666 62666	4111 (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333- (333-))))))))))	

Figure 2. 179

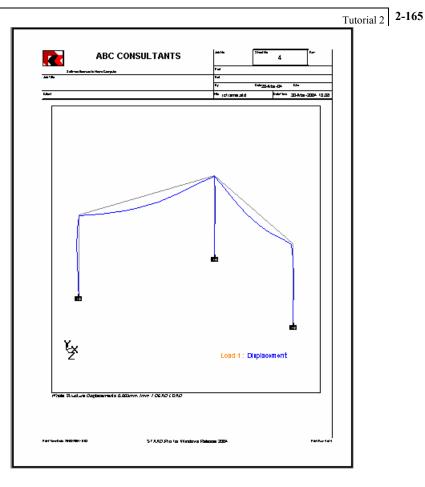


Figure 2. 180

This brings us to the end of this tutorial. Though we have covered a large number of topics, there are several more in various menus and dialog boxes. Happy exploring !!!!! **2-166** Tutorial 2

N o t e s

Tutorial Problem 3: Analysis of a slab

This tutorial provides step-by-step instructions for modelling and analysis of a slab supported along two edges. The following topics are covered:

- Starting the program
- Modelling the slab using quadrilateral plate elements
- Specifying slab properties and material constants
- Specifying supports
- Specifying loads
- Specifying the analysis type
- Viewing results for individual plates

Section 3

3.1 Methods of creating the model

As explained in Section 1.1 of tutorial problem 1, there are two methods of creating the structure data:

- a) using the graphical model generation mode, or graphical user interface (GUI) as it is usually referred to.
- b) using the command file.

Both methods of creating the model are explained in this tutorial. The graphical method is explained from Section 3.2 onwards. The command file method is explained in Section 3.8.

3.2 Description of the tutorial problem

The structure for this project is a slab fixed along two edges. We will model it using 6 quadrilateral (4-noded) plate elements. The structure and the mathematical model are shown in the figures below. It is subjected to selfweight, pressure loads and temperature loads. Our goal is to create the model, assign all required input, perform the analysis, and go through the results.

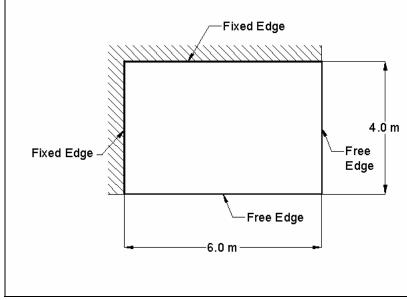


Figure 3.1

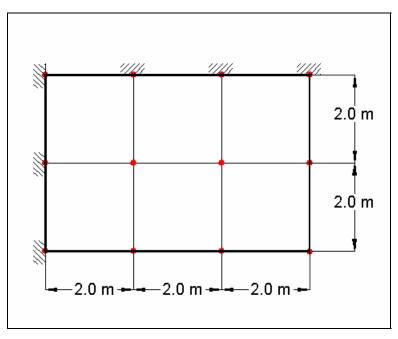


Figure 3. 2

BASIC DATA FOR THE STRUCTURE

ATTRIBUTE	DATA
Element properties	Slab is 300mm thick
Material Constants	E, Density, Poisson, Alpha – Default values for concrete
Supports	Nodes along 2 edges are fixed as shown in Figure 3.2
Primary Loads	Load 1: Selfweight
	Load 2: Pressure Load of 300Kg/sq.m. acting vertically downwards
	Load 3: 75 degree F uniform expansion, plus top surface is 60 degrees hotter than the bottom
Combination Loads	Case 101: Case 1 + Case 2
	Case 102: Case 1 + Case 3
Analysis Type	Linear Elastic

3.3 Starting the program

Select the **STAAD.Pro** icon from the *STAAD.Pro* 2005 program group.

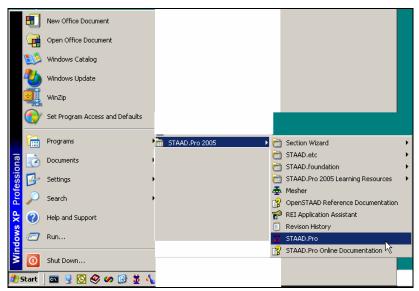


Figure 3.3

The STAAD.Pro Graphical Environment will be invoked and the following screen comes up.

👮 STAAD.Pro				- 8 ×
File Help 僧울날 등 등 등 종 處 × 으±으±값 目 등 함		m a attac attac or Ne her	. M. M. T. A. A. A. M. A. M. T. P. M.	8 .e.
				E W
₽₽₽₽₽₽₽₽₽ ++++++++++++++++++++++++++++			× ?	
イジドド茶を開催日本国工の語	ポペキの国ン 豆をも	@&\$\$ \$\$\$\$\$		
	New File Recent Files		×	
	☑ Space	File Name ;		
λ.	Plane Floor	Structure1		
lyn Iŵ	Truss	Location :		
		C \test\		
+		1		
	A SPACE structure, which is a three-dimensi	onal framed structure with loads applied in		
	any plane, is the most general			
	Length Units	Force Units		
#	Činch CDecimeter €Foot CMeter	C Pound C Newton C Newton C DecaNewton		
•	C Milimeter C Kilometer	C Kilogram C KiloNewton		
	C Centimeter	C Metric Ton C MegaNewton		
	Display this dialog box at the Startup			
·	< Back	Next> Cancel Help		

Figure 3.4

This New dialog box will come up every time we start the program. If you wish to turn this feature off, simply uncheck the Display this dialog box at the Startup box at the lower left hand corner. It can be turned on again at a later time when $File \mid New$ is invoked from the main menu.

Note about the unit system :

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, Aluminum – selected from the program's library (Please refer to Section 5 of the STAAD.Pro 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.

The place from where we can change this setting is under the $File \mid Configure$ menu. To get to that option, first close down the dialog box shown in the earlier figure by clicking on **Cancel**. Then, click on the **File** | **Configure** menu option and choose the appropriate unit system you want. For this tutorial, let us choose the **Metric** units (KN, Meter, etc.).

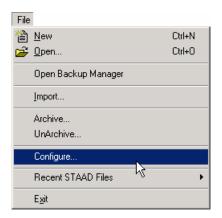


Figure 3.5

			Tutorial 3
onfigure Program		×	
File Options Input File Format	STAAD Default De	sign Codes e Format	
Base Unit Background C		-	
Current Default Base M	/letric		

Figure 3. 6

Click on the Accept button to close the above dialog box.

Following this, select File | New once again.

File	
웥 <u>N</u> ew	Ctrl+N
궏 <u>O</u> pen 서	Ctrl+O
Open Backup Manager	
Import	
Archive	
UnArchive	
Configure	
Recent STAAD Files	•
E <u>x</u> it	

Figure 3.7

The dialog box shown in Figure 3.4 will re-appear.

3.4 Creating a new structure

1. In the *New* dialog box, we provide some crucial initial data necessary for building the model.

The structure type is to be defined by choosing from among *Space*, *Plane*, *Floor* and *Truss*. A *Space* type is one where the structure, the loading or both, cause the structure to deform in all 3 global axes (X, Y and Z). In a *Plane* type, the geometry, loading and deformation are restricted to the global X-Y plane only. A *Floor* type is a structure whose geometry is confined to the X-Z plane. A *Truss* type of structure carries loading by pure axial action. Truss members are deemed incapable of carrying shear, bending and torsion. For our model, let us choose **Space**.

Let us choose **Meter** as the length unit and **Kilo Newton** as the force unit in which we will start to build the model. The units can be changed later if necessary, at any stage of the model creation.

We also need to provide a name in the *File Name* edit box. This is the name under which the structure data will be saved on the computer hard disk. The name "Structure?" (? will be a number) is recommended by the program by default, but we can change it to any name we want. Let us choose the name **Plates Tutorial**.

A default path name - the location on the computer drive where the file will be saved - is provided by the program under *Location*. If you wish to save the file in a different location, type in the name, or click the button and specify the desired path.

After specifying the above input, click on the Next button.

Space		File Name :	
Floor		Plates Tutorial	
Truss		Location :	
		C:\test\	
		,	
Length Units	C Decimeter	Force Units	C Newton
C Foot	Meter	KiloPound	C DecaNewton
O Millimeter	C Kilometer	🔿 Kilogram	KiloNewton
C Centimeter		O Metric Ton	C MegaNewton

Figure 3.8

2. In the next dialog box, we choose the tools to be used to initially construct the model. Add Beams, Add Plates or Add Solids are, respectively, the starting points for constructing beams, plates or solids. Open Structure Wizard provides access to a library of structural templates which the program comes equipped with. Those template models can be extracted and modified parametrically to arrive at our model geometry or some of its parts. If the model is to be created initially using the STAAD command language, the Open Editor box can take us to the STAAD editor. Please remember that all these options are also available from the menus and dialog boxes of the GUI, even after we dismiss this dialog box.

Note: If you wish to use the Editor to create the model, choose Open STAAD Editor, click Finish, and proceed to Section 3.8.

For our model, let us check the **Add Plate** option. Click on the **Finish** button. The dialog box will be dismissed and the STAAD.Pro graphical environment will be displayed.

Where do you want to go ?	x
Add Beam Add Plate Add Solid Open Structure Wizard Open STAAD Editor Edit Job Information	
Begin building your model by creating new joi using the construction grid, drawing tools and	ints and 3-noded and 4-noded plate elements d spreadsheets.
Display this dialog box at the Startup < Back	Finish Cancel Help

Figure 3.9

3.5 Elements of the STAAD.Pro screen

The STAAD.Pro main window is the primary screen from where the model generation process takes place. It is important to familiarize ourselves with the components of that window before we embark on creating the RC Frame. Section 1.5 in tutorial problem 1 of this manual explains the components of that window in detail.

3.6 Building the STAAD.Pro model

We are now ready to start building the model geometry. The steps and, wherever possible, the corresponding STAAD.Pro commands (the instructions which get written in the STAAD input file) are described in the following sections.

3.6.1 Generating the model geometry

The structure geometry consists of joint numbers, their coordinates, member numbers, the member connectivity information, plate element numbers, etc. From the standpoint of the STAAD command file, the commands to be generated are :

```
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

ELEMENT INCIDENCES SHELL

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
```

In this tutorial, we will explore 4 different methods to create the model shown in section 3.2:

- 1. Using a mixture of drawing an element and the Copy/Paste facility.
- 2. Using a mixture of drawing an element and the Translational Repeat facility.
- 3. Using the Structure Wizard facility in the Geometry menu.
- 4. Using the Mesh Generation facility of the main graphical screen.

Creating the Plates - Method 1

Steps:

The Grid Settings

1. We selected the *Add Plate* option earlier to enable us to add plates to create the structure. This initiates a grid in the main drawing area as shown below. The directions of the global axes (X, Y, Z) are represented in the icon in the lower left hand corner of the drawing area. (Note that we could initiate this grid by selecting the *Geometry* | *Snap/Grid Node* | *Plate* menu option also.)

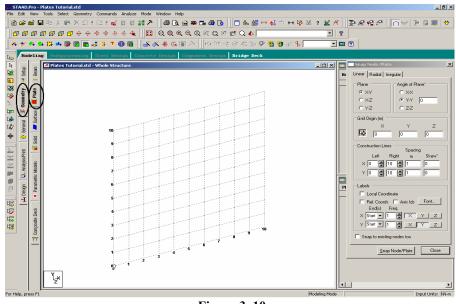


Figure 3.10

3-18 Tutorial 3

It is worth paying attention to the fact that when we chose the Add*Plate* option in section 3.4, the page control area has been automatically set up to have the *Geometry* – *Plate* page be in focus.

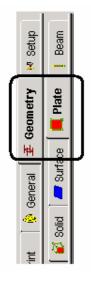


Figure 3.11

2. A Snap Node/Plate dialog box also appears in the data area on the right side of the screen. The Linear tab is meant for placing the construction lines perpendicular to one another along a "left to right - top to bottom" pattern, as in the lines of a chess board. The Radial tab enables construction lines to appear in a spider-web style, which makes it is easy to create circular type models where members are modelled as piece-wise linear straight line segments. The Irregular tab can be used to create gridlines with unequal spacing that lie on the global planes or on an inclined plane. We will use the Linear tab.

In our structure, the elements lie in the X-Z plane. So, in this dialog box, let us choose X-Z as the *Plane* of the grid. The size of the model that can be drawn at any time is controlled by the number of *Construction Lines* to the left and right of the origin of axes, and the *Spacing* between adjacent construction lines. By setting **6** as the number of lines to the right of the origin along X, **4** along Z, and a spacing of **1 meter** between lines along both X and

Z (see next figure) we can draw a frame 6m X 4m, adequate for our model. Please note that these settings are only a starting grid setting, to enable us to start drawing the structure, and they do not restrict our overall model to those limits. In fact, we do not even need this 6m X 4m grid. The method we are using here requires just a 2m X 2m grid since we are about to draw just a single element.

88 Snap Node/Plate	×						
Linear Radial Irreg	gular						
- Plane	Angle of Plane*						
C X-Y	C XX						
● x-z	• Y-Y 0						
O Y-Z	O Z-Z						
Grid Origin (m)							
× (0	Y Z						
Construction Lines							
Left Ri	Spacing Left Right m Skew*						
Local Coordina	ate Axis Ids Font						
End(s) Fr							
X Start 💌 1	× Y Z						
Z Start 💌 1	X Y Z						
Snap to existing	nodes too						
<u>S</u> nap N	lode/Plate Close						

Figure 3.12

Creating element 1

The four corners of the first element are at the coordinates (0, 0, 0), (2, 0, 0), (2, 0, 2), and (0, 0, 2) respectively. To start creating the nodes, let us first activate the Snap Node/Plate button by clicking on it. Then, with the help of the mouse, click at the origin (0, 0, 0) to create the first node.

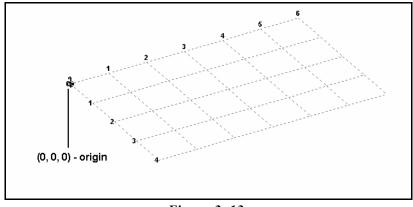


Figure 3. 13

4. In a similar fashion, click on the remaining three points to create nodes and automatically join successive nodes by a plate.

(2, 0, 0), (2, 0, 2) and (0, 0, 2)

The exact location of the mouse arrow can be monitored on the status bar located at the bottom of the window where the X, Y, and Z coordinates of the current cursor position are continuously updated.

	1.1								2
Add nodes/plates to line intersections using cursor. Hold CTRL key down to reset.	ķ.	2,000	γ.	0.000	7.	2.000	Input Units:	kN·m	

Tutorial 3 3-21

When steps 1 to 4 are completed, the element will be displayed in the drawing area as shown below.

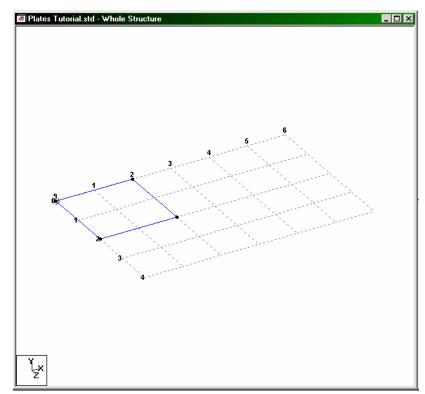


Figure 3. 14

3-22 Tutorial 3

5. At this point, let us remove the grid display from the structure. To do that, click on the **Close** button in the *Snap Node/Plate* dialog box.

📽 Snap Node/Plate 🛛 🔀
Linear Radial Irregular
Plane Angle of Plane*
O XY O XX
• X-Z • Y-Y 0
C Y-Z C Z-Z
Grid Origin (m)
X Y Z
Construction Lines
Spacing Left Right m Skew*
Labels
🗖 Local Coordinate
🗖 Rel. Coords 🔲 Axis Ids 🔤 Font
End(s) Freq.
X Start V 1 X Y Z
Z Start V 1 X Y Z
Snap to existing nodes too
Snap Node/Plate
\smile

Figure 3.15

The grid will now be removed and the structure in the main window should resemble the figure shown below.

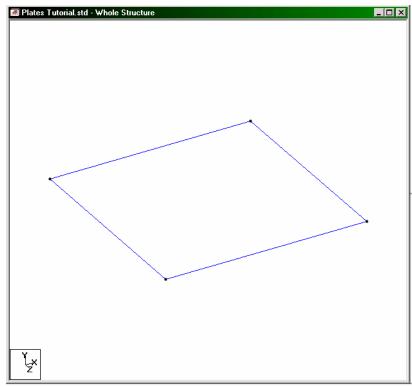


Figure 3.16

- 6. It is very important that we save our work often, to avoid loss of data and protect our investment of time and effort against power interruptions, system problems, or other unforeseen events. To save the file, pull down the *File* menu and select the *Save* command.
- 7. For easy identification, the entities drawn on the screen can be labeled. Let us display the *plate numbers*. (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about switching on node/beam/plate labels.)

3-24 Tutorial 3

The following figure illustrates the plate number displayed on the structure. The structure in the main window should resemble the figure shown below.

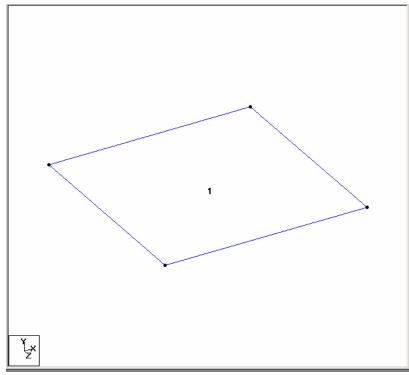


Figure 3. 17

If you are feeling adventurous, here is a small exercise for you. Change the font of the plate labels by going to the *View* menu and selecting the *Options* command, and then selecting the appropriate tab (Plate labels) from the *Options* dialog box.

Creating element 2

8. Examining the structure shown in section 3.2 of this tutorial, it can be seen that the remaining elements can be easily generated if we could copy the existing plate and then, paste the copied unit at specific distances. The program does indeed have a *Copy-Paste* facility and it is under the *Edit* menu.

First, select plate 1 using the *Plates Cursor*

9. Click the right mouse button and choose *Copy* from the pop-up menu (or click on the *Edit* menu and choose *Copy*). Once again, click the right mouse button and select *Paste Plates* (or choose *Paste Plates* from the *Edit* menu) as shown below.

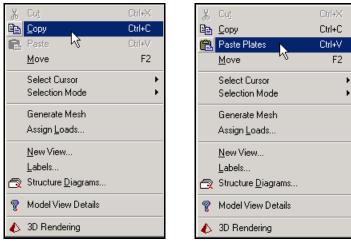


Figure 3. 18

Figure 3. 19

3-26 Tutorial 3

10. Since this facility allows us to create only one copy at a time, all that we can create from element 1 is element 2. The four nodes of element 2 are at distance of X = 2, Y = 0, and Z = 0 away from element 1. So, in the dialog box that comes up, provide 2, 0, and 0 for X, Y and Z respectively and click on the **OK** button.

Paste with Move		×
Move current clipboa	rd selection by:	ОК
X: 2	m	Cancel
Y: 0 r	m	Help
,		Reference Pt
Z: 0	m	

Figure 3. 20

The model will now look like the one shown below.

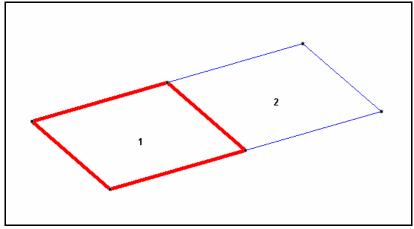


Figure 3. 21

Creating element 3

11. The nodes of element 3 are at X = 4m away from those of element 1. So, let us create the third element by repeating steps 8 to 10 except for providing **4m** for X in the *Paste with Move* dialog box.

Alternatively, we could use element 2 as the basis for creating element 3, in which case, the X increment will be 2m. If you make a mistake and end up pasting the element at a wrong location, you can undo the operation by selecting *Undo* from the *Edit* menu.

After creating the third element, the model should look like the one shown below.

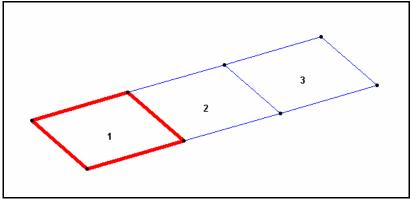


Figure 3. 22

Click anywhere in the screen to un-highlight the highlighted plate.

Creating elements 4, 5 and 6

12. The elements 4, 5 and 6 are identical to the first three elements except that their nodes are at a Z distance of 2m away from the corresponding nodes of elements 1 to 3. We can hence use the *Copy-Paste* technique and specify the Z increment as **2m**.

Select all three of the existing plates by rubber-banding around them using the mouse.

3-28 Tutorial 3

- 13. Click the right mouse button and choose **Copy** from the pop-up menu (or click on the *Edit* menu and choose *Copy*). Once again, click the right mouse button and select **Paste Plates** (or choose *Paste Plates* from the *Edit* menu).
- 14. Provide **0**, **0**, and **2** for X, Y and Z respectively in the *Paste with Move* dialog box that comes up. Then, click on the **OK** button and observe that three new elements are created.

Since some elements are still highlighted, click anywhere in the drawing area to un-highlight those elements.

The model, with all the six plates generated, will now look as shown below.

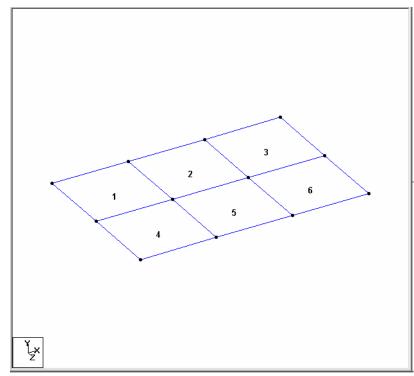


Figure 3. 23

If you want to proceed with assigning the remainder of the data, go to section 3.6.2.

If instead, you wish to explore the remaining methods of creating this model, the current structure will have to be entirely deleted. This can be done using the following procedure.

From the **Select** menu, choose **By All** | **All Geometry** option. The entire structure will be highlighted. Hit the determine the key on your keyboard. A message box shown below will appear. Click on the **OK** button.



Figure 3. 24

As we click on the OK button, the following message box appears. Let us say **Yes**. The message box will be dismissed and the entire structure will be deleted.

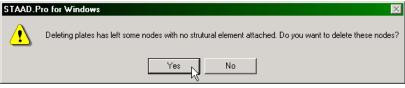


Figure 3. 25

Creating the Plates - Method 2

Steps:

Creating element 1

 In this method, we will be using STAAD's *Translational Repeat* facility to create our model. To utilize this facility, we need at least one existing entity to use as the basis for the translational repeat. So, let us follow steps 1 to 7 from 'Method 1' to create the first element. Once that is done, our model will look like the one shown below.

Note: If you have trouble bringing the grid settings dialog box, go to the *Geometry* menu and select *Snap/Grid Node Plate*.

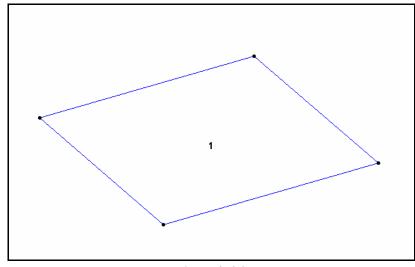


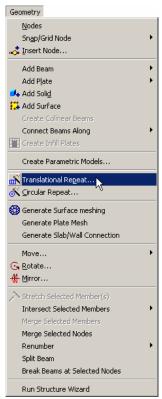
Figure 3.26

Creating elements 2 and 3

2. In Method 1, it required two separate executions of the Copy/Paste function to create elements 2 and 3. That is because, that facility does not contain a provision for specifying the number of copies one would like to create. Translational Repeat is a facility where such a provision is available.

Select plate 1 using the *Plates Cursor* **a**. (Please refer to the 'Frequently Performed Tasks' section at the end of this manual to learn more about selecting plates.)

3. Click on the **Translational Repeat** icon or select the *Geometry* | *Translational Repeat* menu option as shown below.



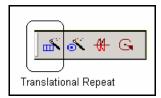


Figure 3. 27

Figure 3. 28

The *3D Repeat* dialog box comes up. By default (when the *Geometry Only* option is not checked), all loads, properties, design parameters, member releases, etc. on the selected entities will automatically be copied along with the entities. By checking the new option labeled *Geometry Only*, the translational repeating will be performed using only the Geometry data. In our example, it does not matter because no other attributes have been assigned yet.

4. To create elements 2 and 3 along the X direction, specify the Global Direction as X, No of Steps as 2 and the Default Step Spacing (along X) as 2m. The Link Steps option is applicable when the newly created units are physically removed from the existing units, and when one wishes to connect them using members. Renumber Bay enables us to use our own numbering scheme for entities that will be created, instead of using a sequential numbering that the program does if no instructions are provided. Let us leave these boxes <u>unchecked</u>. Then, click on OK.

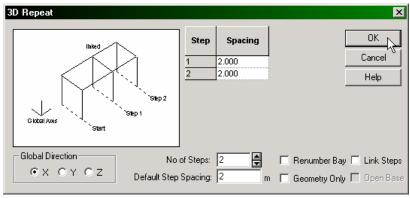


Figure 3. 29

Since element 1 is still highlighted, click anywhere in the drawing area to un-highlight it.

The model will now look as shown below.

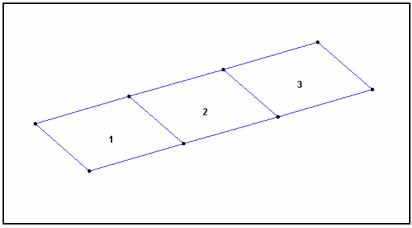


Figure 3.30

Creating elements 4, 5 and 6

- 5. Let us follow the same *Translational Repeat* method to create these elements. Select all the three existing plates by rubber-banding around them using the mouse. Make sure that before you do this, the cursor type is the *Plates Cursor* , else, no plates will be selected.
- Repeat steps 3 and 4 but this time, specify the *Global Direction* as Z, *No of Steps* as 1 and the *Default Step Spacing* as 2m. Leave all the other boxes <u>unchecked</u>. Then, click on OK.

