

STEEL

Quick Start Guide

1. Installation – Launching
2. Example

Preface

This Quick Start Guide provides a fast and friendly introduction on STEEL main features and functionalities. All the features and functions of the program are presented and explained in detail within the complete User's Guide, along with informative examples.

STEEL is an advanced workstation for the designer of Steel Construction, offering a complete and reliable solution in the Analysis and Design of Steel Structures of any type, shape and size. The program integrates a user friendly CAD Interface, along with a reliable calculation environment, based on the existing regulations and standards. The basic core of STEEL consists of four, synergistically cooperating among each other, components, the Space Frame Analysis & Member Design component, the Connections one (for Simple Connections - Truss Connections, Moment Connections - Frame Connections), the Drawing of Frame & Connections module and the Dynamic Spectrum Analysis component. In addition, STEEL is empowered by four sophisticated Add-Ons which provide the program with an unrivaled advantage: a) PreSTEEL, a smart wizard for the quick insertion of the model, based on existing templates of common and typical cases, b) SteelCONNECT, the tool permitting the design of any type of frame connections, c) STEEL Section Editor, enabling the creation of any kind of section, and d) SteelPLOT, for high quality final drawings, full detailed and properly updated according to the calculation results.

Apparently, this is not easy to describe all STEEL features and capabilities in a few pages quick start guide. For this reason, this guide is divided into 2 short parts, Part 1 that describes the installation procedure plus the main menu structure, and Part 2 that describes an example, step-by-step. This example gives a good idea about the philosophy of the program, and shows the user that learning all the options and capabilities is just a matter of getting familiar with the package.

1. Installation - Launching

1.1 Installing STEEL

1. Insert the CD in your computer CD-ROM drive (e.g. D:, E:) or, if you received your software via Internet, run the installation application you downloaded.
2. When the Setup window appears, choose the language for the installation and click OK.
3. When the Welcome page appears (as shown below), click **Next**.



4. Follow the on-screen instructions until the installation is completed.
5. After installation, the program is located within the programs list.

1.2 Program Structure

STEEL comprises of the following parts:

- Members
- Foundation
- Truss Nodes
- Frame Nodes
- Drawing

MEMBERS

It is the basic part of the program. In the working environment of MEMBERS you can describe your structure and calculate it. You can also check the members adequacy. The MEMBERS application is autonomous, which means that the presence of another program or another part of VK STEEL is not required. If no other parts of VK STEEL are installed then structure analysis is confined up to the members check. In other words it is not possible to check the nodes, whether it is a truss or a frame structure, or to produce drawings.

FOUNDATION

The FOUNDATION sub-program takes over the check and design of the column base as well as the foundation members. The structure foundation may be done either in individual bases or in a system of footings and connecting beams. Although it is an independent program it is part of MEMBERS.

TRUSS NODES

The Truss Nodes sub-program takes over the check and design of truss structures nodes. It also provides the possibility to produce drawings of the structure model. To be able to check and design the nodes, the truss must be previously described in MEMBERS. To confirm that the Truss Nodes program has been installed, you have to look for the Connection Design (Truss) command in the ANALYSIS menu of the program. If the command is absent or disabled then the Truss Nodes program is not installed properly.

FRAME NODES

The Frame Nodes takes over the check and design of frame structure nodes. It incorporates the possibility to produce its own technical report – without reference to the one produced by MEMBERS – and connections detail drawings. To confirm that the Frame Nodes program has been installed, you have to look for the Connection Design (Frame) command in the ANALYSIS menu of the program. If the command is absent or disabled then the Frame Nodes program is not installed properly.

DRAWING

STEEL can produce and export most of the drawings that are required for the completion and validation of a project. The following table features the sorts of technical drawings that are available, depending on the program's composition:

DRAWING SORT	VK STEEL PART MEMBERS+		
	NODE	DRAWING	FRAME NODE
STRUCTURE DRAWING	Y	Y	
TRUSS NODE CONNECTION DRAWING	Y	Y	
FOUNDATION DRAWING		Y	
COLUMN (...) DRAWING		Y	
FRAME NODE CONNECTION DRAWING			Y

Table 1. Drawing creation availability depending on the composition of VK STEEL (e.g. to produce the Frame Node Connection you should have MEMBERS and FRAME NODES – the other parts of the program are not essential).

All drawings processing and printing is done in an autonomous drawing program, which is installed in a separate sub-folder, in the basic STEEL program folder. To confirm that the DRAWING part is properly installed, look for the ICAD directory in STEEL. This drawing platform is based in 4M's IntelliCAD and has similar functions with AutoCAD.