All the 6 elements are now created.

Since some of the plates are still highlighted, click anywhere in the drawing area to un-highlight them.

Our model will now look like the one shown below.

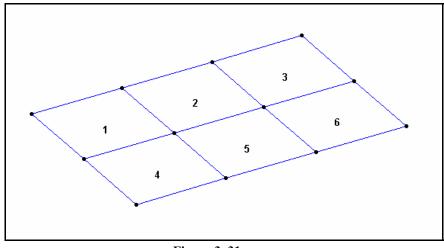


Figure 3. 31

If you want to proceed with assigning the remainder of the data, go to section 3.6.2.

Instead, if you wish to explore the remaining methods of creating this model, the current structure will have to be entirely deleted. This can be done using the following procedure.

From the **Select** menu, choose **By All** | **All Geometry** option. The entire structure will be highlighted. Hit the determine the key on your keyboard. A message box shown below will appear. Click on the **OK** button.



Figure 3. 32

Tutorial 3 3-35

As we click on the *OK* button, the following message box appears. Let us say **Yes**. The message box will be dismissed and the entire structure will be deleted.

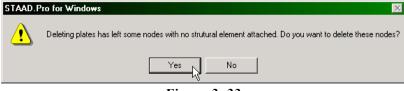


Figure 3.33

Creating the Plates - Method 3

Steps:

There is a facility in STAAD called *Structure Wizard* which offers a library of pre-defined structure prototypes, such as Pratt Truss, Northlight Truss, Cylindrical Frame, etc. A surface entity such as a slab or wall, which can be defined using 3-noded or 4-noded plate elements is one such prototype. We can also create our own library of structure prototypes. From this wizard, a structural model may parametrically be generated, and can then be incorporated into our main structure. *Structure Wizard* can hence be thought of as a store from where one can fetch various components and assemble a complete structure.

1. Select the *Geometry* | *Run Structure Wizard* menu option from the top menu bar.

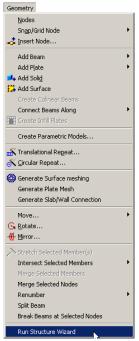


Figure 3. 34

Tutorial 3 3-37

The Structure Wizard window opens up as shown below.

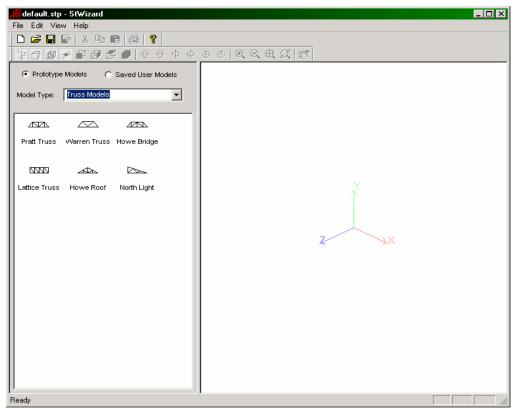


Figure 3.35

(Note that the *Open Structure Wizard* option in the *Where do you want to go?* dialog box in the beginning stage of creating a new structure – see Figure 3.9 – also brings up this facility.)

3-38 Tutorial 3

2. The unit of length should be specified prior to the generation of a model. From the File menu of this Structure Wizard window, click Select Units. In the Select Units dialog box that comes up, we can select one of the units of length from Imperial (inch, feet) or SI/Metric (millimeter, centimeter, meter) system of units. Let us choose Meters and click on the OK button.

default.stp - StWizard				
File Edit View Help				
New	Ctrl+N			
Open	Ctrl+O			
Save	Ctrl+S			
Save As				
Select Units		Select Units		×
Merge Model with STAAD.	Pro Model	- Imperial	- SI/Metric	
Add Plugin		C Inches	O Millimeters	
 Set As Top Most 				
1 C:\SPro2005\\default	.stp	C Feet	C Centimeters	ОК
2 C:\SPro2004\\default	.stp		Meters	Cancel
Exit				
E' 2) (E'	

Figure 3. 36

Figure 3. 37

3. From the *Model Type* list box, select **Surface/Plate Models** as shown below.

👍 default.stp	- StWizard		
File Edit Vie	w Help		
🗋 🖆 🔚 🛛	r X 🖻 🖻 🔗 🥐		
¥00	¥ 🗗 🗗 🖉 🗗 🕂 🔶 🔶 🔶		
Prototype	Models C Saved User Models		
Model Type:	Truss Models 🗸		
	Truss Models		
	Frame Models		
	Surface/Plate Models Solid Models		
Pratt Truss Composite Models			
Import CAD Models			
	VBA-Macro Models		
0000			
Lattice Truss	Howe Roof North Light		

Figure 3.38

Tutorial 3-39

4. To select the *Quad Plate* option, click on it using the mouse.

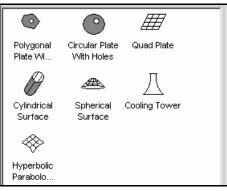


Figure 3. 39

5. Then, using the mouse, either double-click on the **Quad Plate** option or, drag the *Quad Plate* option to the right side of the *Structure Wizard* window as shown below.

File Edit View Help	
D 📽 🖬 🕼 🗴 🖿 🏙 🍊 💡	
	୬ େ ଷ୍ ଷ୍ ଷ୍ ଅ
Prototype Models C Saved User Models	\frown
Model Type: Surface/Plate Models	
Polygonal Circular Plate Quad Plate Plate Wi With Holes	E Contraction of the second seco
<i>I I I I I I I I I I</i>	
Cylindrical Spherical Cooling Tower Surface Surface	Y ↑
*	
Hyperbolic Parabolo	Z XX
Ready	Input Unit: m ///

Figure 3.40

3-40 Tutorial 3

6. A dialog box by the name *Select Meshing Parameters* comes up. In this box, we specify, among other things, two main pieces of information - a) the dimensions of the boundary (or superelement as it is commonly known) from which the individual elements are generated b) the number of individual elements that must be generated. (a) is defined in terms of the X, Y, Z coordinates of its Corners A, B, C and D. (b) is defined in terms of the number of divisions along sides AB, BC, etc.

Let us provide the **Corners**, the **Bias** and the **Divisions** of the model as shown in the figure below. Then, click on the **Apply** button.

Select Meshing Parameters		×
Model Name: Quad Plate Corners X Y Z A: 0 0 0 B: 6 0 0 C: 6 0 4 D: 0 0 4	Length, Bias & Division Length(m) Bias Divn. AB: 6 1 3 BC: 4 1 2 CD: 6 1 3 DA: 4 1 2	Element Type C Triangle I Quadrilateral
All units are in m	Apply	Cancel

Figure 3. 41

If you made a mistake and wish to bring up the above dialog box again, click the right mouse button in the drawing area and choose *Change Property*.

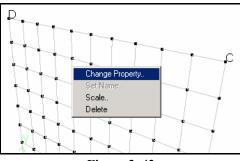


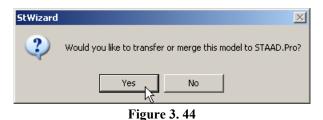
Figure 3.42

 To transfer the model to the main window, select File | Merge Model with STAAD.Pro Model as shown below.

4 default.stp - St\vizard				
File Edit View Help				
New	Ctrl+N			
Open	Ctrl+O			
Save	Ctrl+S			
Save As				
Select Units				
Merge Model with STAAD.Pro Model				
Add Plugin				
Set As Top Most				
1 C:\SPro2004\\default.stp				
2 C:\Spro2003\\default.stp				
Exit				

Figure 3. 43

8. When the following message box comes up, let us confirm our transfer by clicking on the **Yes** button.



The dialog box shown in the next figure comes up. If we had an existing structure in the main window, in this dialog box, we will be able to provide the co-ordinates of a node of the structure in the main window to which we want to connect the piece being brought from the wizard.

If there isn't any existing structure, this box is a means of specifying any distances along X, Y and Z axes by which we want the unit (being brought from the Wizard) to be shifted.

3-42 Tutorial 3

In our case, since we do not have an existing structure in the main window, nor do we wish to shift the unit by any amount, let us simply click on the **OK** button.

Paste Prototype Model	×
Move prototype model by:	ОК
X: 0 m	Cancel
Y: 0 m	Help
Z: 0 m	

Figure 3. 45

The model will now be transferred to the main window.

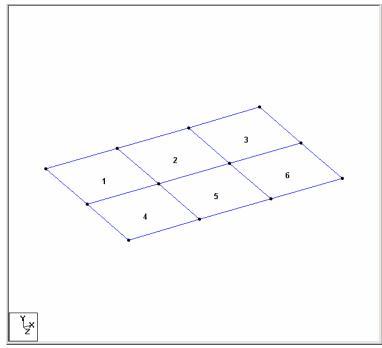


Figure 3. 46

If you want to proceed with assigning the remainder of the data, go to section 3.6.2.

Instead, if you wish to explore the remaining methods of creating this model, the current structure will have to be entirely deleted. This can be done using the following procedure.

From the **Select** menu, choose **By All** | **All Geometry** option. The entire structure will be highlighted. Hit the determine keyboard. A message box shown below will appear. Click on the **OK** button.



Figure 3. 47

As we click on the OK button, the following message box appears. Let us say **Yes**. The message box will be dismissed and the entire structure will be deleted.

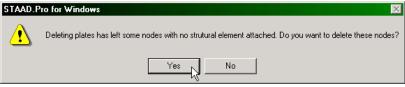


Figure 3.48

Creating the Plates - Method 4

Steps:

The STAAD.Pro GUI contains a facility for generating a mesh of elements from a boundary (or superelement) defined by a set of corner nodes. This facility is in addition to the one we saw in Method 3. The boundary has to form a closed surface and has to be a plane, though that plane can be inclined to any of the global planes.

 The first step in defining the boundary is selecting the corner nodes. If these nodes do not exist, they must be created before they can be selected. So, either click on the Snap Node/Quad Plates icon or select Geometry | Snap/Grid Node | Plate | Quad menu option as shown in the figures below.

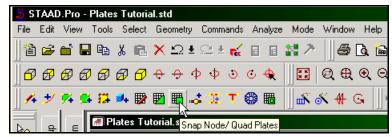


Figure 3. 49

Tutorial 3 3-45

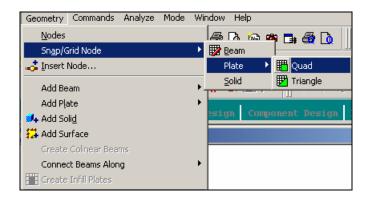


Figure 3. 50

2. A Snap Node/Plate dialog box appears in the data area on the right side of the screen. (We have already seen this dialog box in methods 1 and 2.) As before, let us select the *Linear* tab. In our structure, the elements lie in the X-Z plane. So, in this dialog box, let us choose X-Z as the *Plane* of the grid. The size of the model that can be drawn at any time is controlled by the number of *Construction Lines* to the left and right of the origin of axes, and the *Spacing* between adjacent construction lines. All that we are interested in is the 4 corner nodes of the super-element. So, let us set 1 as the number of lines to the right of the origin along X and Z, and a spacing of **6m** between lines along X and **4m** along Z.

3-46 Tutorial 3

🗱 Snap Node/Plate 🛛 🔀					
Linear Radial Irregular					
Plane Angle of Plane*					
OXY OXX					
• X-Z • Y-Y 0					
O Y-Z O Z-Z					
Grid Origin (m)					
X Y Z					
Construction Lines					
Spacing Left Right m Skew*					
× 0 🖨 1 🖨 6 0					
Z 0 🖨 1 🖨 4 0					
Labels					
Local Coordinate					
Rel. Coords Axis Ids Font					
End(s) Freq.					
X Start 🕶 1 🚔 X Y Z					
Z Start 💌 1 🚔 X Y Z					
Snap to existing nodes too					
Snap Node/Plate Close					

Figure 3. 51

The main drawing area will now look similar to the one shown below.

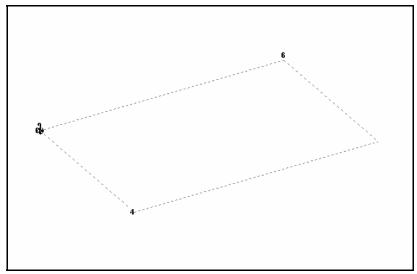


Figure 3. 52

3-48 Tutorial 3

3. To start creating the nodes, let us first activate the Snap Node/Plate button by clicking on it. Holding the 'Ctrl' key down, click at the four corners of the grid as shown below. Those four points represent the four corners of our slab and are (0, 0, 0), (6, 0, 0), (6, 0, 4), and (0, 0, 4). In fact, keeping the 'Ctrl' key pressed and clicking at points on the grid successively, is a way of creating new nodes without connecting those nodes with beams or plates. If the 'Ctrl' key weren't kept pressed, the nodes would become connected.

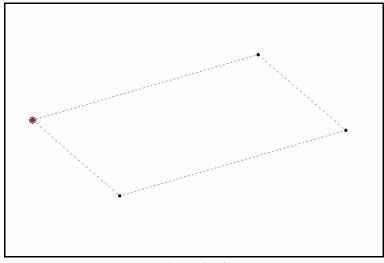


Figure 3. 53

It is worth noting that the purpose of the previous four steps was to merely create the four nodes. Consequently, any of the several methods available in the program could have been used to create those nodes. We could have typed the data into the editor, or in the grid tables of the *Geometry-Plate* page control area, or even used the *Snap Grid/Node – Beam* option of the *Geometry* menu from the top of the screen to graphically create the points.

Tutorial 3-49

4. Let us now **Close** the *Snap Node/Plate* dialog box as shown below.

Snap Node/Plate	: 🛛 🕹				
Linear Radial Irre	gular				
Plane	Angle of Plane*				
O X-Y	C XX				
⊙ x-z	• Y-Y 0				
O Y-Z	O Z-Z				
Grid Origin (m)					
× –	Y Z				
	0 0				
Construction Lines					
Left R X 0 🗣 1 Z 0 🗣 1	6 0				
Labels					
Rel. Coords Axis Ids Font End(s) Freq. X Start 1					
Z Start 💌 1 🐳 X Y Z					
Snap to existing nodes too					
<u>Snap</u>	Node/Plate Close				

Figure 3.54

We are now ready to utilize the second method available in the program for mesh generation.

3-50 Tutorial 3

5. For this, either click on the **Generate Surface Meshing** icon or go to *Geometry* | *Generate Surface Meshing* menu option as shown below.

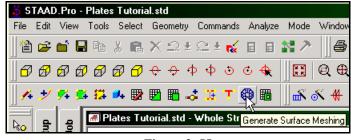


Figure 3.55

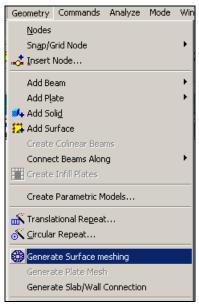


Figure 3.56

6. We now have to select the points which form the boundary of the superelement from which the individual elements will be created. The four points we just created are those four points. So, let us click at the four node points in succession as shown below. Lastly, close the loop by clicking at the start node (or the first clicked point) again.

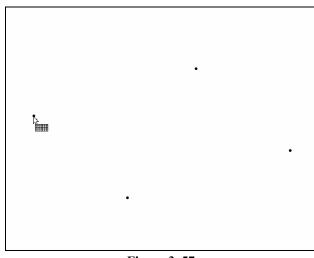


Figure 3. 57

As we click at the start node the second time, the following dialog box comes up. Let us choose the **Quadrilateral Meshing** option and click on the **OK** button.

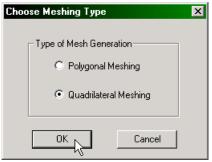


Figure 3.58

3-52 Tutorial 3

7. The Select Meshing Parameters dialog box (as we saw earlier in Method 3), comes up. Notice that this time however, the data for the four corners is automatically filled in. The program used the coordinates of the four nodes we selected to define A, B, C, and D. Provide the **Bias** and the **Divisions** of the model as shown in the figure below. Click on the **Apply** button.

Select Meshing Parameters		×
Model Name: Quad Plate Corners X Y Z A: 0 0 0 0 B: 6 0 0 0 0 C: 6 0 4 0 0 0 D: 0 0 4 0 0 4 0	Length, Bias & Division Length(m) Bias Divin. AB: 6 1 3 BC: 4 1 2 CD: 6 1 3 DA: 4 1 2	Element Type Triangle Quadrilateral Cancel

Figure 3. 59

As we click on the *Apply* button, our model will appear in the drawing area as the one shown below. Click on the button to exit the mesh generating mode.

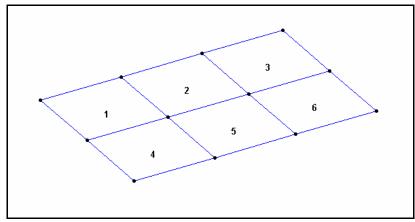


Figure 3.60

3.6.2 Changing the input units of length

As a matter of convenience, for specifying element properties for our structure, it is simpler if our length units are *centimeter* instead of *meter*. This will require changing the current length units of input. The command to be generated is:

UNIT CM KN

Steps:

1. Click on the **Input Units** icon from the appropriate toolbar.



Figure 3. 61

Alternatively, one may select the *Tools* | *Set Current Input Unit* menu option as shown below.

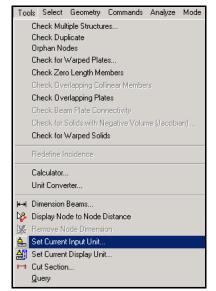


Figure 3. 62

3-54 Tutorial 3

2. In either case, the following dialog box comes up. Set the *Length Units* to **Centimeter** and click on the **OK** button.

S	et Current Inpu	t Units				×
	- Length Units			Force Units		1
	C Inch	C Decimeter		C Pound	C Newton	
	C Foot	C Meter		C KiloPound	C DecaNewton	
	O Millimeter	C Kilometer		O Metric Ton	KiloNewton	
	 Centimeter 			🔿 Kilogram	C MegaNewton	
		ок	ļ	Cancel		

Figure 3. 63

3.6.3 Specifying Element Properties

Just as properties are assigned to members, properties must be assigned to plate elements too. The property required for plates is the plate thickness (or the thickness at each node of elements if the slab has a varying thickness).

The corresponding command which should be generated in the STAAD command file is:

ELEMENT PROPERTY 1 TO 6 THICKNESS 30

Steps:

1. Click on the **Property Page** icon located on the *Structure Tools* toolbar.

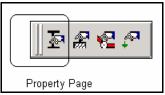


Figure 3. 64

3-56 Tutorial 3

Alternatively, one may go to the *General* | *Property* page from the left side of the screen as shown below.

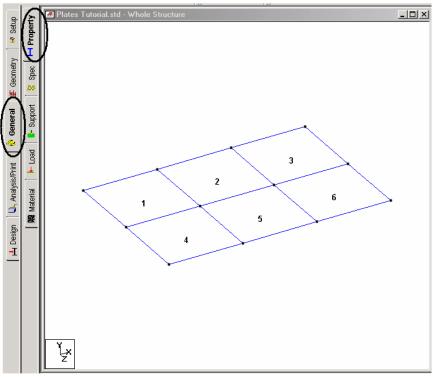


Figure 3.65

2. In either case, the *Properties* dialog box comes up as shown below. Plate thickness is specified through the dialog box available under the **Thickness** button.

👷 Properties - Wh	ole Structure		х		
Section Beta Angle					
Ref Section	Material				
			-		
			-		
Highlight Assigned	d Geometry				
	Edit	Delete			
Values	Section Database	Define			
Materials	Thickness	User Table			
Assignment Method					
C Assign To Selected Beams C Use Cursor To Assign C Assign To Edit List C Assign To View					
	Assign Clos	e Help			

Figure 3. 66

3-58 Tutorial 3

3. The dialog box shown below comes up. Let us provide the *plate thickness* as **30cm**. Notice that the field called *Material* is presently on the **checked** mode. If we keep it that way, the material properties of concrete (E, Poisson, Density, Alpha, etc.) will be assigned along with the plate thickness. The material property values so assigned will be the program defaults. (To see those default values, click on the *Materials* button in the dialog box shown in the previous figure.) Since we want to assign just the default values, let us keep the *Material* box in the **checked** mode itself. Then, click on the **Add** button followed by the **Close** button as shown below.

Plate/Surface Property		x
 Plate/Surface Thickness Surface Thickness 	Plate/Surface Thickness	
	↓ 4 3↓ Node 1: 30 cm	
	+ Node 3: 30 cm	
	1 21 Node 4: 30 cm	
	CONCRETE	
	Add Assign Close Help	

Figure 3. 67

Tutorial 3-59

At this point, the Properties dialog box will look as shown b	below
---	-------

🗱 Properties - Wh	ole Structure	×				
Section Beta Angle	1					
Ref Section	Material					
1 Plate Thicknes						
I						
Highlight Assigned Geometry						
	Edit	Delete				
Values	Section Database	Define				
Materials	Thickness	User Table				
Assignment Method	Assignment Method					
C Assign To Selected Plates © Use Cursor To Assign						
C Assign To Edit List C Assign To View						
I						
	Assian Clos					
	<u>Assign</u> Clos	e Help				

Figure 3.68

4. Since we want the thickness to be applied to all elements of the structure, let us select the *Assignment Method* called **Assign To View** and then click on the **Assign** button as shown in the above figure.

As we click on the *Assign* button, the following message box comes up. Let us click on the **Yes** button to confirm our choice.



Figure 3. 69

The structure will now look as shown below.

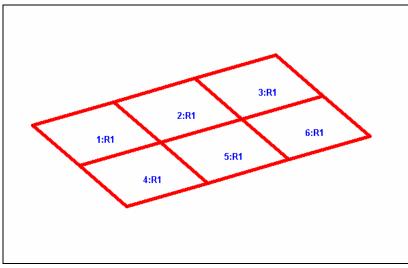


Figure 3. 70

Click anywhere in the drawing area to un-highlight the selected entities. We do this only as a safety precaution. When an entity is highlighted, clicking on any Assign option is liable to cause an undesired attribute to be assigned to that entity.

3.6.4 Specifying Material Constants

In Section 3.6.3, we kept the *Material* check box "on" while assigning the element properties. Consequently, the material constants (E, Density, Poisson's Ratio, etc.) of concrete got assigned to the plates along with the properties, and the following commands were generated in the command file:

```
UNIT METER KN
CONSTANTS
E 2.17185e+007 MEMB 1 TO 6
POISSON 0.17 MEMB 1 TO 6
DENSITY 23.5616 MEMB 1 TO 6
ALPHA 1e-005 MEMB 1 TO 6
```

Hence, there is no longer a need to assign the constants separately. However, if we hadn't assigned them as before, we could go to the menu option *Commands* | *Material Constants* and assign them explicitly as shown in the figure below.

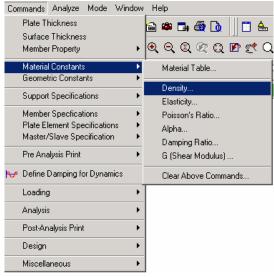


Figure 3.71

3.6.5 Specifying Supports

The slab is fixed-supported along the entire length of two of its sides. However, when modelled as plate elements, the supports can be specified only at the nodes along those edges, and not at any point between the nodes. It hence becomes apparent that if one is keen on better modelling the edge conditions, the slab would have to be modelled using a larger number of elements.

In our case, the commands we need to generate are:

SUPPORTS 1 2 4 5 7 10 FIXED

Steps:

1. To create supports, click on the **Support Page** icon located in the Structure Tools toolbar as shown below.

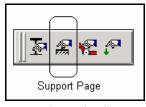


Figure 3. 72

Tutorial 3 3-63

Alternatively, one may go to the *General* | *Support* Page from the left side of the screen.

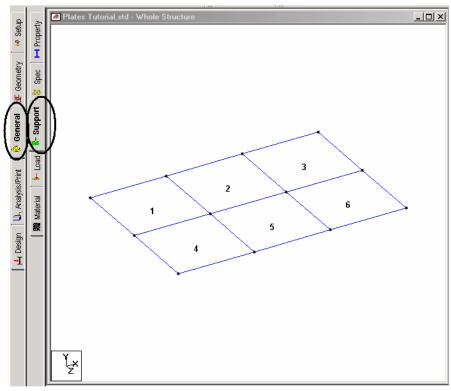


Figure 3.73

3-64 Tutorial 3

- 2. In either case, the *Supports* dialog box comes up as shown in the next figure.
- 3. For easy identification of the nodes where we wish to place the supports, let us turn the **Node Numbers** on.
- 4. Since we already know that nodes 1, 2, 5, 7, 4 and 10 are to be associated with the *Fixed* support, using the *Nodes Cursor* by select these nodes.
- 5. Then, click on the **Create** button in the *Supports* dialog box as shown below.

🐮 🖁 Sup	ports - Whole Structure	×
Ref	Description	
S1	No support	X
0	Edit Create Inment Method Assign To Selected Nodes Assign To View Use Cursor To Assign Assign To Edit List	Delete
	<u>A</u> ssign Close	Help

Figure 3.74

6. The dialog box shown below comes up. The **Fixed** tab happens to be the default which is convenient for this case. Click on the **Assign** button as shown below.

Create Support	t		X
Enforced But Fixed	t Multilinear Spring Pinned	Foundation	Inclined Enforced
- Restrain	t FX	MX MX	
	FY FY	MY	
	FZ FZ	MZ	
	Add Cancel	Assign	Help

Figure 3.75

It is important to understand that the *Assign* button is active because of what we did in step 4 earlier. Had we not selected the nodes before reaching this point, this option would not have been active. After the supports have been assigned, the structure will look like the one shown below.

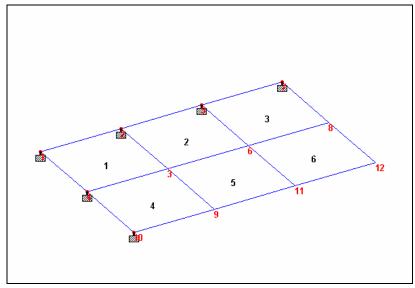


Figure 3. 76

3.6.6 Specifying Primary Load Cases

Three primary load cases have to be created for this structure. Details of these load cases are available at the beginning of this tutorial. The corresponding commands to be generated are listed below.

> UNIT METER KG LOAD 1 DEAD LOAD SELF Y -1.0

LOAD 2 EXTERNAL PRESSURE LOAD ELEMENT LOAD 1 TO 6 PR GY -300

LOAD 3 TEMPERATURE LOAD 1 TO 6 TEMP 40 30

Steps:

1. To create loads, click on the Load Page icon located on the *Structure Tools* tool bar.

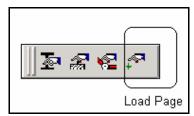


Figure 3.77

Alternatively, one may go to the *General* | *Load* Page from the left side of the screen.

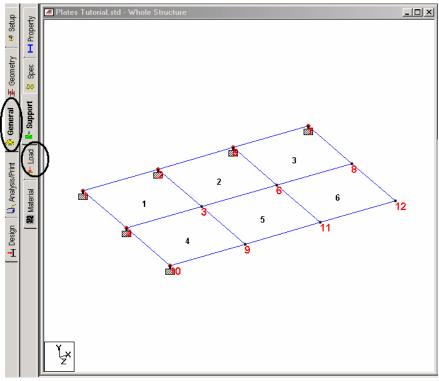


Figure 3.78

2. Notice that the pressure load value listed in the beginning of this tutorial is in *KN* and *meter* units. Rather than convert that value to the current input units, we will conform to those units. The current input units, which we last set while specifying THICKNESS was CENTIMETER. We have to change the force unit to Kilogram and the length units to Meter. To change the units, as before, click on the **Input Units** icon from the top toolbar, or select the *Tools* | *Set Current Input Unit* menu option from the top menu bar. In the *Set Current Input Units* dialog box that comes up, specify the length units as **meter** and the force units as **Kilogram**.

LOAD CASE 1

3. A window titled "Load" appears on the right-hand side of the screen. To initiate the first load case, highlight Load Cases Details and click on the Add button.

Load Definitions Load Cases Details -	×	>	1. Highlight Load Cases Details.
New Add Toggle Load Assignment Method C Assign To Selected Entities C Assign To View	Edit Delete © Use Cursor To Assign C Assign To Edit List	>	2. Click on the Add button.
Assign	Close Help		

Figure 3.79

4. The *Add New Load Cases* dialog box comes up. The drop-down list box against *Loading Type* is available in case we wish to associate the load case we are creating with any of the ACI, AISC or IBC definitions of Dead, Live, Ice, etc. This type of association needs to be done if we intend to use the program's facility for automatically generating load combinations in accordance with those codes.

Notice that there is a check box called *Reducible per UBC/IBC*. This feature becomes active only when the load case is assigned a Loading Type called *Live* at the time of creation of that case. Please refer to STAAD.Pro 2004 Release Report for further details.

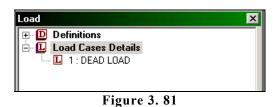
As we do not intend to use the automatic load combination generation option, we will leave the *Loading Type* as **None**. Enter **DEAD LOAD** as the *Title* for *Load Case 1* and click on **Add**.

3-70 Tutorial 3

Add New : Load Cases		x
 Primary Load Generation Define Combinations Auto Load Combination 	Primary	
	Number 1 Loading Type : None	
	Title DEAD LOAD	
	Add Close Help	

Figure 3.80

The newly created load case will now appear under the *Load Cases Details* in the *Load* dialog box.



5. To generate and assign the first load type, **highlight** DEAD LOAD. You will notice that the *Add New Load Items* dialog box shows more options now.



Figure 3.82

6. In the Add New Load Items dialog box, select the **Selfweight Load** option under the Selfweight item. Specify the Direction as **Y**, and the Factor as -1.0. The negative number signifies that the selfweight load acts opposite to the positive direction of the global axis (Y in this case) along which it is applied. Click on the Add button. The selfweight load is applicable to every member of the structure, and cannot be applied on a selected list of members.

Add New : Load Items		×
Selfweight Selfweight Load Nodal Load Member Load Floor Load Floor Load Selfweight Load Selfweight Load Floor Load Selfweight Load Floor Load	Selfweight Load	
	Add Close Help	

Figure 3.83

LOAD CASE 2

7. Next, let us initiate the creation of the second load case which is a pressure load on the elements. To do this, highlight Load Cases **Details.** In the *Add New Load Cases* dialog box, once again, we are not associating the load case we are about to create with any code based Loading Type and so, leave that box as **None**. Specify the *Title* of the second load case as **EXTERNAL PRESSURE LOAD** and click on the **Add** button.

Add	New : Load Cases		×
	Primary Load Generation Define Combinations Auto Load Combination	Primary	
		Number 2 Loading Type : None Reducible per UBC//BC	
		Title EXTERNAL PRESSURE LOAD	
		Add Close Help	

Figure 3.84

To generate and assign the second load type, highlight **EXTERNAL PRESSURE LOAD**.

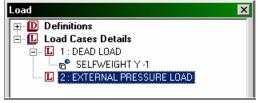


Figure 3.85

8. In the Add New Load Items dialog box, select the **Pressure on Full Plate** option under the *Plate Loads* item enables the load to be applied on the full area of the element. (The *Concentrated Load* is for applying a concentrated force on the element. The *Trapezoidal* and *Hydrostatic* options are for defining pressures with intensities varying from one point to another. The *Partial Plate Pressure Load* is useful if the load is to be applied as a "patch" on a small localised portion of an element.)

Let us keep our focus on the **Pressure on Full Plate** option. Provide -300kg/m2 for *W1* (Force), **GY** as the *Direction* and click on the **Add** button followed by the **Close** button.

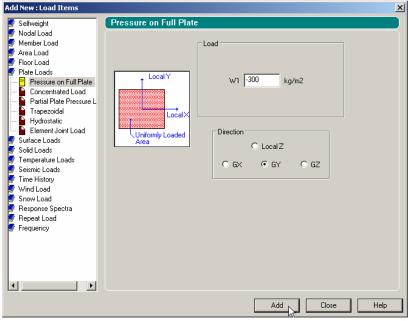


Figure 3.86

3-74 Tutorial 3

Since the pressure load is to be applied on all the elements of the model, the easiest way to do that is to set the *Assignment Method* to Assign To View. Then, click on the Assign button in the *Load* dialog box as shown below.

Load X Definitions Load Cases Details Load Cases Details C Load Cases Details C 2: EXTERNAL PRESSURE LOAD PR GY -300 kg/m2	1. Highlight this
New Add Edit Delete Toggle Load Assignment Method C Assign To Selected Entities © Use Cursor To Assign C Assign To View © Assign To Edit List	
Assign Close Help	- (2. Click on Assign to View) - (3. Click on Assign)

Figure 3.87

After the load has been assigned, the model will look as shown below.

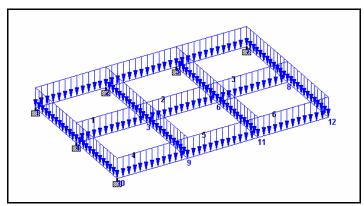


Figure 3.88

LOAD CASE 3

10. Next, let us create the third load case which is a temperature load. The initiation of a new load case is best done using the procedure explained in step 7. In the dialog box that comes up, let us specify the *Title* of the third load case as **TEMPERATURE LOAD** and click on the **Add** button.

Add	New : Load Cases		×
	Primary Load Generation	Primary	
	Define Combinations Auto Load Combination		
		Number 3 Loading Type : None	
		Reducible per UBC//BC	
		Title TEMPERATURE LOAD	
,		Add Close	Help
		T1 0 00	

Figure 3.89

To generate and assign the third load type, as before, highlight **TEMPERATURE LOAD**.

11. Temperature Loads are created from the input screens available under the **Temperature** option in the *Add New Load Items* dialog box.

3-76 Tutorial 3

12. In the Add New Load Items dialog box, make sure that the Temperature item is selected under the Temperature Loads option. Then, provide 40 as the Temperature Change for Axial Elongation and 30 as the Temperature Differential from Top to Bottom and click on the Add button followed by the Close button.

Add New : Load Items			×
 Selfweight Nodal Load Member Load Area Load Floor Load Floor Loads Surface Loads Solid Loads Temperature Loads Stain Seismic Loads Time History Wind Load Snow Load Response Spectra Repeat Load Frequency 	Temperature Temperature Change for Axial Elongation Temperature Differential from Top to Bottom Temperature Differential from Side to Side (Local Z)	40	
		Add	Close Help

Figure 3.90

13. Since we intend to apply the temperature load on all the plates, as before, choose **Assign To View** and click on the **Assign** button in the *Loads* dialog box (see step 9 for explanation).

3.6.7 Creating load combinations

The specifications at the beginning of this tutorial require us to create two combination cases. The commands required are:

LOAD COMBINATION 101 CASE 1 + CASE 2 1 1.0 2 1.0 LOAD COMBINATION 102 CASE 1 + CASE 3 1 1.0 3 1.0

Steps:

LOAD COMBINATION 101

 To initiate and define load case 4 as a load combination, once again, highlight the Load Cases Details option. In the Add New Load Cases dialog box, click on the Define Combinations option from the left-hand side. Enter the Load No: as 101 and the Title as CASE 1 + CASE 2.

Add New : Load Cases		×
 Primary Load Generation Define Combinations Auto Load Combination 	Define Combinations Load No: 101 Name : CASE 1 + CASE 2 Type	
	Add Close He	lp

Figure 3.91

3-78 Tutorial 3

2. Next, in the *Define Combinations* box, select load case 1 from the left side list box and click on the button. Repeat this with load case 2 also. Load cases 1 and 2 will appear in the right side list box as shown in the figure below. (These data indicate that we are adding the two load cases with a multiplication factor of 1.0 and that the load combination results would be obtained by algebraic summation of the results for individual load cases.) Finally, click on the Add button.

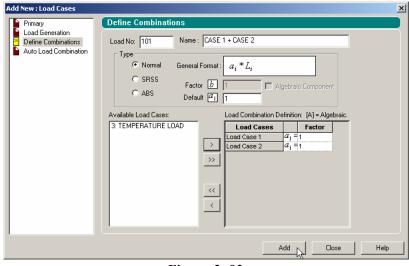


Figure 3.92

Case 101 has now been created.

LOAD COMBINATION 102

3. To initiate and define load case 5 as a load combination, as before, enter the *Load No:* as **102** and the *Title* as **CASE 1 + CASE 3**.

Next, repeat step 2 except for selecting load cases 1 and 3 instead of cases 1 and 2.

Figure 3.93

Thus, load 102 is also created.

If we change our mind about the composition of any existing combination case, we can select the case we want to alter, and make the necessary changes in terms of the constituent cases or their factors.

Load X	Highlight the cases
L (1) x Load 3 New Add Edt Delete Toggle Load Assignment Method C Assign To Selected Entities C Use Cursor To Assign C Assign To View C Assign To Edt.List <u>Assign</u> Close Help	Use Edit/Delete buttons to alter or delete entries

Figure 3.94

Let us exit this dialog box by clicking on the Close button.

It is also worth noting that as load cases are created, a facility for quickly switching between the various cases becomes available at the top of the screen in the form of a load case selection box as shown below.

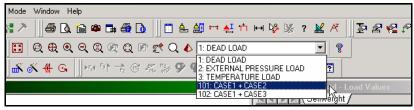


Figure 3.95

We have now completed the task of creating all load cases.

Let us once again save our model by pulling down the **File** menu and selecting the **Save** command or by holding the 'Ctrl' key and pressing the 'S' key.

3.6.8 Specifying the analysis type

The analysis type we are required to do is a linear static type. We will also obtain a static equilibrium report. This requires the command:

PERFORM ANALYSIS PRINT STATICS CHECK

Steps:

1. To specify the Analysis command, first go to **Analysis/Print** Page from the left side of the screen.

Reetup	Plates Tutorial.std - Whole Structure
Æ Geometry Analysis	Analysis/Print Commands X Perform Cable Analysis PDelta Analysis Nonlinear Analysis Change Perform Analysis Perform Imperfection Analysis
H. Design A. Analysis/Print 🔗 General	Print Option Image: No Print Image: Load Data Image: Statics Check Image: Statics Load Image: Statics Load Image: Mode Shapes Image: Both Image: All Image: After Current Add Assign Close Help

Figure 3.96

2. In the *Analysis/Print Commands* dialog box that appears, the instruction for specifying a linear elastic type analysis is provided using the *Perform Analysis* tab. To obtain the static equilibrium report, check the **Statics Check** print option. (In response to this option, a report consisting of the summary of applied loading and summary of support reactions, for each load case, will be produced in the STAAD output file. See section 3.10 for information on viewing this report).

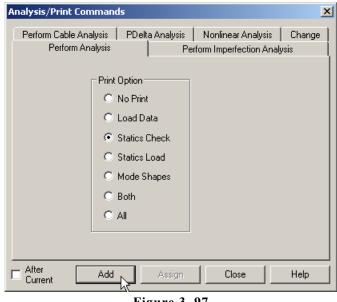


Figure 3.97

Finally, click on the Add button followed by the Close button.

3-84 Tutorial 3

The *Analysis* dialog box in the data area with the newly added instruction will look as shown below.

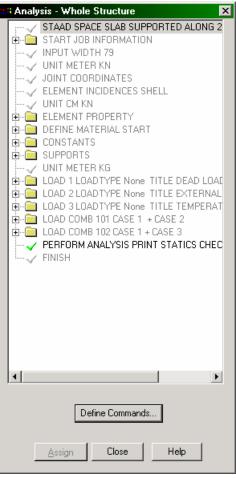


Figure 3.98

Let us save the data once again using the File | Save option.

3.6.9 Specifying post-analysis print commands

Two types of element results can be requested: a) ELEMENT STRESSES at the centroid or any point on the element surface b) the element forces at the nodes. The former consists of stresses and moments per unit width, as explained in sections 1.6.1 and 3.41 of the STAAD Technical Reference Manual. The latter consists of the 3 forces and 3 moments at each node of the elements in the global axis system (see section 3.41 for details).

We would like to obtain both these results. We will also set the units in which these results are printed to KN and Meter for element stresses and Kg and Meter for element forces. This requires the specification of the following commands:

UNIT METER KN PRINT ELEMENT STRESSES LIST 3 UNIT KG METER PRINT ELEMENT FORCE LIST 6

These results will be written in the STAAD output file and can be viewed using the procedure explained in section 3.10.

Steps:

- 1. Go to **Tools** | **Set Current Input Unit** menu option. Set the length and force units to **Meter** and **Kilo Newton** respectively.
- 2. The dialog box for requesting element results is available in the **Post-Print** sub-page of the **Analysis** | **Print** page.

Click on the **Define Commands** button in the data area on the right hand side of the screen.

3-86 Tutorial 3

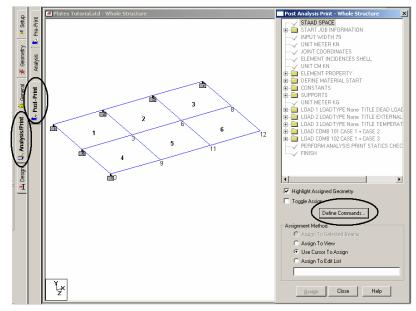


Figure 3.99

3. In the *Analysis/Print Commands* dialog box that appears, select the *Element Forces/Stresses* tab. Choose the **Print Element Stresses** option and click on the **Add** button followed by the **Close** button.

Analysis/Print Commands	X
Section Displacement Force Max Force Story Drift Cg Load List Section Joint Displacement	Section Forces Surface Forces Member Forces Ilement Stress Solid
Print Element Forces Print Element Stresses Print Element Joint Stresses Print Element Stresses at Local X Position D Local Y Position	
After Add Assign Close	Help

Figure 3. 100

4. Again, go to *Tools* | *Set Current Input Unit* menu option. Set the length and force units to **Meter** and **Kilogram** respectively.

Then, repeat steps 2 and 3. In step 3, select the **Print Element Forces** option and click on the **Add** button followed by the **Close** button.

At this point, the *Post Analysis Print* dialog box will look as shown below.

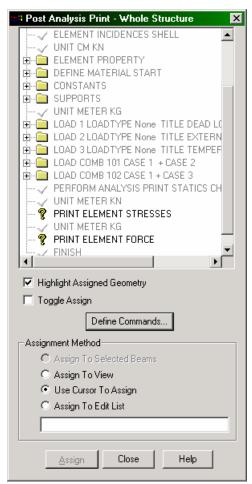


Figure 3. 101

3-88 Tutorial 3

5. To associate the PRINT ELEMENT STRESSES command with element 3, first select the command as shown in the previous

figure. Then, using the Plates Cursor	, click on element no.
3.	

As we select the plate, the *Assignment Method* automatically becomes *Assign to Selected Plates*. Click on the **Assign** button as shown below.

Assignment Method				
Assign To Selected Plates				
O Assign To View				
C Use Cursor To Assign				
C Assign To Edit List				
Assign Close Help				

Figure 3. 102

6. To associate the PRINT ELEMENT FORCE command with element 6, repeat step 5 except for selecting element no. 6 in the place of element no. 3.

We have now completed the tasks of assigning the input for this model. Let us save the file one last time using the **File** | **Save** option.

3.7 Viewing the input command file

Let us now take a look at the data that has been written into the file that we just saved earlier. As we have seen in the previous tutorials, while the model is being created graphically, a corresponding set of commands describing that aspect of the model is being simultaneously written into a command file which is a simple text file. An abstract of those commands was also mentioned under the title "commands to be generated are . ." at the beginning of each section of this tutorial.

The contents of that text file can be viewed in its entirety either by clicking on the **STAAD Editor** icon or, by going to the *Edit* menu and choosing *Edit Input Command File* as shown below.

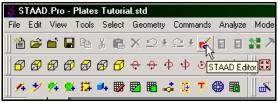




Figure 3. 103

Figure 3. 104

A new window will open up with the data listed as shown here:

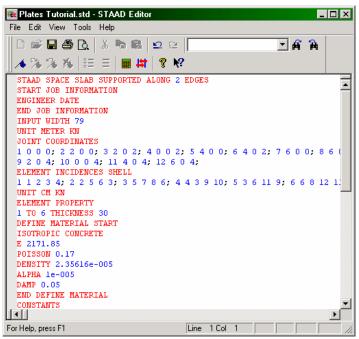


Figure 3. 105

This window and the facilities it contains is known as the *STAAD Editor*.

We could make modifications to the data of our structure in this *Editor* if we wish to do so. Let us *Exit* the *Editor* without doing so by selecting the **File** | **Exit** menu option of the editor window (not the File | Exit menu of the main window behind the editor window).

Instead of using the graphical methods explained in the previous sections, we could have created the entire model by typing these specific commands into the editor. This was one of the methods mentioned in section 3.1 of this tutorial. If you would like to understand that method, proceed to the next section. If you want to skip that part, proceed to section 3.9 where we perform the analysis on this model.

3.8 Creating the model using the command file

Let us now use the command file method to create the model for the above structure. The commands used in the command file are described later in this section.

The STAAD.Pro command file may be created using the built-in editor, the procedure for which is explained further below in this section. 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. A typical editor screen is shown below to illustrate its general appearance.

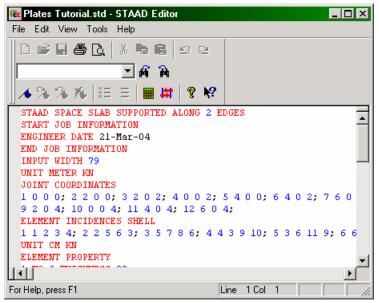


Figure 3. 106

3-92 Tutorial 3

To access the built-in editor, first start the program using the procedure explained in Section 3.3. Next, follow step 1 of Section 3.4 (also, see Figures below).

New	×
Plane File Name : Plane Plates Tutori Truss Location : C: \test\	a
A SPACE structure, which is a three-dimensional framed stru any plane, is the most general. Length Units C Inch C Foot C Foot C Meter C Milimeter C Centimeter C Metric 1	C Newton nd C DecaNewton
Display this dialog box at the Startup	Cancel Help

Figure 3. 107

You will then encounter the dialog box shown below. In that dialog box, choose *Open STAAD Editor*.

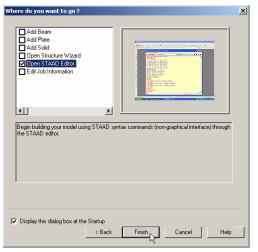
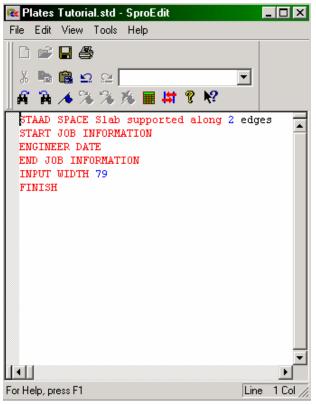


Figure 3. 108

Tutorial 3-93



At this point, the editor screen will open as shown below.

Figure 3. 109

Delete all the command lines displayed in the editor window and type the lines shown in bold below (You don't have to delete the lines if you know which to keep and where to fill in the rest of the commands). The commands may be typed in upper or lower case letters. Usually the first three letters of a keyword are all that are needed -- the rest of the letters of the word are not required. The required letters are underlined. ("SPACE" = "SPA" = "space" = "spa")

Actual input is shown in bold lettering followed by explanation.

STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES

Every input has to start with the word STAAD. The word SPACE signifies that the structure is a space frame structure (3-D) and the geometry is defined through X, Y and Z coordinates.

<u>UNIT METER KN</u>

Specifies the unit to be used for data to follow.

<u>JOI</u>NT <u>COO</u>RDINATES 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

Joint number followed by X, Y and Z coordinates are provided above. Semicolon signs (;) are used as line separators. That enables us to provide multiple sets of data on one line. For example, node 6 has (X, Y, Z) coordinates of (4, 0, 2).

ELEMENT INCIDENCES SHELL 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. For example, element 3 is defined as connected between the nodes 5, 7, 8 and 6.

<u>UNIT CM KN</u> <u>ELEMENT PRO</u>PERTY 1 <u>TO</u> 6 <u>THI</u>CKNESS 30

The length unit is changed from meter to centimeter. Element properties are then provided by specifying that the elements are 30 cm thick.

UNIT METER KN CONSTANTS E 2.17185e+007 ALL POISSON 0.17 ALL DENSITY 23.5616 ALL ALPHA 1e-005 ALL

Material constants, which are E (modulus of elasticity), Density, Poisson's Ratio and Alpha, are specified following the command CONSTANTS. Prior to this, the input units are changed to Meter and KN.

Joints 1, 2, 4, 5, 7 and 10 are defined as fixed supported. This will cause all 6 degrees of freedom at these nodes to be restrained.

<u>UNI</u>T <u>KG</u> LOA</u>D 1 DEAD LOAD

Force units are changed from KN to KG to facilitate the input of loads. Load case 1 is then initiated along with an accompanying title.

SELFWEIGHT Y -1

Load case 1 consists of selfweight of the structure acting in the global Y direction with a factor of -1.0. Since global Y is vertically upward, the factor of -1.0 indicates that this load will act downwards.

LOAD 2 EXTERNAL PRESSURE LOAD

Load case 2 is initiated along with an accompanying title.

ELEMENT LOAD 1 <u>TO</u> 6 <u>PR GY</u> -300

Load 2 is a pressure load on the elements. A uniform pressure of 300Kg/m2 is applied on all the elements. GY indicates that the load is in the global Y direction. The negative sign (-300) indicates that the load acts opposite to the positive direction of global Y.

LOAD 3 TEMPERATURE LOAD

Load case 3 is initiated along with an accompanying title.

<u>TEM</u>PERATURE <u>LOA</u>D 1 <u>TO</u> 6 <u>TEM</u>P 40 30

Load 3 is a temperature load. All the 6 elements are subjected to a in-plane temperature increase of 40 degrees and a temperature variation across the thickness of 30 degrees. This increase is in the same temperature units as the Alpha value specified earlier under CONSTANTS.

LOAD COMB 101 CASE 1 + CASE 2 1 1.0 2 1.0

Load combination 101 is initiated along with an accompanying title. Load cases 1 and 2 are individually factored by a value of 1.0, and the factored values are combined algebraically.

LOAD COMB 102 CASE 1 + CASE 3 1 1.0 3 1.0

Load combination 102 is initiated along with an accompanying title. Load cases 1 and 3 are individually factored by a value of 1.0, and the factored values are combined algebraically.

PERFORM ANALYSIS PRINT STATICS CHECK

The above command instructs the program to proceed with the analysis. A static equilibrium report is also requested with the help of the words PRINT STATICS CHECK.

<u>UNIT MET</u>ER <u>KN</u> <u>PRI</u>NT <u>ELE</u>MENT <u>STR</u>ESS <u>LIS</u>T 3

The stresses and unit width moments are requested at the centroid of element 3 in KN and Meter units.

<u>UNIT KG MET</u>ER <u>PRI</u>NT <u>ELE</u>MENT <u>FOR</u>CE <u>LIS</u>T 6

The forces and moments for all 6 d.o.f at the corner nodes of element 6 are requested in KG and Meter units.

<u>FIN</u>ISH

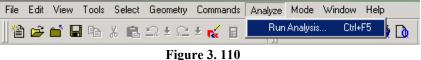
This command terminates the STAAD run.

Let us save the file and exit the editor.

Tutorial 3 3-97

3.9 Performing the analysis and design

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



If the structure has not been saved after the last change was made, you should save the structure first by using the Save command from the *File* menu.

When you select the *Run Analysis* option from the *Analyze* menu, the following dialog box appears:

Select Analysis	Engine	х
Analysis Option		
	 STAAD Analysis 	
	C Stardyne Advanced Analysis	
Run	Analysis Cancel Apply Help	

Figure 3. 111

We are presented with the choice of 2 engines : the STAAD engine and the STARDYNE Advanced Analysis engine. The STARDYNE Analysis engine is suitable for advanced problems such as Buckling Analysis, Modal Extraction using various methods, etc. However, if the calculations call for steel or concrete design, UBC

load generation, etc., we have to select the STAAD engine. For this problem, let us use the **STAAD** engine.

Then, click on the Run Analysis button.

As the analysis progresses, several messages appear on the screen as shown in the next figure.

a STAAD Analysis and Design	
<pre>++ Processing Element Stiffness Matrix. ++ Processing Global Stiffness Matrix. ++ Finished Processing Global Stiffness Matrix. ++ Finished Triangular Factorization. ++ Finished Joint Displacements. ++ Galculating Joint Displacement Calculation. ++ Calculating Member Forces. ++ Analysis Successfully Completed ++ ++ Processing Element Forces. ++ Processing Element Stresses. ++ Creating Displacement File (DSP) ++ Creating Reaction File (REA) ++ Creating Element JT Stress File (EJT) ++ Creating Element JT Force File (ECF) ++ Done. 0 Error(s), 0 Warning(s)</pre>	21:40:0 21:40:0 0 sec 21:40:0 0 sec 21:40:0 0 sec 21:40:1 21:40:1 21:40:1 21:40:1 21:40:1 21:40:1 21:40:1 21:40:1
** End STAAD.Pro Run Elapsed Time = 1 Secs ** Output Written to File: Plates Tutorial.anl	_
 View Output File Go to Post Processing Mode Stay in Modelling Mode 	Done

Figure 3. 112

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

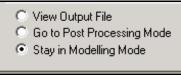


Figure 3. 113

3-100 Tutorial 3

These options are indicative of what will happen after we click on the *Done* button.

The *View Output File* option allows us to view the output file created by STAAD. The output file contains the numerical results produced in response to the various input commands we specified during the model generation process. It also tells us whether any errors were encountered, and if so, whether the analysis and design was successfully completed or not. Section 3.10 (also, see section 1.9) offers additional details on viewing and understanding the contents of the output file.

The Go To Post Processing Mode option allows us to go to graphical part of the program known as the Post-processor. This is where one can extensively verify the results, view the results graphically, plot result diagrams, produce reports, etc. Section 3.11 explains the Post processing mode in greater detail.

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

3.10 Viewing the output file

During the analysis stage, an output file containing results, warnings and messages associated with errors if any in the output, is produced. This file has the extension .anl and may be viewed using the output viewer. Section 1.9 of this manual contains information on viewing this file.

In Sections 3.6.8 and 3.6.9, we had provided instructions to the program to write some very specific results in the output file. Let us examine those results.

PRINT STATICS CHECK

This instruction was provided so that we could view a report indicating if the structure is in equilibrium for the various load cases. If we scroll down the output file, we will see information which looks like the following:

```
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 1 )
 SUMMATION FORCE-X =
                         0.00
 SUMMATION FORCE-Y = -17298.83
 SUMMATION FORCE-Z =
                         0.00
 SUMMATION OF MOMENTS AROUND THE ORIGIN-
 MX=
        34597.65 MY=
                         0.00 MZ=
                                    -51896.48
***TOTAL REACTION LOAD( KG METE ) SUMMARY (LOADING
                                                      1)
 SUMMATION FORCE-X =
                         0.00
 SUMMATION FORCE-Y = 17298.83
 SUMMATION FORCE-Z =
                         0 00
 SUMMATION OF MOMENTS AROUND THE ORIGIN-
 MX=
       -34597.65 MY=
                         0.00 MZ=
                                    51896.48
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING
                                                 1)
MAXIMUMS AT NODE
```

X = 0.00000E+00	0		
Y = -3.20681E-01	12		
Z = 0.00000E+00	0		
RX = 9.80406E-04	12		
RY = 0.00000E+00	0		
RZ = -6.49355E-04	9		

For each primary load case, the report consists of:

- a. Summary of total applied loads for all 6 degrees of freedom, with moments calculated about the origin of the coordinate system (0, 0, 0)
- b. Summary of total reactions from the supports of the structure, with moments calculated about the origin of the coordinate system (0, 0, 0)
- c. Maximum displacements (3 translations and 3 rotations) in the structure induced by this load case

Each of the 6 terms of item (a) must be equal and opposite to the corresponding term of item (b). A failure to meet this criteria would mean the analysis results (for a linear elastic analysis) may be erroneous. Factors such as instability conditions or improperly applied loads can cause the equilibrium check to fail.

It is quite important to examine the maximum displacements since they tell us whether the deformations are within tolerable limits.

Since the information presented above is for load case 1, and since load case 1 is the selfweight of the structure, we can conclude that the structure weighs 17298.83 Kgs.

STRESSES FOR ELEMENT 3

This information consists of the stress and "moment over unit width" values at the centroid of element 3. A typical sample is presented below.

ELEMENT STRESSES FORCE, LENGTH UNITS = KN METE						
FORCE OR STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH						
ELEMENT LO	DAD SQX VONT	SQY VONB	MX SX	MY SY	MXY SXY	
3	1 -18.13 1308.63	72.86 1308.63	-3.96 0.00	-20.42 0.00		
					24 ANGLE= -11.1 .24 ANGLE= -11.1	

These terms are explained in Section 1.6.1 of the STAAD Technical Reference Manual in elaborate detail. Example problem 18 of the Examples Manual explains the method involved in calculating the principal stresses SMAX, SMIN and TMAX.

FORCES FOR ELEMENT 6

This information consists of the force and moment values at each of the 4 corner nodes of element 6. A typical sample is presented below.

```
      ELEMENT FORCES
      FORCE,LENGTH UNITS= KG
      METE

      JOINT
      FX
      FY
      FZ
      MX
      MY
      MZ

      ELE.NO. 6 FOR LOAD CASE 1

      6
      0.0000E+00
      4.5323E+02
      0.0000E+00
      -1.1313E+03
      0.0000E+00
      7.9082E+02

      8
      0.0000E+00
      5.0615E+02
      0.0000E+00
      -3.2047E+02
      0.0000E+00
      2.3981E+02

      12
      0.0000E+00
      -7.2078E+02
      0.0000E+00
      1.0346E-03
      0.0000E+00
      -1.3733E-03

      11
      0.0000E+00
      -2.3860E+02
      0.0000E+00
      -4.6697E+02
      0.0000E+00
      -6.0136E+02
```

The corner forces are in the global coordinate system. These values may be used to create a free body diagram of the element and verify that the element is in equilibrium. In this case for example, the FX and FZ forces are zero, and sum of the FY forces is also zero.

The complete text of the output file is shown in the next few pages.

```
*****
         *
                    STAAD.Pro
                   Version
                                  Bld
         *
                   Proprietary Program of
                   Research Engineers, Intl.
         *
                   Date=
                   Time=
               USER ID:
         *****
 1. STAAD SPACE SLAB SUPPORTED ALONG 2 EDGES
 2. START JOB INFORMATION
 3. ENGINEER DATE
 4. END JOB INFORMATION
 5. INPUT WIDTH 79
 6. UNIT METER KN
 7. JOINT COORDINATES
 8. 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. 9 2 0 4; 10 0 0 4; 11 4 0 4; 12 6 0 4
10 ELEMENT INCIDENCES SHELL
11. 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
12. UNIT CM KN
13. ELEMENT PROPERTY
14. 1 TO 6 THICKNESS 30
15. DEFINE MATERIAL START
16. ISOTROPIC CONCRETE
17. E 2171.85
18. POISSON 0.17
19. DENSITY 2.35616E-005
20. ALPHA 1E-005
21. DAMP 0.05
22. END DEFINE MATERIAL
23. CONSTANTS
24. MATERIAL CONCRETE MEMB 1 TO 6
25. SUPPORTS
26. 1 2 4 5 7 10 FIXED
27 T.OAD 1 DEAD T.OAD
28. SELFWEIGHT Y -1
29. LOAD 2 EXTERNAL PRESSURE LOAD
30. UNIT METER KG
31. ELEMENT LOAD
32. 1 TO 6 PR GY -300
33. LOAD 3 TEMPERATURE LOAD
34. TEMPERATURE LOAD
35. 1 TO 6 TEMP 40 30
36. LOAD COMB 101 CASE 1 + CASE 2
37. 1 1.0 2 1.0
38. LOAD COMB 102 CASE 1 + CASE 3
39. 1 1.0 3 1.0
40. PERFORM ANALYSIS PRINT STATICS CHECK
        PROBLEM STATISTICS
 NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =
                                           12/
                                                    6/ 6
 ORIGINAL/FINAL BAND-WIDTH= 8/ 5/ 36 DOF
                            3, TOTAL DEGREES OF FREEDOM =
                                                            72
 TOTAL PRIMARY LOAD CASES =
 SIZE OF STIFFNESS MATRIX =
                               3 DOUBLE KILO-WORDS
 REQRD/AVAIL. DISK SPACE = 12.1/ 3521.0 MB, EXMEM = 461.2 MB
      STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.
                                                           1
      DEAD LOAD
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 1 )
   SUMMATION FORCE-X =
                          0.00
   SUMMATION FORCE-Y = -17298.83
   SUMMATION FORCE-Z =
                            0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
                                 0 00 MZ=
  MX=
           34597.66 MY=
                                               -51896 48
```

```
***TOTAL REACTION LOAD ( KG METE ) SUMMARY (LOADING 1 )
  SUMMATION FORCE-X =
                          0.00
  SUMMATION FORCE-Y =
                       17298.83
  SUMMATION FORCE-Z =
                         0 00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
         -34597.66 MY=
  MX=
                                0.00 MZ=
                                             51896.48
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING 1)
        MAXIMUMS AT NODE
  X = 0.0000E+00
                    0
  Y = -3.20681E-01
                     12
  Z = 0.0000E+00
                       0
  RX= 9.80406E-04
                     12
  RY= 0.00000E+00
                       0
  RZ= -6.49355E-04
                      9
      STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO.
                                                          2
      EXTERNAL PRESSURE LOAD
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 2 )
  SUMMATION FORCE-X =
                          0.00
   SUMMATION FORCE-Y =
                       -7200.00
   SUMMATION FORCE-Z =
                          0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
                                0.00 MZ= -21600.00
           14400.00 MY=
  MX=
***TOTAL REACTION LOAD ( KG METE ) SUMMARY (LOADING 2 )
  SUMMATION FORCE-X =
                          0.00
   SUMMATION FORCE-Y =
                        7200.00
   SUMMATION FORCE-Z =
                          0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
                                0.00 MZ=
  MX=
        -14400.00 MY=
                                             21600.00
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING
                                             2)
       MAXIMUMS AT NODE
                    0
  X = 0.0000E+00
  Y = -1.33471E-01
                     12
  z = 0.0000E+00
                      0
  RX= 4.08058E-04
                     12
  RY= 0.00000E+00
                       0
  RZ= -2.70270E-04
                      9
      STATIC LOAD/REACTION/EQUILIBRIUM SUMMARY FOR CASE NO. 3
      TEMPERATURE LOAD
***TOTAL APPLIED LOAD ( KG METE ) SUMMARY (LOADING 3 )
  SUMMATION FORCE-X = 0.00
   SUMMATION FORCE-Y =
                           0.00
   SUMMATION FORCE-Z =
                          0.00
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX=
             0.00 MY=
                          0.00 MZ=
                                                0.00
***TOTAL REACTION LOAD ( KG METE ) SUMMARY (LOADING
                                                  3)
   SUMMATION FORCE-X = -0.01
   SUMMATION FORCE-Y =
                          0.00
   SUMMATION FORCE-Z =
                          0.02
  SUMMATION OF MOMENTS AROUND THE ORIGIN-
  MX=
             0.00 MY= -0.10 MZ=
                                                  0.00
```

Tutorial 3 3-107

```
MAXIMUM DISPLACEMENTS ( CM /RADIANS) (LOADING
                                                    3)
                     AT NODE
          MAXIMUMS
    X = 2.01178E-01
                        12
    Y = 8.97365E-01
                         12
                        11
    z = 1.66240E-01
    RX= -3.51264E-03
                         12
    RY= -2.41785E-04
                        11
    RZ= 2.62398E-03
                        12
  41. UNIT METER KN
   42. PRINT ELEMENT STRESSES LIST 3
  ELEMENT STRESSES
                   FORCE, LENGTH UNITS= KN METE
  -----
    STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH
 ELEMENT LOAD
                    sqx
                              SOY
                                           мх
                                                       MY
                                                                   MXY
                    VONT
                              VONB
                                           sx
                                                      SY
                                                                   SXY
                    TRESCAT TRESCAB
         1
                 -18.13
    3
                            72.86
                                         -3.96
                                                  -20.42
                                                                 -3.35
                 1308.63 1308.63
                                        0.00
                                                    0.00
                                                                  0.00
     1404.81 1404.81
TOP: SMAX= -220.33 SMIN= -1404.81 TMAX= 592.24 ANGLE= -11.1
     BOTT: SMAX= 1404.81 SMIN= 220.33 TMAX=
                                                   592.24 ANGLE= -11.1
                                                  -8.50
                  -7.54 30.33 -1.65
                                                            -1.39
           2
                  544.67
                             544.67
                                         0.00
                                                     0.00
                                                                  0.00
                  584.70
                            584.70

        TOP:
        SMAX=
        -91.70
        SMIN=
        -584.70
        TMAX=
        246.50
        ANGLE=
        -11.1

        BOTT:
        SMAX=
        584.70
        SMIN=
        91.70
        TMAX=
        246.50
        ANGLE=
        -11.1

        3
        96.73
        -59.42
        -30.45
        -14.83
        18.43

                          5300.82
                10779.68
                                      -5044.81
                                                -2309.43
                                                               3890 01
                10912.04
                            5585.51
     TOP : SMAX= 269.79 SMIN=-10642.25 TMAX= 5456.02 ANGLE= -34.9
     BOTT: SMAX= 624.74 SMIN= -4960.77 TMAX= 2792.75 ANGLE= -36.2
                                                              -4.74
         101
                 -25.67
                             103.18
                                     -5.61
0.00
                                                  -28.92
                 1853.30
                          1853.30
                                                    0.00
                                                                  0.00
                1989.51
                          1989.51
     TOP : SMAX= -312.03 SMIN= -1989.51 TMAX= 838.74 ANGLE= -11.1
     BOTT: SMAX= 1989.51 SMIN= 312.03 TMAX= 838.74 ANGLE= -11.1
                  78.60
                            13.44
                                       -34.41
                                                                15.08
         102
                                                   -35 24
                                                            3890.01
                10642.95
                            5713.09
                                       -5044.81
                                                  -2309.43
               11074.45
                           6408.55
     TOP : SMAX= -923.15 SMIN=-11074.45 TMAX= 5075.65 ANGLE= -37.3
     BOTT: SMAX= 1848.83 SMIN= -4559.72 TMAX= 3204.28 ANGLE= -32.1
        **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
                                                              MAXIMUM
          MAXIMUM MINIMUM MAXIMUM MAXIMUM
          PRINCIPAL
                     PRINCIPAL
                                       SHEAR
                                                   VONMISES
                                                                TRESCA
           STRESS
                        STRESS
                                      STRESS
                                                    STRESS
                                                                  STRESS
        1.989513E+03 -1.107445E+04 5.456020E+03 1.077968E+04 1.107445E+04
PLATE NO. 3
                           3
                                          3
                                                        3
                                                                    3
CASE NO.
             101
                           102
                                          з
                                                        з
                                                                    102
  43. UNIT METER KG
```

```
44. PRINT ELEMENT FORCE LIST 6
 ELEMENT FORCES
                 FORCE, LENGTH UNITS= KG METE
  _____
 **NOTE- IF A COMBINATION INCLUDES A DYNAMIC CASE OR IS AN SRSS OR ABS COMBINATION
        THEN RESULTS CANNOT BE COMPUTED PROPERLY.
   GLOBAL CORNER FORCES
JOINT
        FX
                   FY
                              FZ
                                         ΜХ
                                                   MY
                                                              MZ
            ELE.NO.
                      6 FOR LOAD CASE
                                        1
   6 0.0000E+00 4.5323E+02 0.0000E+00 -1.1313E+03 0.0000E+00 7.9082E+02
   8 0.0000E+00 5.0615E+02 0.0000E+00 -3.2047E+02 0.0000E+00 2.3981E+02
  12 0.0000E+00 -7.2078E+02 0.0000E+00 1.0346E-03 0.0000E+00 -1.3733E-03
  11 0.0000E+00 -2.3860E+02 0.0000E+00 -4.6697E+02 0.0000E+00 -6.0136E+02
           ELE.NO.
                     6 FOR LOAD CASE
                                          2
   6 0.0000E+00 1.8864E+02 0.0000E+00 -4.7087E+02 0.0000E+00 3.2915E+02
   8 0.0000E+00 2.1067E+02 0.0000E+00 -1.3338E+02 0.0000E+00 9.9813E+01
  12 0.0000E+00 -3.0000E+02 0.0000E+00 8.9143E-05 0.0000E+00 1.6633E-05
  11 0.0000E+00 -9.9306E+01 0.0000E+00 -1.9436E+02 0.0000E+00 -2.5029E+02
           ELE.NO.
                     6 FOR LOAD CASE
                                         з
   6 -2.9880E+05 6.6190E+02 -3.0717E+05 6.3684E+03 2.7912E+03 -5.5444E+03
   8 3.0633E+05 -9.9013E+02 -3.2773E+05 4.3052E+03 -3.7431E+03 4.3521E+03
  12 3.2019E+05 4.6301E-03 3.2019E+05 -6.0036E+03 4.0162E-03 6.0036E+03
  11 -3.2773E+05 3.2822E+02 3.1471E+05 -4.0135E+03 9.5183E+02 -2.8311E+03
           ELE.NO.
                      6 FOR LOAD CASE 101
   6 0.0000E+00 6.4187E+02 0.0000E+00 -1.6022E+03 0.0000E+00 1.1200E+03
   8 0.0000E+00 7.1681E+02 0.0000E+00 -4.5385E+02 0.0000E+00 3.3963E+02
  12 0.0000E+00 -1.0208E+03 0.0000E+00 3.8188E-04 0.0000E+00 -9.6018E-05
  11 0.0000E+00 -3.3790E+02 0.0000E+00 -6.6133E+02 0.0000E+00 -8.5165E+02
           ELE.NO.
                       6 FOR LOAD CASE 102
   6 -2.9880E+05 1.1151E+03 -3.0717E+05 5.2371E+03 2.7912E+03 -4.7536E+03
   8 3.0633E+05 -4.8398E+02 -3.2773E+05 3.9847E+03 -3.7431E+03 4.5919E+03
  12 3.2019E+05 -7.2078E+02 3.2019E+05 -6.0036E+03 4.0162E-03 6.0036E+03
  11 -3.2773E+05 8.9625E+01 3.1471E+05 -4.4804E+03 9.5183E+02 -3.4325E+03
  45. FINISH
          ********** END OF THE STAAD Pro BIN ***********
            **** DATE=
                                                ****
                               TIME=
       For questions on STAAD.Pro, please contact
          Research Engineers Offices at the following locations *
                                            Email
                     Telephone
       * USA:
                  +1 (714)974-2500
                                     support@reiusa.com
       * CANADA
                 +1 (905)632-4771
                                      detech@odandetech.com *
       * CANADA
                 +1 (604)629 6087
                                      staad@dowco.com
       * UK
                  +44 (1454) 207-000
                                       support@reel.co.uk
       * FRANCE
                  +33(0)1 64551084
                                       support@reel.co.uk
       * GERMANY
                 +49/931/40468-71
                                      info@reig.de
       * NORWAY
                  +47 67 57 21 30
                                       staad@edr.no
       * SINGAPORE +65 6225-6015/16
                                       support@reiasia.net
       * INDIA +91(033)2357-3575 support@calcutta.reiusa.com *
       * JAPAN
                  +81(03)5952-6500 eng-eye@crc.co.jp
       * CHINA
                  +86(411)363-1983
                                      support@reiasia.net
      *
         North America
                                      support@reiusa.com
       *
         Europe
                                      support@reel.co.uk
       *
         Asia
                                      support@reiasia.net
```

3.11 Post-Processing

If there are no errors in the input, the analysis is successfully completed. The extensive facilities of the Post-processing mode can then be used to

- a) view the results graphically and numerically
- b) assess the suitability of the structure from the standpoint of safety, serviceability and efficiency
- c) create customized reports and plots

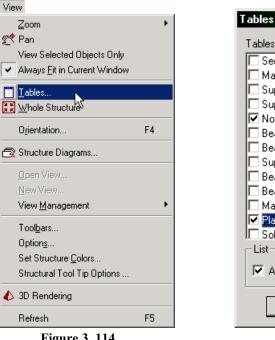
The procedure for entering the post processing mode is explained in section 2.11.1 of this manual.

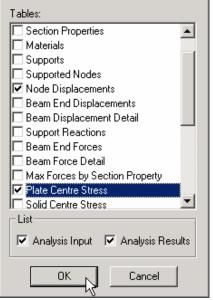
Node results such as displacements and support reactions are available for all models. The methods explained in the first two tutorials – see sections 2.11.2 to 2.11.7 – may be used to explore these. If beams are present in the model, beam results will be available too (see sections 2.11.8 to 2.11.18 for information on these). For this example, we will look at the support reactions. We do not have any beams in our model, so no results will be available for this type of entity.

For plates, the results available are stresses, and "unit width" moments. There are several different methods for viewing these results, as explained in the next few sections.

3.11.1 Viewing stress values in a tabular form

From the View menu, select Tables (or click the right mouse button in the drawing area and select Tables). Choose Plate Centre Stress.





X

Figure 3. 114

Figure 3. 115

		She	ar		Membrane		Bei	ending Moment		
late	L/C	SQX N/mm2	SQY N/mm2	SX N/mm2	SY N/mm2	SXY N/mm2	Mx kNm/m	My kNm/m	Mxy kNm/m	
1	1 DEAD LOA	-0.009	-0.009	0.000	0.000	0.000	-4.009	-4.024	-2.850	
	2 EXTERNAL	-0.004	-0.004	0.000	0.000	0.000	-1.669	-1.675	-1.186	
	3 TEMPERAT	0.046	0.045	-7.183	-6.026	2.740	-54.530	-54.641	3.041	
	101 CASE 1	-0.013	-0.013	0.000	0.000	0.000	-5.678	-5.699	-4.036	
	102 CASE 1 +	0.037	0.036	-7.183	-6.026	2.740	-58.539	-58.665	0.192	
2	1 DEAD LOA	-0.030	0.039	0.000	0.000	0.000	-1.082	-12.080	-4.655	
	2 EXTERNAL	-0.012	0.016	0.000	0.000	0.000	-0.450	-5.028	-1.937	
	3 TEMPERAT	0.076	0.084	-7.137	-1.328	3.440	-53.051	-39.690	8.983	
	101 CASE 1	-0.043	0.056	0.000	0.000	0.000	-1.533	-17.108	-6.592	
	102 CASE 1 +	0.046	0.123	-7.137	-1.328	3.440	-54.133	-51.770	4.328	
3	1 DEAD LOA	-0.018	0.073	0.000	0.000	0.000	-3.960	-20.417	-3.348	
	2 EXTERNAL	-0.008	0.030	0.000	0.000	0.000	-1.648	-8.498	-1.393	
	3 TEMPERAT	0.097	-0.059	-5.045	-2.309	3.890	-30.447	-14.826	18.432	
	101 CASE 1	-0.026	0.103	0.000	0.000	0.000	-5.608	-28.915	-4.741	
	102 CASE 1 +	0.079	0.013	-5.045	-2.309	3.890	-34.407	-35.244	15.084	
4	1 DEAD LOA	0.071	-0.022	0.000	0.000	0.000	-11.863	-3.558	-5.485	
	2 EXTERNAL	0.029	-0.009	0.000	0.000	0.000	-4.937	-1.481	-2.283	
	3 TEMPERAT	-0.077	0.079	-5.626	-5.374	4.506	-37.063	-30.141	17.817	
	101 CASE 1	0.100	-0.031	0.000	0.000	0.000	-16.800	-5.039	-7.768	
	400.0005.4	0.000	0.077	7 000	C 074	4 200	40.000	000.000	40.000	

The following table will be displayed.

Figure 3. 116

The table has the following tabs:

Shear, Membrane and Bending: These terms are explained in Section 1.6.1 of the STAAD Technical Reference Manual. The individual values for each plate for each selected load case are displayed.

Summary: This tab contains the maximum for each of the 8 values listed in the Shear, Membrane and Bending tab.

Principal and Von Mises: These terms too are explained in Section 1.6.1 of the STAAD Technical Reference Manual. The individual values for each plate for each selected load case are displayed, for the top and bottom surfaces of the elements.

Summary: This tab contains the maximum for each of the 8 values listed in the Principal and Von Mises tab.

Global Moments: This tab provides the moments about the global X, Y and Z axes at the center of each element.

3.11.2 Printing the tables

All of these tables may be printed by clicking the right mouse button in the table area and selecting the *Print* option.

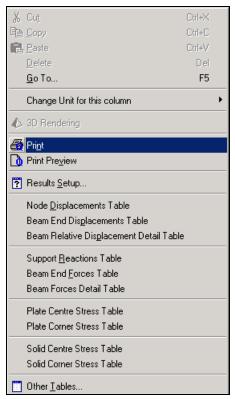


Figure 3. 117

3.11.3 Changing the units of values which appear in the above tables

The length and force units of the stresses and moments are displayed alongside the individual column headings for the terms. To view the values in a different set of units, go to the **Tools** menu and select **Set Current Display Unit**.

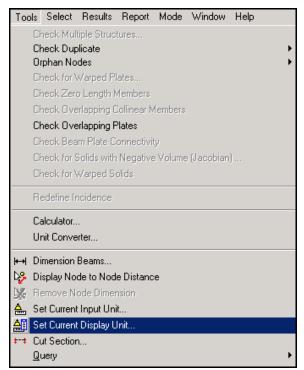


Figure 3. 118

In the dialog box that comes up, click on the *Force Units* tab and specify the required unit from the *Stress* and *Moment* fields. Click on the *Apply* button for the changes to take effect immediately. Once you are sure that you have chosen the proper unit combination, click on the *OK* button.

Options		×
View Highlight	Force Units	
Tolerance	Force	
Assign DIgs Node Labels	Force kg 💌	Show 🖪 曼 dec places
Beam Labels	Distr. Force kN/m	Show 🕄 🖨 dec places
Solid Labels	Moment kNm	Show 🕄 🖨 dec places
Annotation Structure Units	Distr. Moment kNm/m	Show 🖨 dec places
Section Units		Show 3 🖨 dec places
Dimension		Show 3 🖨 dec places
	Rotation rad 💌	Show 3 🚔 dec places
	OK Apply	Cancel Help

Figure 3. 119

3.11.4 Limiting the load cases for which the results are displayed

When we entered the post-processing mode, we chose all load cases in the *Results Setup* dialog box. The tables hence contain results for all the load cases. To change that load list, choose **Select Load Case** from the **Results** menu.

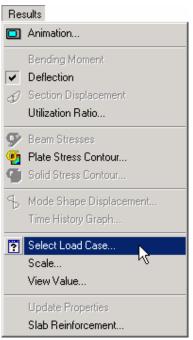


Figure 3. 120

3-116 Tutorial 3

In the dialog box that comes up, select the load cases you want,

and click on the single arrow, so they are transferred from the *Available* category to the *Selected* category. Then, click on the *OK* button.

Results Setup		x
Loads Range Result View Options		
Load Cases		
Available:	Selected:	
2 EXTERNAL PRESSURE LOAD 101 CASE 1 + CASE 2 102 CASE 1 + CASE 3	> 1 DEAD LOAD 3 TEMPERATURE LOAD	
	<	
	OK Apply Help	

Figure 3. 121

3.11.5 Stress Contours

Stress contours are a color-based plot of the variation of stress or moment across the surface of the slab or a selected portion of it. There are 2 ways to switch on stress contour plots:

a) From the left side, select the **Contour** sub-page under the **Plate** page.

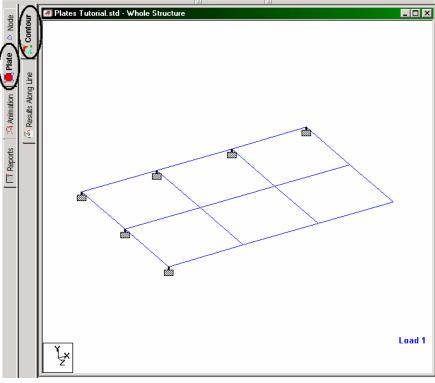


Figure 3. 122

3-118 Tutorial 3

b) From the top of the screen, click on the *Results* menu and select the *Plate Stress Contour* option.



Figure 3. 123

grams Structure	Loads and R	esults	Scales	Labels
Force Limits	Animation	Design Resul	ts Plate	Stress Contour
Force Limits Load Case: Stress Type Stress type: Norral Contour Type Normal Fill C Normal Line Absolute Values View Stress Inde View Stress Inde Re-Index for new Show Displaced	DEAD LOAD	Use Custom D 1 2 3 4 5 7 8 9 10	ivisions	Bitress Contour
Minimum: N/A Maximum: N/A No of values: 15	N/mm2 N/mm2	11 12 13 14 15 15		
	ОК	Cancel	Apply	Help

In each of the above cases, the following dialog box appears.

Figure 3. 124

- From the *Stress type* field, select the specific type of stress for which you want the contour drawn.
- From the *Load Case* selection box, select the load case number.
- Stress values are known exactly only at the plate centroid locations. Everywhere else, they are calculated by linear interpolation between the center point stress values of adjacent plates. The *Enhanced* type contour chooses a larger number of points compared to the *Normal* type contour in determining the stress variation.
- *View Stress Index* will display a small table consisting of the numerical range of values from smallest to largest which are represented in the plot.

Let us set the following:

- Load case 102
- Stress Type Von Mis Top
- Contour Type Normal Fill
- Index based on Center Stress
- View Stress Index
- Re-Index for new view

Force Limits Animation	Design	Besults	Plate Stress Co	ontou
	-			
Load Case: 102: CASE 1 + C/	ASE 3	_		
itress Type				
Stress type: Von Mis Top	•			
Contour Type	Line Curl	tom Divisions		
Normal Fill	ose cus			
C Enhanced Fill		A	В	
	1	<= 1.18385	1.88671	
O Normal Line	2	1.88671	2.58957	
Absolute Values	3	2.58957	3.29243	
Index Based on Center Stress	4	3.29243	3.99528	
	5	3.99528	4.69814	
 View Stress Index 	6	4.69814	5.401	
Re-Index for new view	7	5.401	6.10386	
Show Displaced Shape	8	6.10386	6.80671	
	9	6.80671	7.50957	
Use Custom Limits	7 10 11	7.50957 8.21243	8.91529	
Minimum: 1.18385 N/mm2	12	8.91529	9.61814	
Minimum: 1.18385 N/mm2	13	9.61814	10.321	
	14	10.321	11.0239	
Maximum: 12.4296 N/mm2	15	11.0239	11.7267	
No of	16	11.7267	>= 12.4296	
values: 15				

Figure 3. 125

Click on the Apply button.

The following diagram will be displayed. We can keep changing the settings and click on *Apply* to see all the various possible results in the above facility.

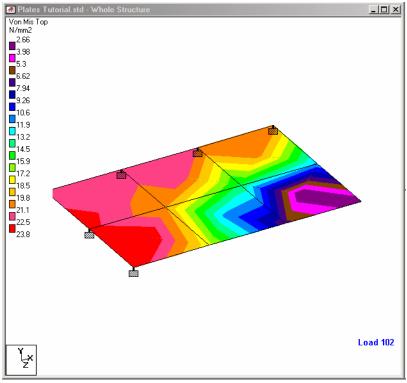
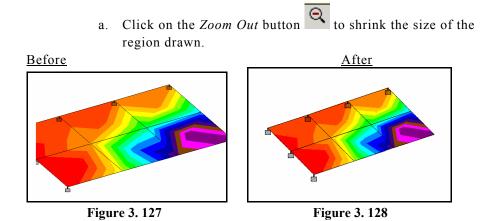


Figure 3. 126

Let us keep the dialog box open to examine the feature (Animation) explained in the next section.

If some portion of the structure appears truncated, we can bring that portion into view by choosing one of the following methods:



b. Use the *Pan* button to physically shift the position of the structure away from the index.

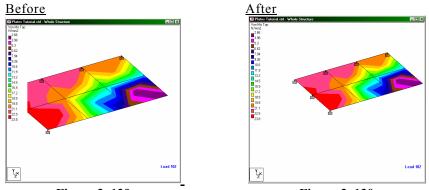


Figure 3. 129

Figure 3. 130

3.11.6 Animating stress contours

The same dialog box shown in the previous section may be used to obtain the stress contours in an animated view. This is a method of getting a "dynamic" instead of static representation of the plot. After making the choices as explained in that section, click on the *Animation* tab of that dialog box. Switch on **Stress** followed by **Apply**.

iagrams								
Structure	Loads an			ales	Labels			
Force Limits	Animation	Design Re	esults	Plate	Stress Contour			
Diagram Type								
C No Animation	I							
O Deflection								
C Section Displacement								
C Mode								
Stress								
⊢Animation Setup								
Full Screen (This can actually	y use less memo	ory than a	large win	dow)			
Extra Frames	E	xtra frames abo	ve minimu	ım neede	d.			
Target FPS 1		ote: The speed uaranteed.	cannot b	е				
Use Metafiles (Note: May sa	for offscreen da ave mem at expo							
	Note: W	/hile animating ł	hit F12 to	view full s	creen			
		_						
	OK	Close		Apply	Help			

Figure 3. 131

To stop the animation, click on **No Animation** and click **Apply** again.

3.11.7 Creating AVI Files

AVI files are a mechanism by which a dynamic result, such as, a deflection diagram in animation, may be captured and recorded. Presently, this facility is available in STAAD for node deflection, beam section displacement, mode shape and plate stress contour diagrams.

The facility in STAAD for creating AVI files is under the **Tools** menu and is called **Create AVI File**. These files can then be viewed using tools such as the Windows Media Explorer.



Figure 3. 132

The following dialog box will appear.

Create AVI File	×
Total No. of Frames 30	
Frame Rate /sec 6	
Animation Type	1
O Deflection	
O Sectional Displacement	
C Mode Shape	
Stress Contour Setup	
Plate Stress Type Von Mis Top	
Solid Stress Type	
OK J Cancel	1

Figure 3. 133

In an animated view, the movement from one extremity to the other is captured as several frames. The number of frames that comprise such a movement is controlled by specifying a value for Total No. of Frames. The speed of motion is controlled by the Frame Rate /sec. The rest of the options in the above dialog box are for the type of diagram from which the AVI file is to be created. Certain items such as Mode Shape and Plate Stress contour do not become active (remain grayed out) if the required data of that type are not present in the STAAD file, such as a modal extraction, or finite elements. After making the appropriate selections, click **OK**. Following this, another dialog box appears, where we are instructed to specify a file name for the .AVI file. Subsequently, we are prompted for the type of video compression. AVI files can be quite large, and compression is a technique by which one may reduce the size of these files.

Video Compression		×
Compressor:		ОК
Full Frames (Uncompressed)	•	Cancel
Compression Quality:	Þ	Configure About

Figure 3. 134

Finally, a message indicating that the operation was successful will appear at the end of the file creation process as shown below.



As mentioned earlier, other tools like Windows Media Explorer may be used to view the AVI file. The file with the extension .AVI should be located in the same folder where the STAAD input file is present.

3.11.8 Viewing plate results using element query

Element Query is a facility where several results for a specific element can be viewed at the same time from a single dialog box. Let us explore this facility for element 4. Select element 4 and double click on it. Alternatively, select element 4, and from the *Tools* menu, choose *Query* – *Plate*.

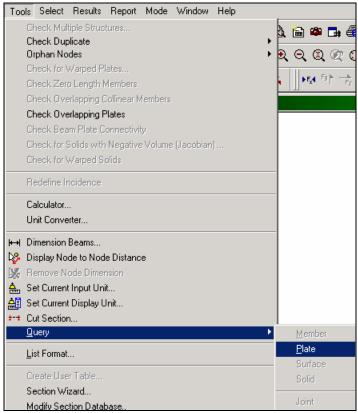


Figure 3. 136

The various tabs of the query box enable one to view various types of information such as the plate geometry, property constants, stresses, etc., for various load cases, as well as print those values.

3-128 Tutorial 3

Some typical screens are shown.

Princ Stress and Di Geometry	isp Property Constar		ier Stresses Center Stresses	
	Plate No : 4			
4 3	Node	m	YZ mm	
×	4 0 3 2 9 2 10 0	0 0 0	2 2 4	
10 9	10 0	0	4	
Edge Lengths & Area-	BC	CD	DA	
Length (m) 2	2	2	2	
Area (cm2)40000				
Area (cm2)40000				

Figure 3. 137

Plates Tutorial.std - I	Plate			×	
Princ Stress a Geometry	Property Constants Plate No : 4	Corner Stresses stants Center Stresses			
а 10	a ³	Node 4 3 9 10	Thickness m 0.3 0.3 0.3 0.3 0.3 VChange Propert	у	
Material Properties Elasticity(kN/mm2) 21.7 Poisson 0.17	······	2402.61 1e-005	CONCRET Assign Ma		
			Print	Close	

Figure 3. 138

Tutorial 3 3-129

📲 Plates Tu	torial.std - Pla	te			×			
Geom	Princ Stress and I	Disp Property (Corner Stre	sses er Stresses			
Geom	euy	Plate No : -		Cont				
	Load List : 1:DEAD LOAD							
Plate	Center Stresses- SQX N/mm2	SQY N/mm2	SX N/mm2	SY N/mm2				
	0.0705826 -0 SXY N/mm2	0.0219774 MX kNm/m	0 MY kNm/m	0 MXY kNm/m				
		1.8626	-3.55841	-5.48485				
- Principal	/Von Mises / Tre		- 1					
	Top (N/mm2)	0.972647	I Von Mis 0.946155	5 Tresca	-			
	Bottom (N/mm2)		0.946155	0.917227				
				Print	Close			

Figure 3. 139

📲 Plates Tut	orial. sto	l - Plate							X		
Geome) ss and Di	Property Constants Disp			Center Stresses Corner Stresses			es		
			Plate No	o:4							
			Load List :			1:DEAD LOAD			•		
94		î i	- Plate (Corner Disp	lacem	ents					
	×	× N		No		lode X		Y cm		Z ¢m	
			4	0		-1.4509e	-009	0			
	У		3 9	0		-0.03449	13	0			
10		1	9	0		-0.10631		0			
		-	10	0		-2.29409	le-00	0			
Plate F	Principal	Stresses									
		SMA N/mn		SMIN N/mm2		rmax I/mm2		Angle			
	Тор	-0.05542	:01 -0).972647	0.45	8614	26.4	37			
Ī	Bottom 0.972647		7 0.	.0554201	0.45	8614	26.4	369			
									_		
						F	Print		Close		

Figure 3. 140

	Futorial.std	·			
Geo	ometry		erty Constants		enter Stresses
Princ Stress		s and Disp	and Disp		Stresses
Load	List : 1:DE	AD LOAD	×	4 10	کہ ^ع ر بر عراق
		-		1	
Node	4	3	9	10	▲
Node Max Top N/mm2	4 0.915746	3 1.17106	0.752406	3.01445	
Мах Тор		-	-		
Max Top N/mm2 Max Bottom N/mm2 SX N/mm2	0.915746	1.17106	0.752406	3.01445	
Max Top N/mm2 Max Bottom N/mm2 SX	0.915746 0.915746	1.17106 1.17106	0.752406	3.01445 3.01445	

Figure 3. 141

The length and force units in which the values appear in the above tables can also be modified using the method explained in section 3.11.3.

3.11.9 Producing an onscreen report

Occasionally, we will come across a need to obtain results conforming to certain restrictions, such as, say, the resultant node displacements for a few selected nodes, for a few selected load cases, sorted in the order from low to high, with the values reported in a tabular form. The facility which enables us to obtain such customized on-screen results is the Report menu on top of the screen.

Let us produce a report consisting of the plate principal stresses, for all plates, sorted in the order from Low to High of the Principal Maximum Stress (SMAX) for load cases 101 and 102.

The first step to do this is to select all the plates using the *Plates Cursor* Then, go to the **Report** | **Plate Results** | **Principal Stresses** menu option as shown below.

Report	
Beam Property	
Joint Displacement Support Reactions	
Beam End Forces Section Displacement Section Forces Beam Stresses	
Plate Results	Plate Stresses
Floor Vibration Report	Principal Stresses

Figure 3. 142

3-132 Tutorial 3

In the dialog box that appears, select the *Loading* tab. Make sure that load cases **101** and **102** are selected as shown below.

Plate Forces	×
Sorting Loading Report	
Load Cases	
Available:	Selected:
1 DEAD LOAD 2 EXTERNAL PRESSURE LOAD 3 TEMPERATURE LOAD	> 101 CASE 1 + CASE 2 102 CASE 1 + CASE 3 >>
	OK Cancel Help

Figure 3. 143

Next, select the *Sorting* tab. Choose **SMAX** under the *Sort by Plate Stress* category and set the *Sorting Order* **List from Low to High**. (If we wish to save this report for future use, we may select the *Report* tab, provide a title for the report, and turn on the *Save ID* option.) Then, click on the **OK** button.

Plate Forces	×
Sorting Loading Report	
Sort By Plate Stress	If Sorting Done
SMAX	
C SMIN	Absolute Values
C TMAX	
	Set Sorting Order
	C List from High to Low
	• List from Low to High
C List with no sort done	
	OK Cancel Help

Figure 3. 144

The following figure shows the table of maximum principal stress with SMAX values sorted from Low to High.

📰 Plates Tutorial.std - Plate Forces 📃 🗖							
Plate	L/C		SMAX kN/m2	SMIN kN/m2	TMAX kN/m2	Angle	
1	102	Тор	-14469.039	-25389.348	5460.155	-39.310	
2	102	Тор	-5096.584	-24016.936	9460.176	-25.434	
4	102	Тор	-4802.948	-25810.191	10503.621	-42.339	
3	102	Тор	-718.747	-20732.025	10006.640	-36.105	
6	102	Тор	-640.096	-2919.152	1139.528	-35.686	
5	102	Тор	-425.714	-11661.900	5618.093	-20.461	
3	101	Тор	-312.033	-1989.513	838.740	78.931	
1	101	Тор	-110.158	-648.272	269.057	45.074	
4	101	Тор	-78.487	-1377.476	649.495	26.437	
2	101	Тор	58.845	-1301.568	680.207	69.876	
1	102	Bottom	204.161	-9879.255	5041.708	-38.739	
6	101	Тор	308.160	-524.463	416.312	53.363	
6	101	Bottom	524.463	-308.160	416.312	53.363	
1	101	Bottom	648.272	110.158	269.057	45.074	
5	101	Тор	785.309	-800.408	792.859	54.271	
5	101	Bottom	800.408	-785.309	792.859	54.271	
4	102	Bottom	1140.090	-11776.030	6458.060	-47.230	
2	101	Bottom	1301.568	-58.845	680.207	69.876	
4	101	Bottom	1377.476	78.487	649.495	26.437	
5	102	Bottom	1394.659	426.818	483.920	3.448	
3	101	Bottom	1989 513	312 033	838 740	78 931	

Figure 3. 145

Tutorial 3 3-135

To print this table, click the right mouse button anywhere within the table. A list of options will appear.

X Cut	Ctrl+X
Fa Copy	Ctrl+C
🕞 Paste	Ctrl+V
Delete	Del
<u>G</u> o To	F5
Change Unit for this column	•
🚯 3D Rendering	
🛃 Pri <u>n</u> t	
D Print Pre <u>v</u> iew	
🕐 Results <u>S</u> etup	
Node <u>D</u> isplacements Table	
Beam End Displacements Table	
Beam Relative Dis <u>p</u> lacement Deta	il Table
Support <u>R</u> eactions Table	
Beam End <u>F</u> orces Table	
Beam Forces Detail Table	
Plate Centre Stress Table	
Plate Corner Stress Table	
Solid Centre Stress Table	
Solid Corner Stress Table	
📋 Other <u>T</u> ables	

Figure 3. 146

Select the print option to get a hardcopy of the report.

To transfer the contents of this table to a Microsoft Excel file, click at the top left corner of the table with the left mouse button. The entire table will become highlighted. Click the right mouse button and select *Copy*. Then, open an Excel worksheet, click at the desired cell and click *Paste*.

3.11.10 Viewing Support Reactions

Since supports are located at nodes of the structure, results of this type are available along with other node results like displacements. To view the reactions graphically, click on the **Node** page on the left side of the screen, and select the **Reactions** sub-page.

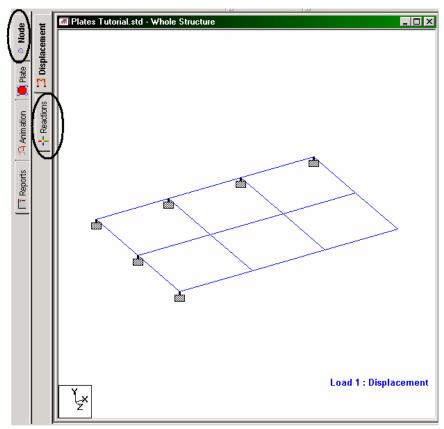


Figure 3. 147

The reactions at the supports will be displayed on the drawing as shown below.

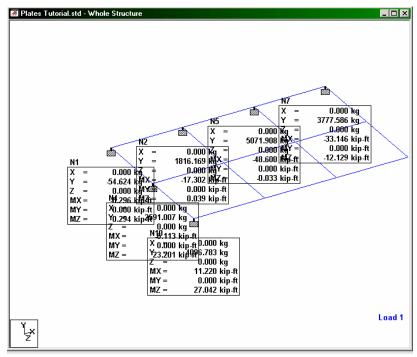


Figure 3. 148

The six values - namely, the 3 forces along global X, Y and Z, and the 3 moments Mx, My and Mz, in the global axis system are displayed in a box for each support node. The words N1, N2, etc. stand for the node numbers of the supports.

3-138 Tutorial 3

We may choose to "un-display" one or more of the 6 terms of each support node in the following manner.

From the Results menu, choose View Value.

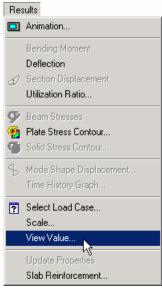


Figure 3. 149

In the dialog box that comes up, click on the **Reactions** tab. Let us **<u>switch off</u>** Global X and Global Z under the *Direct* category. Then, click on the **Annotate** button followed by the **Close** button.

*** Annotation - Plates Tutor	ial.std - Whole Structure 🛛 🗙			
Ranges Beam Results Node Reactions				
Direct	Bending			
🗖 Global X	🔽 Global rX			
🔽 Global Y	🔽 Global rY			
🗖 Global Z	🔽 Global rZ			
Diagram				
🗖 Show Line				
Scale 1e-007 🚔 kg per m				
Scale 1e-007 🚔 kip⁻ft perm				
Remove All	Annotate Close			

Figure 3. 150

The drawing will now contain only the remaining 4 terms (see figure below).

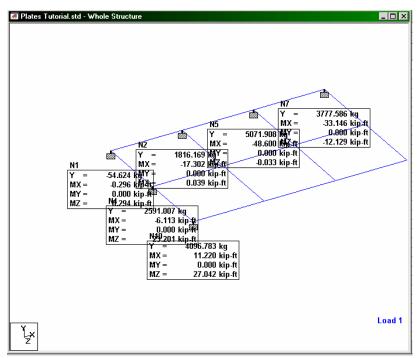


Figure 3. 151

To change the load case for which the reactions are displayed, select the desired case from the load selection box.

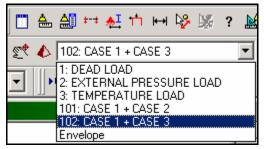


Figure 3. 152

For better clarity in viewing the results in the drawing area (and for reducing the clutter on the screen), a variety of methods are available. For example, keep the mouse pressed on top of Zoom In button, and watch the drawing get progressively bigger. Use the *Pan* button to physically shift the drawing around. Other options like *Dynamic Zoom* and *Zoom Window* buttons may also be used. To restore the original view, click on the *Display Whole Structure* button. (Some of these options are explained in greater detail in the 'Frequently Performed Tasks' section at the end of this manual.)

Icon	Name
•	Zoom In
<u></u>	Pan
Q	Dynamic Zoom
Q	Zoom Window
	Display Whole Structure

3-142 Tutorial 3

📰 Plates Tutorial.std - Support Reactions:							
	NIA AII A S	Summary <u> </u>	invelope /				
	Horizontal Vertical Horizontal Moment						
Node	L/C	Fx kg	Fy kg	Fz kg	Mx kNm	My kNm	Mz kNm
1	1 DEAD LOA	0.000	-54.624	0.000	-0.401	0.000	0.399
	2 EXTERNAL	0.000	-22.735	0.000	-0.167	0.000	0.166
	3 TEMPERAT	172.8E3	2649.422	152.5E3	-78.401	70.559	78.382
	101 CASE 1 +	0.000	-77.359	0.000	-0.568	0.000	0.565
	102 CASE 1 +	172.8E3	2594.799	152.5E3	-78.802	70.559	78.781
2	1 DEAD LOA	0.000	1816.168	0.000	-23.458	0.000	0.053
	2 EXTERNAL	0.000	755.913	0.000	-9.763	0.000	0.022
	3 TEMPERAT	-208.18E3	5522.880	177.2E3	-148.111	361.463	28.141
	101 CASE 1 +	0.000	2572.082	0.000	-33.221	0.000	0.075
	102 CASE 1 +	-208.18E3	7339.049	177.2E3	-171.568	361.463	28.194
4	1 DEAD LOA	0.000	2591.007	0.000	-8.287	0.000	31.456
	2 EXTERNAL	0.000	1078.411	0.000	-3.449	0.000	13.092
	3 TEMPERAT	295.97E3	3055.179	-241.91E3	18.182	-38.685	106.888
	101 CASE 1 +	0.000	3669.418	0.000	-11.737	0.000	44.548
	102 CASE 1 +	295.97E3	5646.186	-241.91E3	9.895	-38.685	138.344

The table on the right side of the screen contains the reaction values for all supports for all selected load cases.

Figure 3. 153

This table can also be displayed from any mode by clicking on the *View* menu, choosing *Tables*, and switching on *Support Reactions*.

The method explained in section 3.11.3 may be used to change the units in which these values are displayed. The summary tab contains the maximum value for each of the 6 degrees of freedom along with the load case number responsible for it.

📰 Plat	🖩 Plates Tutorial.std - Support Reactions:							_ 🗆 ×
	▲ ▲ ▶ ► All All Summary (Envelope /							
Horizontal Vertical Horizontal Moment								
	Node	L/C	Fx kg	Fy kg	Fz kg	Mx kNm	My kNm	Mz kNm
Max Fx	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Min Fx	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Max Fy	5	102 CASE 1 +	-267.41E3	9209.981	42301.568	-150.203	-163.894	-12.020
Min Fy	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Max Fz	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Min Fz	10	3 TEMPERAT	339.42E3	-7766.295	-367.75E3	23.440	32.043	-3.983
Max Mx	10	102 CASE 1 +	339.42E3	-3669.513	-367.75E3	38.652	32.043	32.681
Min Mx	2	102 CASE 1 +	-208.18E3	7339.049	177.2E3	-171.568	361.463	28.194
Max My	2	3 TEMPERAT	-208.18E3	5522.880	177.2E3	-148.111	361.463	28.141
Min My	7	3 TEMPERAT	-332.6E3	-7599.259	237.67E3	24.478	-261.486	-20.956
Max Mz	4	102 CASE 1 +	295.97E3	5646.186	-241.91E3	9.895	-38.685	138.344
Min Mz	7	102 CASE 1 +	-332.6E3	-3821.673	237.67E3	-20.462	-261.486	-37.400

Figure 3. 154

This brings us to the conclusion of this tutorial. Additional help on using plates is available in Examples 9, 10 and 18 in the Examples Manual.

3-144 Tutorial 3			
Ν			
0			
t			
е			
S			

Tutorial Problem 4: Interoperability (using STAAD.Pro and STAAD.etc)

- Section **4**

This tutorial is relevant only for users who have purchased the program called STAAD.etc. Since this tutorial is created using the Demo CD, the user will not have the ability to alter certain input items.

This chapter provides a step-by-step tutorial on using the interoperability features between STAAD.Pro and the program called STAAD.etc. The structure created in Tutorial Problem 1 will be used as the basis for this tutorial. This tutorial covers the following topics:

- Understanding STAAD.etc
- Description of the tutorial problem
- Using the Interactive Mode in STAAD.Pro
- Designing a footing based on results from STAAD.Pro
- Designing a base plate based on results from STAAD.Pro
- Saving the Interactive Design information as a STAAD.etc file

4.1 Understanding STAAD.etc

STAAD.etc is the "engineer's structural toolkit" containing various component analysis and design module groups including foundations, masonry, timber, steel, concrete and general analysis. As a standalone product, STAAD.etc can be used to design small portal frames, continuous beams, footings, retaining walls one-way slabs, connections, shear walls and much more. STAAD.etc allows the engineer to complete the design cycle on the primary structure by analyzing and designing several structural components or accessories within the STAAD.Pro environment using STAAD.Pro's results database.

4.2 Description of the tutorial problem

Using the structure created in Tutorial Problem 1 as the primary model, STAAD.etc will be utilized to analyze and design a footing and a base plate inside the STAAD.Pro environment for the model.

4-4 Tutorial 4

4.3 Using the Interactive Mode in STAAD.Pro

Start the STAAD.Pro 2003 Program from your STAAD.Pro program folder, as explained in Section 1.2. From **File** | **Open**, open the file "Tut_01_portal.std" from the

../SPRO2003/STAAD/Examp/US directory as shown below (please note that the .. represents the drive or root directory the program was installed in) or from wherever the file was saved:

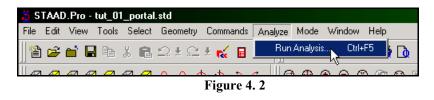
Open				? ×
Look in: 🔂	US		- 🗢 🖻	
 a) EXAMP19.9 b) EXAMP20.9 c) EXAMP21.9 c) EXAMP22.9 c) EXAMP23.9 c) EXAMP24.9 	STD 🔹 EXAN STD 🔹 EXAN STD 🔹 INTR STD 🔹 Tut (4P25.STD 4P26.STD 4P27.STD 0.STD 11_portal.std 12_roframe.std	VER01.ST VER02.ST VER03.ST VER04.ST VER05.ST VER06.ST	D BVERC D BVERC D BVER1 D BVER1
		*.std)		Open Cancel
Preview		Јор	🔽 Verify list da	ita while reading
		Job Client Job No. Part Ref		

Figure 4.1

The picture in the preview pane is made possible by a file with the extension .emf. When a model is created using the graphical method, or when a model is saved from the graphical mode, the .emf file is created alongside the .std file. If you follow the procedure explained in Section 1.5 of this manual for creating tutorial 1, the Tut-01-portal.emf file will be produced, and if you re-open the Tut-01-portal.std subsequently, you will see the picture in the preview pane.

If you have not performed the analysis and design for "Tut_01_portal.std", please execute the following steps:

1. After opening the file in STAAD.Pro, go to **Analyze** | **Run Analysis** from the top menu bar as shown below.



2. A dialog box providing a choice of engines to run the file with will then popup. Select the **STAAD Analysis** engine as the default engine and click on the **Run Analysis** button.

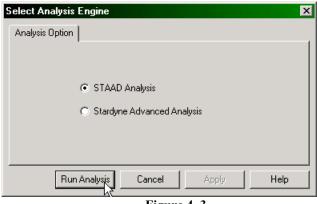


Figure 4. 3

3. If the analysis is successfully completed, a dialog box showing the different steps STAAD took to analyze the structure will come up. Click **Done** to proceed to the next step. If a message reporting errors in the input file were found, please refer to Tutorial 1 to rectify those problems.

🚡 STAAD Analysis and Design	×
<pre>++ Performing Steel Design ++ Start Steel Design ++ Finished Design ++ Finished Design ++ Processing all setting up Load Vector. ++ Processing Global Stiffness Matrix. ++ Processing Global Stiffness Matrix. ++ Finished Processing Global Stiffness Matrix. ++ Finished Processing Global Stiffness Matrix. ++ Finished Triangular Factorization. ++ Galculating Joint Displacements. ++ Finished Joint Displacement Calculation. ++ Calculating Member Forces. ++ Analysis Successfully Completed ++ ++ Calculating Section Forces ++ Performing Steel Design ++ Start Steel Design ++ Finished Design ++ Finished Design ++ Creating Reaction File (DSP) ++ Creating Section Forces ++ Creating Section Displace File (SCN) ++ Creating Design information File (DGN) +* End STAAD.Pro Run Elapsed Time = 3 Secs</pre>	10:11:41 0 sec 10:11:41 10:11:41 10:11:41 1 sec 10:11:42 0 sec 10:11:42 10:11:42 0 sec
** Output Written to File: PORTAL.anl	•
C View Output File C Go to Post Processing Mode Stay in Modelling Mode Execute	Done

Figure 4. 4

Starting the Interactive Mode in STAAD.Pro

After "tut_01_portal.std" has been successfully analyzed and designed, click on **Mode** | **Interactive Designs** | **Component Design** from the top menu bar as shown in the next figure. If this menu item is "grayed-out", the STAAD.etc program has not been installed properly or is missing. Please install or reinstall the STAAD.etc program from the STAAD.suite CD.



4-7

Tutorial 4

Figure 4.5

The STAAD.etc interface to STAAD.Pro is shown in the bottom right-hand corner of the screen. At the time of release of STAAD.Pro 2003 Build 1001, there are only three STAAD.etc modules interfaced with STAAD.Pro 2003. These modules are the footing, slab and base plate analysis/design as shown below.

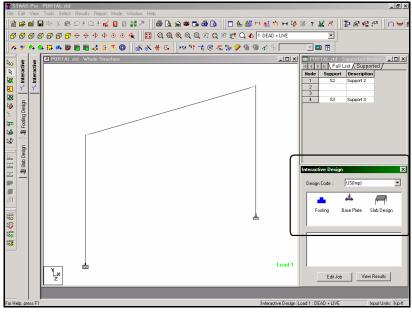


Figure 4.6

Using the Drag and Drop Facilities

To run any of the STAAD.etc modules, hold the left-mouse button down over a particular module and drag it to a specific joint or node in the main structure window.

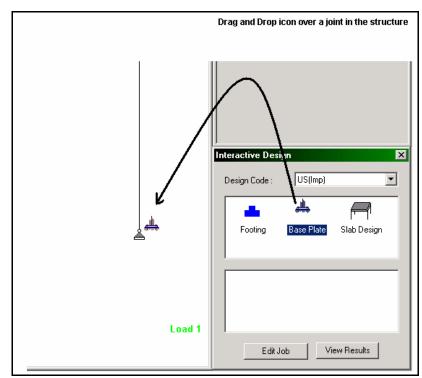


Figure 4. 7

4.4 Designing a Footing based on results from STAAD.Pro

In this section of the tutorial, a footing will be designed for node #1. The details of the input of the footing will not be discussed here. For further information about the input for the STAAD.etc footing module, please refer to the STAAD.etc HTML Help accompanied with the STAAD.etc program.

To identify Node #1, click on *Select* | *By List* | *Nodes*... from the main menu bar. Select number 1 in the list box.

Sele	act.			
	<u>N</u> odes Cursor		£	
			L .	
	Beams Cursor		L .	
	Physical Member Cursor		L .	
	Plates Cursor		L .	
	Surface Cursor		L .	
3	Solids Cursor		L .	
	Plates & Solids Cursor		L .	
₽ ≱	Geometry Cursor			
	Selection Mode	►		
व्रिय	Load Edit Cursor			
1	Support Edit Cursor		L .	
₩/	Release Edit Cursor			
A	Text Cursor			
	Beams Parallel To	►		
	By <u>R</u> ange	►	L .	
	By <u>G</u> roup Name		L .	
	Entity at node	►		
	Entity Connects Beam	≁	L .	
	Entity Connects Plate	×	L .	
	Entity Connects Solid	•		
	By All	►		
	By Inverse	►		
	By List	►		Nodes 📐
	By Specification	►		Beams
	By Missing Attributes	•		Plates
	Highlight Entities Sequentially			Solids Geometry
		_		a somouy

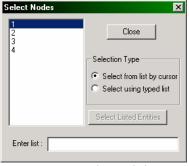
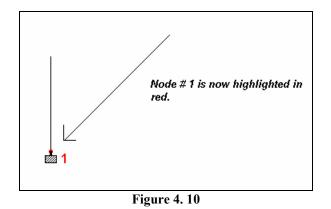


Figure 4.9

Figure 4.8

4-10 Tutorial 4

Node #1 is now marked by a red dot in the main structure window as shown below.



Once Node #1 is identified, drag the footing icon from the Interactive Design dialog box and drop it over #1. The input for the footing pops up. Provide the name of the footing in the **Job Name** edit box.

Tutorial 4 4-11

Staad.etc Interactive Design			×
Service Load Factors	Material Data	Reinforcement D	ata
Geometry Pedestal	Loading	Design Load Fact	tors
Node: 1			
Job Name : abcdefg	\leftarrow		Important to fill this information
🔲 Concrete Pedestal	Code : US	•	
Footing Min-Max			
Minimum Width :	72	in 💌	
Maximum Width :	720	in 💌	
Minimum Length :	72	in 💌	
Maximum Length :	720	in 💌	
Ratio :	1.5		
Minimum Footing Thickness	24	in 💌	
Column Data			
Column Depth:	12	in 💌	
Column Width:	12	in 💌	
- Soil Depth-			
Soil Depth :	24	in 💌	
- Iteration Increment			
Increment in Length	12	in 💌	
Increment in Thickness	1	in 💌	
	D *	umo 1 11	

Figure 4. 11

If the *Column Depth* and *Width* are not provided under the *Column Data* box, obtain those data (either from the member query or from the property table) and provide them. STAAD.etc defines the Column Depth as the dimension of the supported column along the local Y axis of the column. The Column Width is defined as the dimension of the supported column along the local Z axis of the column.

Leave the *Concrete Pedestal* option **unchecked**. Switch to the *Loading* tab and select the *Dead Load* as **Load Case 1** (the Selfweight load case) and the *Live Load* and *Wind Load* as **None**. Check the *Reversal of Wind* option off as shown below.

Staad.etc Interactiv	ve Design			X
Service Load Fac	ctors	Material Data Loading	1	ement Data
Geometry Select Load C Dead Load :		n for	Uesign L	oad Factors
Live Load :	None		•	
Wind Load :	None		•	
☐ Re	eversal of Win	d		
	ок	Cancel	Apply	Help

Figure 4. 12

Keep all the other data as the defaults. Click on **OK** to begin designing the footing for Node #1. If the design completes successfully, the results for the footing will be displayed in a separate window. Each interactive design module produces a separate results window containing three tabs at the bottom. These tabs are *Results, Calculations and Output Drawing*.

The different facets of the	results output are as	follows:
-----------------------------	-----------------------	----------

Tab	Description	Sample
Results	Displays all results (analysis and design) in tabulated form	Setting - Support No. J Geometric Output Geometric Output Case Setting Seti
Calculations	Displays the step-by-step intermediate calculations explaining the derivation of all results.	I OL_01_portAlstd = froendation Design = Support No. 1: Trial Length, L Trial Vength, L Trial Width, W Check Trial Depth against Punching Shear Strength, V: BReteiro Depth, 4 ~ D - Cime - 1.0 Por rectangular column, b= BarDu = 12.000 / 12.000 Increase durinil 0.85° V.2 all asial design leads From ACT Bg. J1-35, %.* (2 + 40b.)*bx*da*/1000°F? = (2 + 401 000)*122.000*21.000*√4000.000 Prom ACT Bg. J1-37, %.* 4*bbx*da*/1000°F? = (2 + 40*21.000*132.000*132.000*21.000*√4000.000
Output Drawing	Detailed output drawing schematic	

Figure 4. 13

4-14 Tutorial 4

Editing and Viewing Interactive Designs

All interactive design results for a particular STAAD file are listed in the Interactive Design dialog box which can be brought up by clicking on *Mode* | *Interactive Design* from the top menu bar. To view any interactive design result, select the design job and click on *View Results*. To edit any job, select the job and click on *Edit Job* to bring up the input dialog box with the existing data.

Interactive Des	ign		x
Design Code :	US(Imp)	•]
-	#	r f f f f f f f f f f f f f f f f f f f	
Footing	Base Plate	Slab Design	
Foundation US	demo		
Edit J	lob Vi	ew Results	
	Figure 4. 1	4	

STAAD.Pro also has the ability to design group footings. This tutorial does not cover the basics of running a group footing inside of STAAD.Pro.

4.5 Designing a base plate based on results from STAAD.Pro

The base plate and other modules in the Interactive Design mode of STAAD.Pro are similar to the footing module explained previously. A brief description of the base plate input will be explained here. For a detailed description of the base plate module, please refer to the STAAD.etc HTML Help.

Following the steps in the footing module tutorial in Section 4.4, the input for a base plate can be supplied after the base plate icon is dragged and dropped over a particular support joint. A base plate can be designed only for a member with a support.

Staad.Etc Interactive Design	X			
Geometry Data Stiffeners Data Anchor Bolt Data Sh	Geometry Data Stiffeners Data Anchor Bolt Data Shear Lug Data			
Job: Test Job: Test Design code: ASD Job: Test Geometry Min. Plate Length: 14 In Max. Plate Length: 120 In Max. Plate Length: 120 In Min. Plate Width: 120 In Min. Plate Width: 120 In Min. Plate Width: 120 In Plate Thickness: 0.5 In Plate Thickness: 0.5 In Plate Thickness: 0.5 In Plate Thickness: 0.5 In Plate Width: 28 In Plate Width: 28	Select Load Case to Design for Load: IDEAD + LIVE ¥ Bolt Distances Bolt along Length : 2 D1: 55 n ¥ D3: 55 n ¥ D4: 55 n ¥ D4: 55 n ¥			
OK Cancel	Apply Help			

Figure 4.15

To check the adequacy of an existing base plate for a specific load case, provide the Plate Length, Plate Width and Thickness of that plate. However, to have the program design a base plate, set the Plate Length, Plate Width and Thickness to zero (0).

4-16 Tutorial 4

4.6 Saving the interactive design as a STAAD.etc file

All interactive design jobs are saved with the STAAD.Pro ".std" input file as a separate file with the extension ".etc". For example, in this tutorial, the STAAD file is named $tut_01_portal.std$. Thus, the name of the interactive design file will be $tut_01_portal.etc$ and will be saved in the same directory as the " $tut_01_portal.std$ " file.

STAAD.Pro is capable of inherently reading STAAD.etc files without the need for importing or exporting the file. Similarly, any ".etc" file generated by STAAD.Pro can be read in STAAD.etc. This is particularly useful when the engineer wants to design other structural components on top of the ones already produced in STAAD.Pro.

4-17

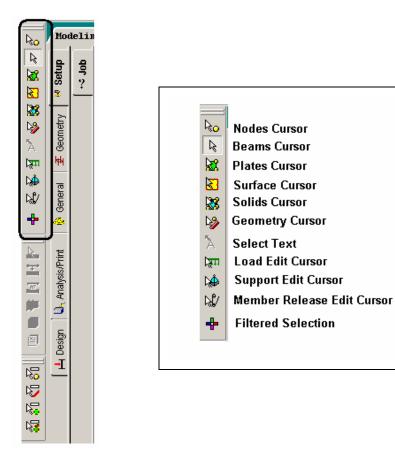
N o t e s

4-18

N o t e s **Frequently Performed Tasks**

1. Selecting nodes, beams, plates, etc.

The Selection Toolbar



lcon	Corresponding Menu/Sub-menu options	Purpose	Description
Nodes Cursor	Select Nodes Cursor Select Geometry Commands / Nodes Cursor Beams Cursor Physical Member Cursor Physical Member Cursor Surface Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Selection Mode Comment Cursor Support Edit Cursor Support Edit Cursor Plates Edit Cursor Comment Cursor	Used to select nodes graphically	First, select the Nodes Cursor. Then, click on the nodes you wish to select. To select multiple nodes, hold down the Control key while selecting, or create a rubber-band style window around the desired nodes.
Beams Cursor	Select Beams Cursor Select Geometry Commands / Nodes Cursor Beams Cursor Physical Member Cursor Surface Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Plates & Solids Cursor Selection Mode Time Load Edit Cursor Support Edit Cursor Compared to Cursor	Used to select beams graphically	First, select the Beams Cursor. Then, click on the members you wish to select. To select multiple members, hold down the Control key while selecting, or create a rubber-band style window around the desired beams.

Frequently Performed Tasks

FPT-3	5
-------	---

lcon	Corresponding Menu/Sub-menu options	Purpose	Description
Plates Cursor	Select Plates Cursor Select Geometry Commands / Nodes Cursor Beams Cursor Physical Member Cursor Plates Cursor Solids Cursor Solids Cursor Selection Mode Load Edit Cursor Support Edit Cursor Versor Load Edit Cursor Versor Text Cursor	Used to select plates graphically	First, select the Plates Cursor. Then, click on the plates you wish to select. To select multiple plates, hold down the Control key while selecting, or create a rubber-band style window around the desired plates.
Surface Cursor	Select Surface Cursor Select Geometry Commands Nodes Cursor Beams Cursor Physical Member Cursor Surface Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Selection Mode Commetry Cursor Selection Mode Commetry Cursor Support Edit Cursor Support Edit Cursor Commetry Cursor	Used to select surface graphically	First, select the Surface Cursor. Then, click on the surface you wish to select. To select multiple surfaces, hold down the Control key while selecting, or create a rubber-band style window around the desired surfaces.

lcon	Corresponding Menu/Sub-menu options	Purpose	Description
Solids Cursor	Select Solids Cursor Select Geometry Commands / No Nodes Cursor Beams Cursor Physical Member Cursor Plates Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Plates & Solids Cursor Selection Mode The Load Edit Cursor Support Edit Cursor Support Edit Cursor Plates E Edit Cursor Comments Cursor Comments Cursor Comments Cursor Support Edit Cursor Comments Cursor Comme	Used to select solids graphically	First, select the Solids Cursor. Then, click on the solids you wish to select. To select multiple solids, hold down the Control key while selecting, or create a rubber-band style window around the desired solids.
Geometry Cursor	Select Geometry Cursor Select Geometry Commands No Nodes Cursor Beams Cursor Physical Member Cursor Physical Member Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Selection Mode Commetry Cursor Selection Mode Commetry Cursor Selection Mode Commetry Cursor Support Edit Cursor Plates Edit Cursor Commetry Cursor	Used to select any geometry graphically. It is a mechanism for selecting nodes, beams, plates and solids, or, any combination of these, simultaneously.	First, select the Geometry Cursor. Then, click on the entity you wish to select. To select multiple entities, hold down the Control key while selecting, or create a rubber-band style window around the desired entities.

FPT-4 Frequently Performed Tasks

Frequently Performed Tasks

Icon	Corresponding Menu/Sub-menu options	Purpose	Description
Load Edit Cursor	Select Load Edit Cursor Select Geometry Commands / Nodes Cursor Beams Cursor Physical Member Cursor Physical Member Cursor Surface Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Selection Mode Coad Edit Cursor Support Edit Cursor Release Edit Cursor Text Cursor	Used to graphically select a component of an existing load case for modification.	First, select the Load Edit Cursor. Then, double-click on the diagram of the load component you wish to edit.
Support Edit Cursor	Select Support Edit Cursor Select Geometry Commands Solution Solution Soluti	Used to graphically select a support whose parameters one may wish to modify.	First select the Support Edit Cursor. Then, double-click on the support icon at the node where you wish to modify the details of the existing support.

FPT-5

lcon	Corresponding Menu/Sub-menu options	Purpose	Description
Release Edit Cursor	Select Release Edit Cursor Select Geometry Commands No Nodes Cursor Beams Cursor Physical Member Cursor Surface Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Selection Mode Comment Cursor Selection Mode Comment Cursor Support Edit Cursor Support Edit Cursor Carson	Used to graphically select and modify an existing member release specification	First, select the Release Edit Cursor. Then, double-click on the member on which a currently defined START or END release condition has to be modified.
Select Text	Select Text Cursor Select Geometry Commands / No Nodes Cursor Beams Cursor Physical Member Cursor Physical Member Cursor Surface Cursor Solids Cursor Plates & Solids Cursor Plates & Solids Cursor Selection Mode Carton Cursor Selection Mode Carton Cursor Support Edit Cursor Plates Edit Cursor Carton Cursor	Used to enter the mode for editing pre- created text labels	To edit any pre- created text, first, select the Text Label Cursor. Then, double-click on the text that you wish to modify.

FPT-6 Frequently Performed Tasks

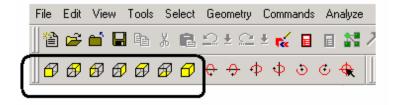
Frequently Performed Tasks

lcon	Corresponding Menu/Sub-menu options	Purpose	Description
Filtered Selection	Select Geometry Commands Analyze I Image: Ima	Enables the user to select multiple types of geometric entities (nodes, beams, surfaces, etc.) with specific attributes in one pass.	Please refer to STAAD.Pro 2004 2 nd Edition Release Report.

FPT-7

2. Viewing the structure from different angles

The Rotation Toolbar





lcon	Description	Example
View From + Z	Displays the structure as seen from the front. When the global Y axis is vertical, this is the elevation view, as looking towards the negative direction of the Z-axis.	
View From - Z	Displays the structure as seen from the back. When the global Y axis is vertical, this is the elevation view, as seen looking towards the positive direction of the Z-axis.	
View From + X	Displays the structure as seen from the right side. When the global Y axis is vertical, this is the side elevation, as seen looking towards the negative direction of the X-axis.	

FPT-10 Frequently Performed Tasks

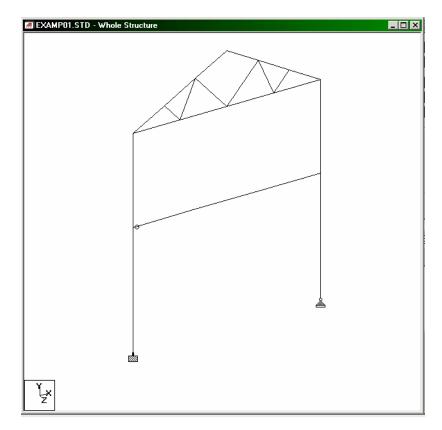
lcon	Description	Example
View From - X	Displays the structure as seen from the left side. When the global Y axis is vertical, this is the side elevation, as seen looking towards the positive direction of the X-axis.	
View From + Y	Displays the structure as seen from the top looking down. When the global Y axis is vertical, this is the plan view, as seen from the sky looking down.	
View From - Y	Displays the structure as if one is looking up skywards.	

Frequently Performed Tasks FPT-11

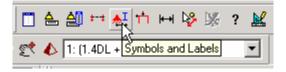
lcon	Description	Example
Isometric View	Displays the structure in the isometric view. The angle which defines isometric view is generally X = 30, Y = 30, Z = 0	

3. Switching on labels for nodes, beams, plates, etc.

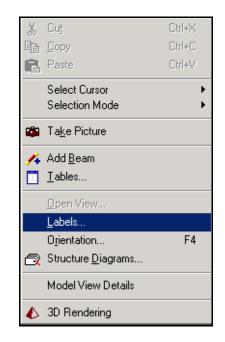
Labels are a way of identifying the entities we have drawn on the screen. To demonstrate this facility, let us open EXAMP01.std. The structure will look as shown below.



- 1. To switch the node and beam labels on, we may utilize any one of the following methods:
 - a. Click on the Symbols and Labels icon as shown below.



b. Click the right mouse button anywhere in the drawing area. In the dialog box that appears, choose **Labels** (as shown in the figure below).



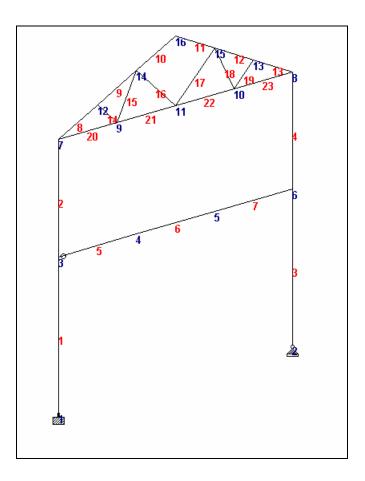
c. Select the **View** menu followed by the **Structure Diagrams** option from the top menu bar, and the **Labels** tab of the dialog box that comes up.



2. In all of the above three cases, the following *Diagrams* dialog box will appear. Make sure that the **Labels** tab is selected. Then, switch the appropriate labels on and click on **OK**.

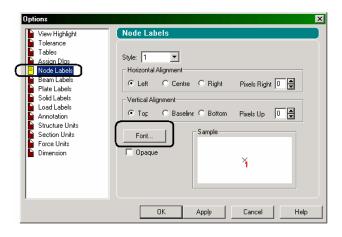
Diagram s			×
Force Limits Structure	Animation	Desi Scales	gn Results Labels
Nodes Node Numbers (N) Node Points (K) Supports (S) Dimension (D) Beams Beam Numbers (B) Beam Orientation (0) Beam Spec (A) Releases (R) Beam Ends (E) Start Color End Color Plates Plate Numbers (P) Plate Orientation (T)	Display Wind Load General Show Axes Window Show Axes At Org (Material (M) Show Diagram Info Surfaces	ling (F) I Distribution (Y) I Contributory Area (H I (W) U (^A) I Master S I)	nber (^M) p (^G) elope (^E) (^B) H) Slave (L)
Solids Solid Numbers (C)	Always Use STAA Always Use Currer Always Use Currer ode (Use labeling cursors	D.Pro Default Label It Label Settings. to ON/OFF individua	Settings.
	ccess to the labels using k s Shift + the letter shown i		
	OK Canc	el Apply	Help

The following figure shows the structure with the node and beam labels displayed on it.



3. To change the font of the node/beam labels, go to the **View** menu and select the **Options** command. Then, select the appropriate tab (Node Labels / Beam labels) from the **Options** dialog box. Choose the **Font** option and make the necessary changes.

View		
Z	oom	•
٢ الح	an	
V	iew Selected Objects Only	
✓ A	lways <u>F</u> it in Current Window	
Ī	ables	
	/hole Structure	
0	gientation	F4
🗟 S	tructure Diagrams	
0	pen View	
<u>N</u>	ew View	
Vi	ïew <u>M</u> anagement	•
T	ool <u>b</u> ars	
0	Iption <u>s</u>	
S	et Structure <u>C</u> olors ^{KS}	
	tructural Tool Tip Options	
S	tructural I ool Tip Uptions D Rendering	



4. <u>Displaying a portion of the model by isolating it from</u> <u>the rest of the structure</u>

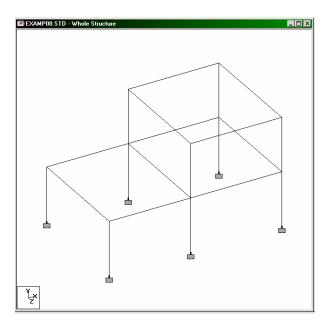
Sometimes, the large number of entities that are drawn on the screen may make it difficult to clearly see the details at any particular region of the structure. In such cases, one is confronted with the task of de-cluttering the screen or looking at specific regions or entities while removing the rest of the structure from the view.

There are different methods in STAAD.Pro by which the user can view a portion of the structure. The following pages demonstrate three different methods.

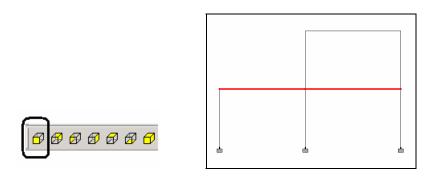
Method 1 - View | New View

Steps:

1. To demonstrate this method, let us open EXAMP08.STD file. The structure will look as shown below.

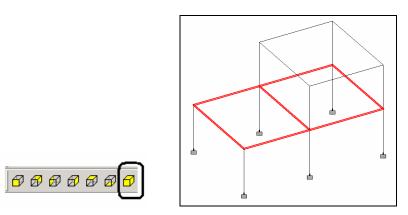


Let us say that we wish to view only the members which are at the midheight level. The first step to doing this is to select those members. The quickest way to select them is to bring up the View
 From + Z and creating a rubber-band around them, after making sure that the *Beams Cursor* is the active cursor.



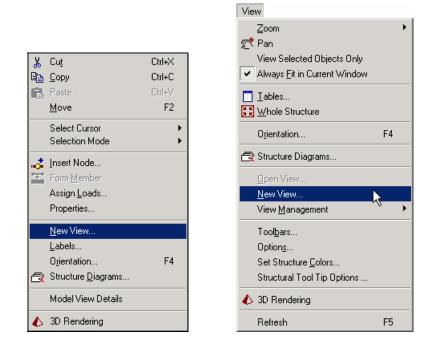
3. To visually verify that the correct members have been selected,

click on the **Isometric View** icon \square . The selected members will appear in one color and the rest of the structure in another.



FPT-20 Frequently Performed Tasks

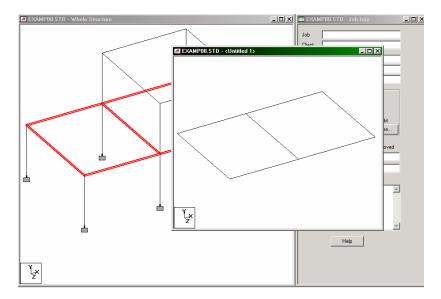
4. Next, either click the right mouse button and select the **New View** option or, pull down the *View* menu and select the *New View* option.



5. In either case, the following dialog box comes up. These radio buttons determine whether the selected view would be opened in a new ('child') window or whether it would replace the current ('parent') view window. Let us select the **Create a new window for the view** option and click on the **OK** button.



The portion of the structure that we selected will now be displayed in a new window as shown below. We can create more than one 'child' view window this way.



All sorts of operations can be performed inside the 'New View' such as adding and deleting members, assigning properties, loads, supports, etc. A new view of a selected portion offers the advantage of de-cluttering the screen and limiting the displayed objects to just a few chosen entities.

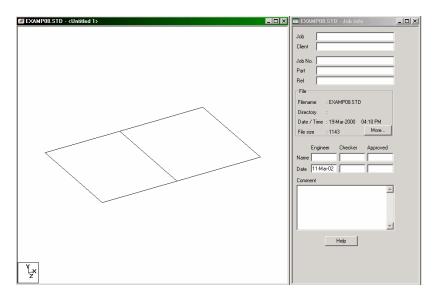
6. To return to the 'parent' view window, simply close the new view ('child') window.

FPT-22 Frequently Performed Tasks

7. Next, let us try the *Display the view in the active window* option. Follow steps 2 to 4. Then, select the **Display the view in the active window** option and click on the **OK** button.



The original structure will be hidden and, the portion of the structure that we selected will instead be displayed in the current ('parent') window as shown below.



We may restore the original view of the structure by simply clicking on the **Display Whole Structure** icon.

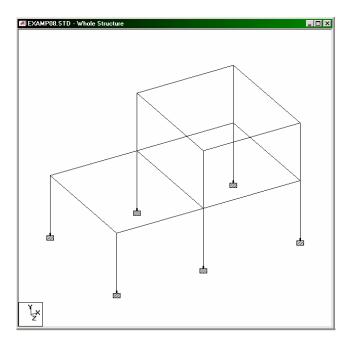


These new views may be saved by going to the *View* | *View* Management | Save View menu option. Provide a title for the new view. These saved views may later be opened by going to the View | Open View menu option.

Method 2 - Tools | Cut Section

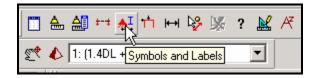
Steps:

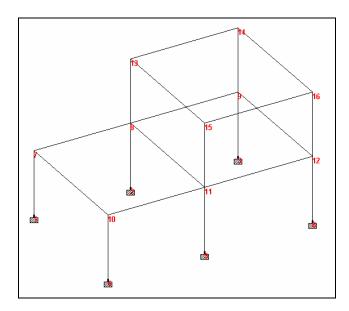
1. To demonstrate this method, let us once again open EXAMP08.STD file. The structure will look as shown below.



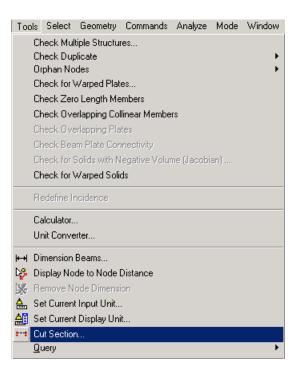
We will once again try to view just the members at the mid-height level.

2. Click on the **Symbols and Labels** icon and in the *Diagrams* dialog box that comes up, switch the **Node Numbers** on. The reason for doing this will become evident in the next step.





3. From the Tools menu, select Cut Section.



The following dialog box comes up.

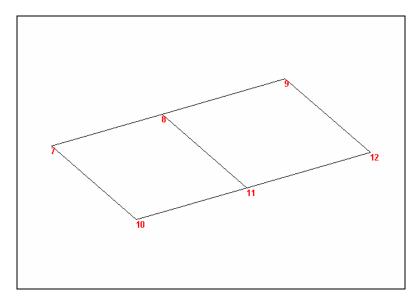
Section X
Range By Joint Range By Min/Max Select to View
© X - Y Plane With Node # © Y - Z Plane
C X - Z Plane
OK Cancel Show All Help

The Range By Joint method

Since the beams of the mid-height level happen to contain joint # 10 (or any of the joint numbers from 7 to 12), it would be convenient if we could instruct the program to display the entities lying in the X-Z plane, and passing through any one of those joints 7 to 12. The facilities of the *Range By Joint* tab enable us to do that.

4. Let us select that tab (it happens to be the default). Here, we can select the plane of the section by clicking on one of the options – X-Y, Y-Z, or X-Z. We have to choose the X-Z plane for our example. Then, in the *Node* # drop down list, provide a joint that lies on the sectional plane. In our example, we may choose any one of the nodes between 7 and 12. Let us choose **node** # 10. Click on the **OK** button.

The following figure shows the cut portion of the original structure displayed in the drawing area.



5. To restore the original view, simply click on the **Display Whole Structure** icon.

🙀 ହେଇ କେର୍ଷ 🖉 😡 🕅
🕂 🗹 🕂 🖓 Display Whole Structure

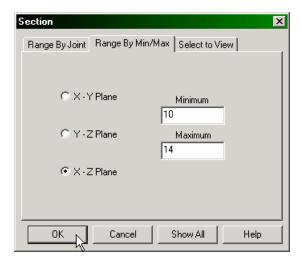
Alternatively, go to the dialog box shown in step 4 and click on the *Show All* button.

The Range By Min/Max method

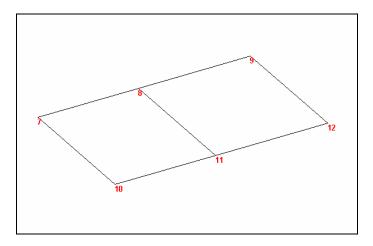
Steps:

Another convenient way of choosing the beams at the mid-height level is to specify that those beams lie in the X-Z plane between the Y range values of 12ft - a and 12ft + a, where, "a" is an arbitrary value such as say, 2ft.

6. To do this, let us select the Range By Min/Max tab. Here, we can again select the plane of the section by clicking on one of the options – X-Y, Y-Z, or X-Z. We have to choose the X-Z plane for our example. The *Minimum* and *Maximum* edit boxes represent the boundary distances along the axis perpendicular to the sectional plane. Every object lying between these two distances will be displayed. In our example, let us provide 10 as the *Minimum* and 14 as the *Maximum* distance. (Before doing so, make sure that the current input units of length, displayed at the bottom right corner of the STAAD program window, are feet.) Then, click on the OK button.



The following figure shows the cut portion of the original structure displayed in the drawing area.



7. To restore the original view, again, click on the **Display Whole Structure** icon or click on the *Show All* button in the dialog box shown in step 6.

👯 ଉଇ୍ଭ୍ର୍ର୍	QQQ
, ≺Display Whole Structure	MX PM MZ

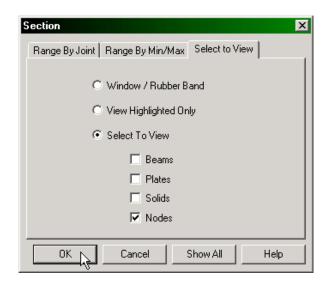
The Select to View method

For demonstrating this method, instead of viewing specific beams, let us look at the nodes instead.

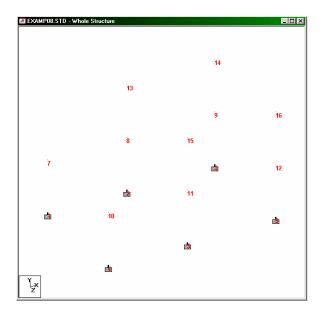
Steps:

Using this option, the portion of the structure we wish to view can be selected by specifying the objects which lie in that portion. The *Window/Rubber Band* option allows us to select the portion of the structure to view by specifying a rubber-band window around it. The *View Highlighted Only* option displays only the selected (highlighted) objects on screen and, the members and elements we wish to view must be selected before choosing this option. The *Select To View* option allows us to view only Beams, Plates, and/or Solids, depending on the corresponding check boxes that are activated.

8. For our example, let us choose the **Select To View** option and check the **Nodes** check box. Then, click on the **OK** button as shown below.



The following figure shows the view of the structure with only the nodes displayed.



9. To restore the original view, click on the **Display Whole Structure** icon or click on the *Show All* button in the dialog box shown in step 8.

🖳 ହେଇ୍ର୍ର	Q (R ()
Display Whole Structure	$(\underline{\mathbb{M}}^d_{X}$	$\mathbb{F}_{M_{\mathrm{N}}}$	Mz MD

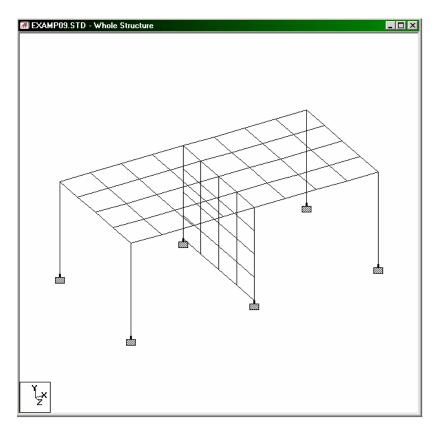
These new views may be saved by going to the *View* | *View* Management | Save View menu option. Provide a title for the new view. These saved views may later be opened by going to the *View* | Open View menu option.

Method 3 – View | Zoom

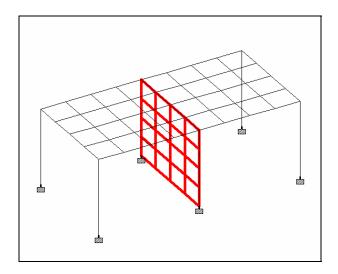
The *Zoom* menu option allows us to zoom in (magnify) or out (reduce) on a portion of the structure, as one would, using a telescope.

Steps:

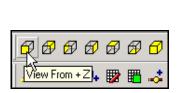
1. To demonstrate this method, let us open EXAMP09.STD file. The structure will look as shown below.

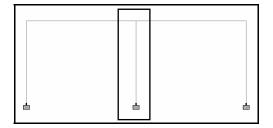


2. Say, we want to magnify the portion of the structure shown in the figure below.



To select the plates as shown in the above figure, click on the View From + Z icon. Then, using the *Plates Cursor* , select the plates by rubber-banding around them as shown below.





By the way, looking at the model *from View From* + Y, or, *View From* - X, and using the rubber-band window methods can be just as effective in selecting these elements.

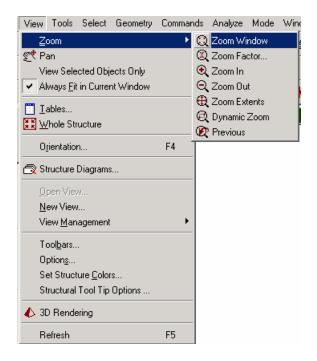
FPT-34 Frequently Performed Tasks

4. To restore the original isometric view, click on the **Isometric View** icon. We do this only to make sure that we have selected the correct entities.



 To magnify the selected portion of the structure, either click on the Zoom Window icon or, pull down the View menu and select the Zoom | Zoom Window menu option as shown below.



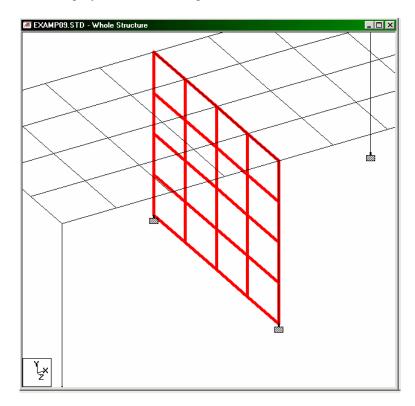


The cursor changes as shown below.

6. Using the cursor, drag a window around the selected portion of the structure.

+

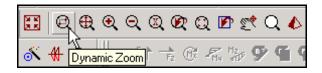
The magnified view of the selected portion of the structure will now be displayed in the drawing area as shown below.



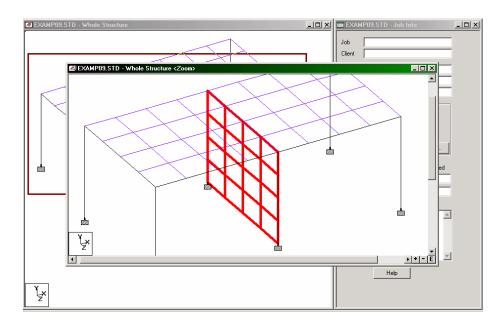
If we click on the *Zoom In* icon, it magnifies or enlarges the view of the structure even further by a pre-defined factor. You can keep the mouse cursor pressed over the Zoom In or Zoom Out icons, and the screen will continuously refresh to draw the structure larger or smaller.



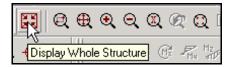
To display the enlarged view in a new window, use the *Dynamic Zoom* icon instead of the *Zoom Window* option.



The region you wish to see will have to be selected by creating a rubber-band window around it. That region will now be displayed in a new window in the drawing area as shown in the next figure.



7. To restore the view of the full structure, click on the *Display Whole Structure* icon.



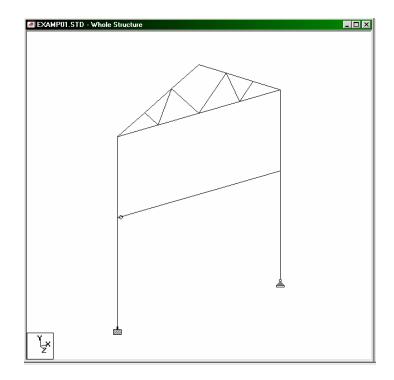
These new views may be saved by going to the *View* | *View* Management | Save View menu option. Provide a title for the new view. These saved views may later be opened by going to the *View* | Open View menu option.

5. Creating Groups

Group names are a means for easily identifying a collection of entities like Beams, Plates or Solids using a single moniker. By grouping these entities, we need to assign attributes such as member properties and material constants just to the group, a simple process, compared to the task of assigning them to the individual members.

Steps:

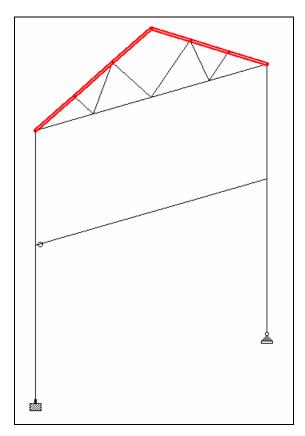
1. To demonstrate this, let us open EXAMP01.STD file. The structure will look as shown below.



Let us classify the members of the topmost level of this structure into three groups – Top Chords, Bottom Chords and Transverse Truss.

Top Chords:

2. Make sure that the *Beams Cursor* has been selected. Then, using the mouse, select the inclined members as shown in the figure below.



3. From the Tools menu, choose Create New Group.

Too	ols	Select	Geometry	Commands	Analyze	Mode	Window
	Cł	neck Mul	tiple Structu	res			
		neck Dup					•
		phan No					•
			Warped Plat				
			o Length Me				
				llinear Membe	rs		
			erlapping Pla				
			im Plate Cor				
				legative Volur	ne (Jacobi	anj	
	Ur	IECK FOR V	Warped Soli	OS			
	Re	edefine Ir	ncidence				
	Ca	alculator					
	Ur	nit Conve	arter				
₩	Di	mension	Beams				
₽\$	Di	splay No	de to Node I	Distance			
%	Re	emove N	ode Dimens	ion			
촖	Se	et Curreni	t Input Unit				
			t Display Uni	it			
‡-‡		it Section	n				
	Qu	lery					•
	Lis	at Format					
	Cr	eate Use	er Table				
	Se	ection Wi	izard				
	M	odify Sec	tion Databa	se			
	Cr	eate <u>N</u> ev	v Group				Ctrl+G
Æ	In	sert Text					

4. In the *Give Group Name* dialog box, type **_TOPCOR** for *Group name* (Group Names must begin with the underscore '_' character). Since group names can be assigned to Nodes, Beams, Plates, Solids as well as a general category called Geometry, it is very important that we choose the proper entity type. Hence, specify the *Select Type* as **Beam**.

Give Group Name		×
Group Name :	_TOPCOR	_
Select Type :	Beam	•
ОК	Cancel	Help

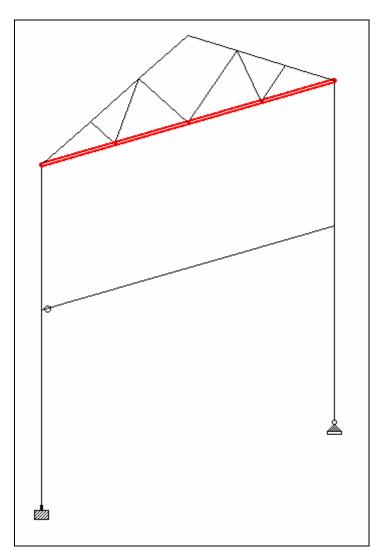
5. Click on the **OK** button. Notice that the *Assign Methods* in the *Create Group* dialog box is presently set to *Associate to Selected Geometry*. Click on the **Associate** button.

Create Group	×
1 : _TOPCOR (Beam)	Create
	Delete
	Highlight
	Close
Assign methods C Associate to View C Associate to Selected G	eometry
C Associate to List List 8 To 13	
	Help

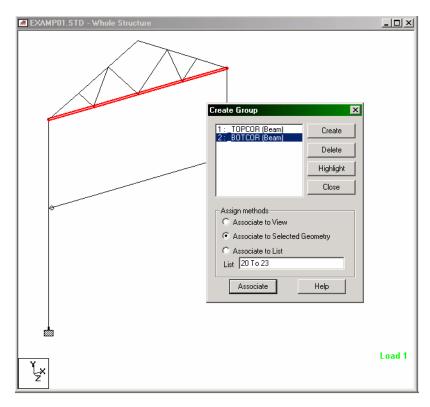
Click anywhere in the drawing area to un-highlight the highlighted members. This is necessary before we start selecting the members for the next group name.

Bottom Chords:

6. Using the mouse, select the bottom chords as shown in the figure below.



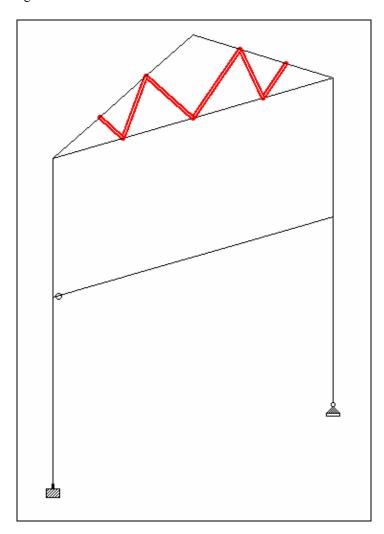
7. Next, to bring up the *Give Group Name* dialog box, click on the **Create** button. Then, follow steps 4 and 5 and assign the *Group* Name _BOTCOR to these members.



Click anywhere in the drawing area to un-highlight the highlighted members.

Transverse Truss:

8. Using the mouse, select the Transverse Truss members as shown in the figure below.



9. Next, follow step 7 except for specifying the *Group Name* as _TRNTRUS.

RXAMP01.STD - Whole Structure		_ 🗆 ×
	Create Group	
	1: TOPCOR (Beam) 2: BOTCOR (Beam) 3: TRNTRUS (Beam) Delete Highlight Close	
	Assign methods C Associate to View C Associate to Selected Geometry C Associate to List List 14 To 19 Associate Help	
Ť,×		Load 1

Click anywhere in the drawing area to un-highlight the highlighted members. Also, **Close** the *Create Group* dialog box.

Three groups have now been created. If we now go to the *Select* menu, an option called *By Group Name* should be available. Let us click on that option, and the following dialog box comes up.

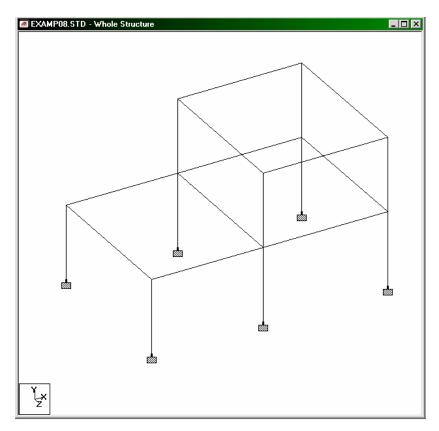
Select		
ko Nodes Cursor		
Beams Cursor		
Physical Member Cursor		
🧝 Plates Cursor		
🔀 Surface Cursor		
🔀 Solids Cursor		
Plates & Solids Cursor		
🍪 Geometry Cursor		
🕂 Filtered Selection Cursor		
Selection Mode		
Configure Selection Filter		
्रिया Load Edit Cursor	-	
म्लिम Load Edit Cursor 🏚 Support Edit Cursor		
Ale Support Edit Cursor		
	-	
A Text Cursor	_	
Beams Parallel To		
By <u>R</u> ange 🕨		
By Group Name	Colort Course	×
Entity at node	Select Groups	A
Entity Connects Beam	Group Name Show	w Label
Entity Connects Plate		
Entity Connects Solid	G1: _TOPCOR	✓G1: Close
By All	G2: _BOTCOR G3: _TRNTRUS	✓G2: ✓G3:
By Inverse		
By List		
By Specification		
By Missing Attributes	_	
Highlight Entities Sequentially		

It is apparent that we can now select the members using the group name method. If the check boxes for *Show Label* are switched off, labels such as beam numbers will not be displayed for these specific members, even if the beam numbering icon is switched on for the entire structure.

6. Displaying Loads on the screen

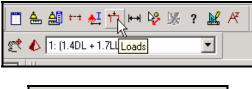
Steps:

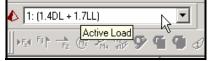
1. To demonstrate this feature, let us open EXAMP08.STD file. The structure will look as shown below.



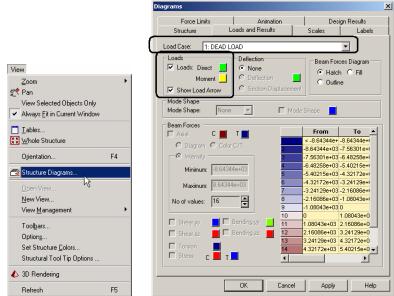
FPT-48 Frequently Performed Tasks

2. The easiest way to display a load is to click on the **Loads** icon as shown below. To choose a specific load, we can click in the list box called **Active Load** and choose the one we want.



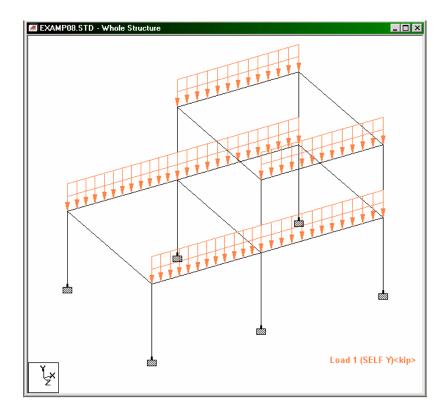


Alternatively, we may pull the *View* menu down and select the *Structure Diagrams* menu option. In the *Diagrams* dialog box that opens, select the *Loads and Results* tab. Check the *Loads* check box and select the load case we want from the *Load Case* list box. Then, click on the *Apply* button.



Drag the dialog box out of the way if it obstructs the view of the structure.

The following figure shows load case 1 displayed on our structure.



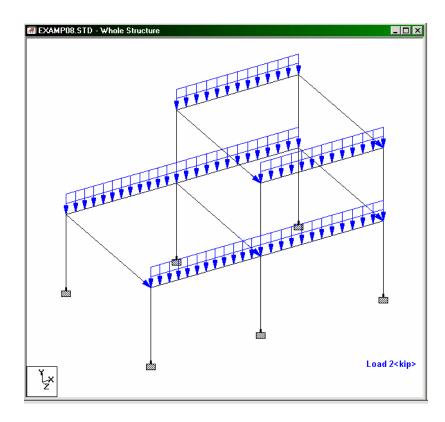
FPT-50 Frequently Performed Tasks

3. We can change the color in which load icons are drawn. Click on the color button alongside the *Loads* checkbox, and make a new choice from the color palette (see figure below).

Force Limits		nation		ign Results
Structure	Loads and Resu	uits	Scales	Label
oad Case: 1: DEA	.D LOAD			•
Loads	Deflection-		□	ces Diagram
🗹 Loads: Direct 📘	None			h O Fill
Moment _	C Deflection	on 📃	O Outlin	
Show Load Arrow	C Section	Displacement		
Mode Shape	_			
· -	one 🔻	☐ Mode	Shape	
<u>-</u>				
Beam Forces — C			From	То
		1	< -8.64344e+	-8.64344e+i
C Diagram C C	olor C71	2	-8.64344e+03	-7.56301e+i
└ Intensity		3	-7.56301e+03	-6.48258e+I
Minimum: -8.6	4344e+03	4	-6.48258e+03	-5.40215e+i
		_	-5.40215e+03	
Maximum: 8.64	4344e+03	-	-4.32172e+03	
		-	-3.24129e+03	
No of values: 16		-	-2.16086e+03	
1			-1.08043e+03	
Shear yy	🗖 Bending yy		0	1.08043e+0
snear yy			1.08043e+03	
	🗖 Bending zz		2.16086e+03	
🗖 Shear zz 📕		13	3.24129e+03	
				5.40215e+0
	-	14	4.321728+03	
	т	<u>14</u> ◀	4.321728+03	Þ

Let us assign say, **blue** color, as shown in the above figure. Select **load case 2** from the *Load Case* list box.

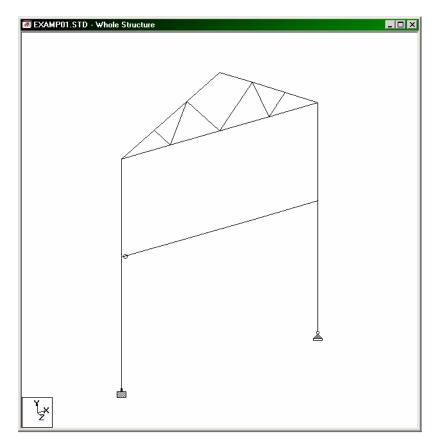
The following figure shows load case 2 displayed (in blue) on our structure.



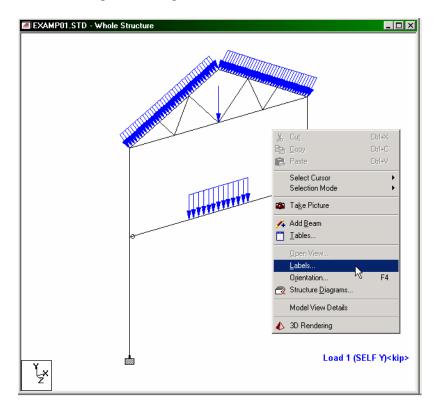
7. Displaying Load Values on the screen

Steps:

1. In the previous exercise, we saw the method for turning on load icons, but not the numeric value of those loads. To view load values, let us open EXAMP01.STD file. The structure will look as shown below.



Let us first display the load by clicking on the Loads icon 1.
 After this, click the right mouse button and choose the Labels option. Alternatively, we may click on the Symbols and Labels icon 1.
 or go to View | Structure Diagrams and select the Labels tab in the Diagrams dialog box.

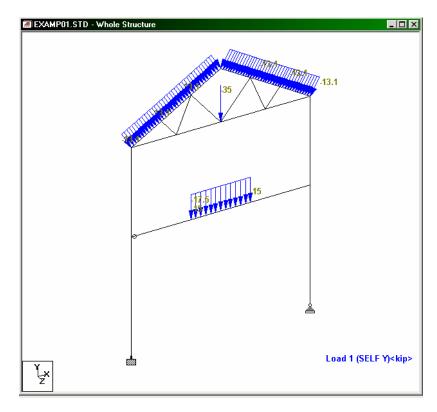


FPT-54 Frequently Performed Tasks

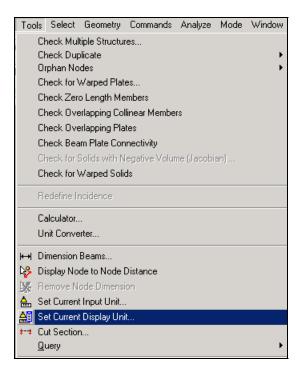
3. The dialog box shown below appears. Switch on the Load Values option under the *Loading Display Options* category as shown below.

Diagrams			×				
Force Limits Structure	Animation	Scales	gn Results) Labels				
Nodes Node Numbers (N) Supports (K) Dimension (D)	Properties C References (J) C Sections (X) C None	Physical Member Member Nu Design Grou Design Env Design Briel	mber (^M) ир (^G) elope (^E)				
Beams Beam Numbers (B) Beam Orientation (O) Beam Spec (A) Releases (R) Beam Ends (E)	Loading Display Options Load Values (V) Display Floor Loading Display Floor Load Dis Display Wind Load Co Display Wind Load (W	stribution (Y) ontributory Area (н)				
Start Color End Color	General ✓ Show Axes Window (^A) Master Slave (L) Show Axes At Org (I) Material (M) ✓ Show Diagram Info (G)						
Solids	Solids						
Colla Halileois (c) C Always Use Current Label Settings. Use partial labeling mode (Use labeling cursors to ON/OFF individual labels) For quick access to the labels using keyboard hotkeys, press Shift + the letter shown in brackets.							
[OK Cancel	Apply	Help				

The following figure shows our structure with the load values displayed on it.



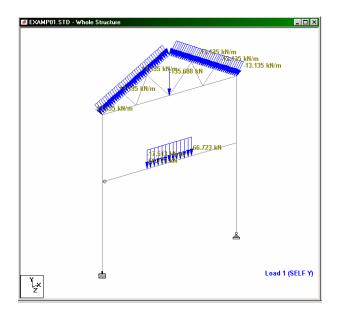
4. To change the unit in which load values are displayed, go to **Tools** | **Set Current Display Unit** menu command.



In the *Options* dialog box that comes up, select the Force Units tab. Then, change the selection under the Force, Distributed Force, etc. options as shown below.

Options		X
View Highlight	Force Units	
Tolerance Tables	Force	
Assign Dlgs Node Labels	Force kip 🗸 Show 3 🖨 dec places	
Beam Labels Plate Labels	Distr. Force kN/m Show 3 dec places	
Solid Labels	Moment kNm Show 3 🖨 dec places	
Annotation Structure Units	Distr. Moment kNm/m Show 3 🖨 dec places	
Section Units	Stress N/mm ² Show 3 🖨 dec places	
Force Units Dimension	Acceleration m/sec ² V Show 3 dec places	
	Rotation rad V Show 3 V dec places	
1	OK Apply Cancel Help	E,

The following figure shows our structure with the load values displayed in KN/m.



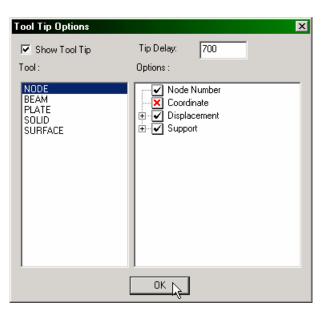
8. Structural Tool Tip Options

Structural tool tips offer a facility for displaying any customized input or output information about a node, beam, plate or solid element when the mouse cursor is placed over the structural entity. The tool tips are similar to the ones displayed when the mouse cursor hovers over a toolbar icon. When the mouse cursor is moved away from that spot, the information displayed is switched off too.

To activate the structural tool tips, go to **View** | **Structural Tool Tip Options** from the top menu.



The dialog box shown in the figure below will prompt for the data to be displayed when the mouse hovers a particular structural entity.

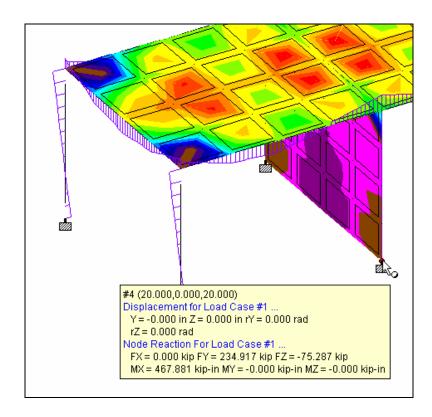


To turn the structural tool tips on or off, check the *Show Tool Tip* box in the upper left hand corner. The *Tip Delay* box signifies the amount of time it takes from when the mouse cursor first appears over an entity to when the tool tip actually pops up. This number is expressed in milliseconds (i.e. 1000 = 1 second). The options (items that can be displayed) for each entity are shown under the *Options* box. A check mark signifies that the particular data item will be displayed in the tool tip. An option with a "+" next to it signifies that further options can be enabled or disabled.

The figure below shows the options that can be turned on or off for Node Displacements.

Tool Tip Options		×
Show Tool Tip	Tip Delay: 700 Options :	
NODE BEAM PLATE SOLID SURFACE	✓ Node Number ✓ Coordinate ✓ Displacement ✓ Y ✓ Y ✓ Y ✓ Y ✓ R ✓ RY ✓ RZ ✓ MaxMinX ✓ MaxMinY ✓ MaxMinZ ✓ MaxMinRX ✓ MaxMinRX ✓ MaxMinRX	
	OK	

A red "X" indicates the data will not be shown in the tool tip. Simply click on the check box to turn an option on or off. The resulting tool tip that is displayed from the options chosen in the previous figure is shown in the figure below.



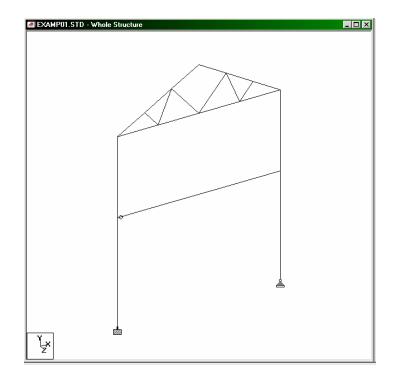
The tool tips automatically display the results for the active load case. All values are reported in the current display units.

9. Identifying Beam Start and End

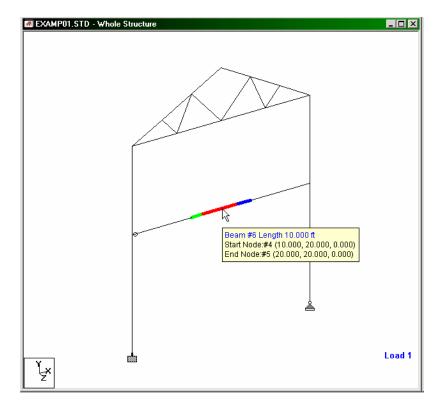
When assigning attributes like member releases or member offsets, one additional item of information which accompanies this information is whether that attribute is applied at the "START" node of a member or the "END" node. Hence, there has to be a way to quickly identify these two points without mistaking one for the other.

Steps:

1. To demonstrate this, let us open EXAMP01.STD file. The structure will look as shown below.

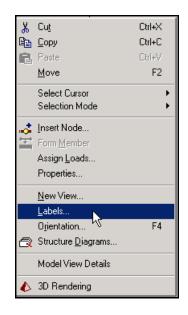


- 2. There are two different ways to identify the start and/or end of a beam:
 - a) simply place the mouse over a specific beam. The resulting tool tip will display the start node and end node of that beam in two distinct colors along with their corresponding coordinates. Of course, it is necessary to have the tool tips display turned on for this method to work.



FPT-64 Frequently Performed Tasks

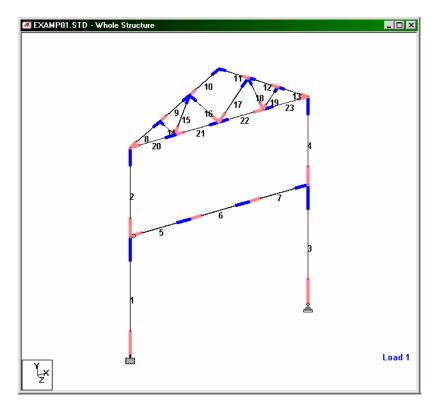
b) Click the right mouse button anywhere in the drawing area and choose Labels.



The following *Diagrams* dialog box will appear. Make sure that the *Labels* tab is selected. Then, switch the **Beam Ends** option on. To alter the color in which the start and/or end is displayed, simply click on the color palette next to the *Start Color* and/or *End Color* options and choose the color you want. Then, click on **Apply** for the changes to take effect immediately. Click on **OK** to close the dialog box.

)iagram s			
Force Limits Structure	Animation	Design F Scales	Results Labels
Nodes Node Numbers (N) Node Points (K) Supports (S) Dimension (D)	Properties C References (J) C Sections (X) O None	Physical Members Member Numbe Design Group (* Design Envelop Design Brief (*B	`G) e (^E)
Beams Beam Numbers (B) Beam Orientation (O) Beam Spec (A) Releases (R) Beam Ends (E) Start Color End Color	Loading Display Options Load Values (V) Display Floor Load D Display Floor Load D Display Wind Load C Display Wind Load (V General Show Axes Window (' Show Axes At Org (I) Material (M)	istribution (Y) iontributory Area (H) M)	/e (L.)
Plates Plate Numbers (P) Plate Orientation (T)	Surfaces		ion (^T)
Solids Solid Numbers (C) Use partial labeling m	 Always Use STAAD. Always Use Current I Ode (Use labeling cursors to 	Label Settings.	-
	ccess to the labels using key s Shift + the letter shown in b		
	OK Cancel	Apply	Help

The following figure shows the structure with the start and end of all beams displayed in two separate colors.

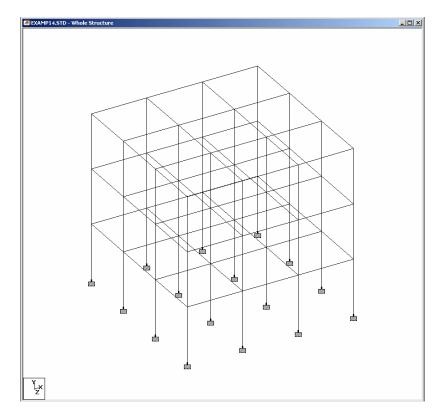


10. Plotting from STAAD.Pro

Explained below are 5 methods for plotting the drawing of the STAAD model and STAAD result diagrams.

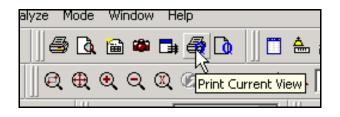
Method 1 : Using the Print Current View option

1. We shall use example problem 14 to illustrate this feature. Open the example. The model should appear as shown below.



FPT-68 Frequently Performed Tasks

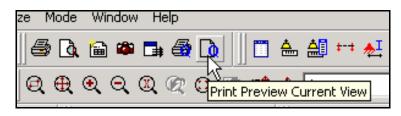
2. To plot this picture, click on the **Print Current View** icon as shown in the following figure.



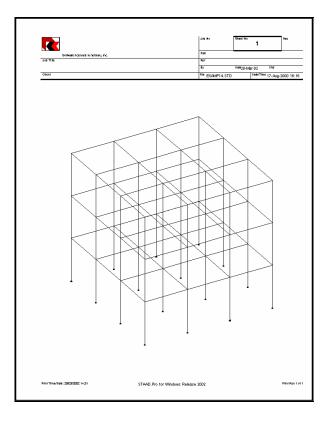
3. The standard Windows *Print* dialog box will appear asking you to choose the printer to which you wish to plot the drawing.

Prin	ıt		<u>? ×</u>
Г	Printer		
	Name:	\\RECAWEB01\Xerox DocuT	ech 6135 PS
	Status:	Ready	
	Туре:	Xerox DocuTech 6135 PS2 v1.	3
	Where:	XER0X6135:DP6135	
	Comment:		Print to file
Γ	Print range		Copies
	• All		Number of copies: 1 🚔
	C Pages	from: 1 to:	
	C Selecti	on	1 2 3 Collate
			OK Cancel

4. If you wish to catch a glimpse of the plot, as it would appear on paper, select the **Print Preview Current View** option.



A preview screen similar to the one shown below will be displayed.



Method 2 : Using the *Take Picture* option

1. Bring up the diagram of the STAAD model you wish to print. On the *Print* toolbar of the STAAD screen, you will find an icon of a camera called **Take Picture**.



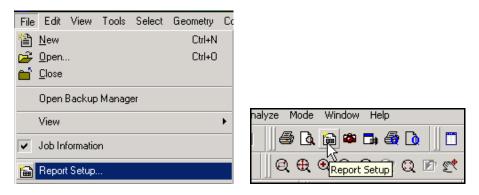
Alternatively, this option can be accessed from *Edit* | *Take Picture*.

Edit	
🗠 Undo	Ctrl+Z
으 Redo	Ctrl+Y
X Cut	Ctrl+X
唱 Copy	Ctrl+C
💼 Paste	Ctrl+V
\times <u>D</u> elete	Del
🍘 Ta <u>k</u> e Picture	N
Copy Picture	N
😴 Edit Input Command	d File
Create New VB Ma	cro
Edit Existing VB Ma	icro

2. A dialog box will appear prompting you to provide a Picture ID and caption.

Picture 1	×
ID: Picture 1	
Caption: Whole Structure	
OK Help	

3. Next, select **Report Setup** from the **File** menu, or click on the *Report Setup* icon as shown below.



4. A *Report Setup* dialog box will appear. In the **Items** tab, select **Pictures** from the available list of items.

Report Setup					X
Picture Album	Options		ne and Logo	Load/S	į
ltems Loa	d Cases	Modes	Ranges	Steel De	esign
Available		Selec	ted		
Input	•				
Input Output		Job I	nfo		
Pictures		>			
Reports		>>			
Materials					
Supports					
Releases					
Primary Load Cases		<			*
Combination Load Ca	ses 🔡 🦳				–
Loading Generators	.	<<			
	Report Detai	Increments	10		
	incport Detai	rincrements			
	OK	Ca	ncel Pr	int	Help

If more than one picture is available, select the one you wish to plot by pressing the → button. Next, go to the Picture Album tab and switch on Full Page. Click OK.

Report Setup	×
Items Load Cases Modes Ranges Picture Album Options Name and Logo	Steel Design
Name: Picture 1 Delete Picture	Display Size
× · · · · ·	Height: 174.7 mm Width: 192.81 mm
Caption: Whole Structure	
OK Cancel	Print Help

6. From the *File* menu, select *Print* – *Report*. The diagram will be plotted.

Alternatively, from the File menu, select Export Report | MS Word File.

							Fi	requently	Perform	ned Tasks
File	Edit \	/iew	Tools	Select	Geometry	Co	mmands	Analyze	Mode	
眢	<u>N</u> ew				Ctrl+N		🔬 🗉	E 👬	≻	
2	<u>0</u> pen				Ctrl+O			10		
6	<u>C</u> lose						o 4	⊙∭₿	: Q	
	Open Ba	ackup	Manage	er			т 🌐	0 🎢	% (#)	
	View					۲	Structu	Ile		
~	Job Info	rmatior	n							
6	Report 9	Setup								
	Printer S	etup								
	Print					×				
à	Print Pre	<u>v</u> iew P	Report							
	Export R	leport				۲	<u>T</u> ext	File		
	Import						<u>M</u> S \	Word File		
	Export									

FPT-73

7. In the *Save As* dialog box, type in the name of the ".doc" file you want and click on the *Save* button. In the template dialog box, select the *Normal* template.

Template			<u>?</u> ×
Look jn: 🔂	Templates	▼ ← 🗈	r 🗐 🕂
図 ~\$Normal. 図 Normal.dol	**		
, File <u>n</u> ame:	Normal.dot		<u>O</u> pen
Files of <u>type</u> :	Templates (*.dot)	•	Cancel

Microsoft Word will now start building the file with the picture in it. Once this task is completed, Word will open this file, and program can be accessed from the Windows Taskbar. Browse the file, and if you are sure the image meets your needs, select *Print* from the *File* menu and follow the instructions.

Method 3 : Using the Export View option

1. STAAD.Pro has a facility to export the drawing to a graphic image file. The icon for this facility is called **Export View** and is available in the Print toolbar.



2. When the diagram that you wish to plot is displayed on the screen, click on the **Export View** icon. Select the graphic format in which you wish to save the file (i.e., .bmp, .jpg, .tif, .gif, etc.)

Save As	, , ,		?×
Save in: 🔂	222) 🗈 🕈 💌	≝
File name:			Save
Save as type:	Bitmap File(*.bmp)	•	Cancel
	Bitmap File(*.bmp)		
	Jpeg File(*.jpg) Tiff File(*.tif)	2	
	Gif File(*.gif)	~\\ 	
	Pcx File(*.pcx)		
	Targa File(*.tga)		

Provide a file name and save the file.

3. Then go to a document publishing or graphic editing program such as Microsoft Word or Adobe Photoshop. Import the drawing from the file saved above. In Word, this can be done using *Insert* | *Picture* | *From File*. Once you are sure the image meets your needs, select *Print* from the *File* menu and follow the instructions.

Method 4 : Using the Copy Picture option

1. Highlight the window containing the diagram that you wish to plot. This can be done by making sure the title bar of that window has the color which indicates that it is in focus. From the **Edit** menu, choose **Copy Picture**.

Edit		
💭 Undo	Ctrl+Z	
🖳 Redo	Ctrl+Y	
X Cut	Ctrl+X	
Ba Copy	Ctrl+C	
🔁 Paste	Ctrl+V	
🗙 <u>D</u> elete	Del	
🝘 Ta <u>k</u> e Picture		
C <u>o</u> py Picture	2	
Edit Input Command File		
Create New VB Ma Edit Existing VB Ma		

2. Next, run a graphics program which offers facilities for handling graphic images such as Microsoft Paint, Adobe Photoshop, etc. In the Edit menu of that program, select Paste. You will notice that the drawing from the STAAD window is now pasted in that program. Using the tools provided by the graphics program, you can modify the drawing if you wish. Once you are sure the image meets your needs, select *Print* from the *File* menu and follow the instructions.

Method 5 : Using the Windows Clipboard

When the diagram that you wish to plot is displayed on the screen, press the "Print Screen" key on your keyboard or "Shift-Print Screen", depending on how your keyboard is set up. The entire image currently in view will be copied to the Windows clipboard. Next, run a graphics program which offers facilities for cropping portions of a graphic image (such as Microsoft Paint, Hijaak, Full Shot, etc.). In the *Edit* menu of those programs, select *Paste*. You will notice that the entire STAAD screen including the diagram, menus, toolbar icons, tables, etc., which were in view, are part of the pasted image. Using the tools provided by the graphics program, crop the image so as to preserve just the drawing and eliminate unwanted portions. Using the *File* | *Print* option, print the drawing.

N o t e s

- N o t e

- s