1.3 Basics of the Working Environment

STEEL is based on the object oriented data processing technology. The formation of the simulation to be analyzed, the graphic display of the mathematical model, is reduced to the insertion of the elements and calculation of their properties.

STEEL elements are the nodes and members. Depending on the element type, the program forms the properties directory.

The "node" object, for example, is the meeting point of the center axes of the members, so the properties directory that is formed for every such element includes geometric features (e.g. coordinates in the global coordinate system, load values that are implemented in the node coordinates).

On the contrary, the "member" object represents an entity with physical essence (e.g. a steel beam of a defined length) and as it is obvious, in the properties directory physical features (e.g. steel quality, weight etc) are included.

The "object-oriented" term is referred to the complete independence of each element and the possibility for the user to alter one of its attributes, without the modification of the attributes of the neighboring elements or the whole model.

Thanks to this technology, apart from the easy elements input of elements and modification of their attributes, it is possible to copy, cut and paste those attributes at any time.

Another important convenience is the possibility to undo an action or even a whole series of actions that have preceded. As is the case of most programs created for windows, the directory of the actions to be undone is refreshed when the file is saved. In other words,

when the Save command has been chosen the possibility to undo previous actions is no longer available

All work on STEEL is based on user – program interaction. The program, depending on the work stage, prepares the appropriate choices frame, in which the user should work by choosing the commands and defining the values of the parameters. Those choices commit the program in its turn to accordingly prepare for the next stage.

Information exchange between the user and the program is structured in various entities, so that the procedure is simplified and the efficiency improvement.

The entities that are used in STEEL are:

NAME	Nature – Usefulness
Title Bar	Informational. The name of the program and information on the project are included.
Menu Bar	Functional. It includes all the commands of the program, organized in menus.
Tool Bar	Functional. It includes almost all the commands of the program, organized in shortcuts (buttons).
Status Bar	Informational - functional. Useful messages about the work stage in the Project Window appear there. It also features some commands.
Project Window or Graphic Display Window	Informational – Functional. It features the model, in every work stage, as it is formed and saved, much information relating to the changes that are being done in it, while according to the choice, access to the most useful commands is provided, through accordingly formed pop-up menus.
Dialog Box	Functional. The program carries many different dialog boxes, different to each other, according to the functions they are destined for. It is one of the most essential tools of the interactive communication between user and program.
Properties Frame	Informational – Functional. On the one hand they feature all the properties of the object that has been chosen and on the other hand they provide the user with the possibility to modify them.
Message Slide	Informational. It features some of the useful information on the object that has been chosen.
Pop-up menu	Informational. It is the same as all other menus, only in this case it is called by right-clicking on the project window. It includes shortcuts and their content It is formed according to the object choice

Table 2. Table entities and their interpretation.

WORK ENVIRONMENT PRESENTATION

Title Bar

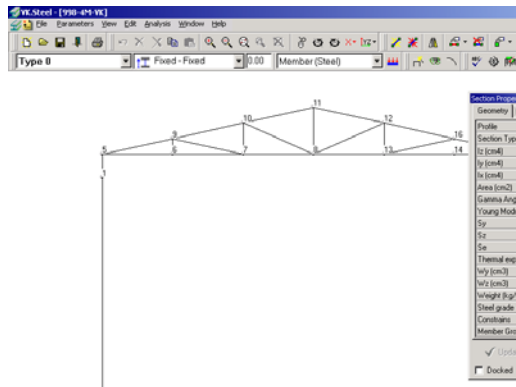
It includes the program logo as well as information on the number of the current project

Toolbars

They may be floating or permanently placed along the sides of the main window. They include all shortcuts (buttons) of the program commands

Menu Bar

All program commands are grouped in one of the menus that comprise the menu bar



Window operation keys

They are the typical WINDOWS keys for maximizing, minimizing and exiting the basic window and the current project window

Horizontal and vertical sliding bars

Changing their position also moves the model through the graphic display window.

Graphic Display Window

The model is displayed here and it is processed with the use of the graphic processing tools.

Message Slide

According to the object choice, information is provided e.g. about the nod or the member, their number and their coordinates

Status Bar

Useful information about current program functions is provided

Properties Frame

In this frame you can search and modify the properties of the members you have chosen. The fields of the frame is automatically updated if the member choice has changed

2. Example

The main goal of the example that follows is the quick familiarization of the user with the working environment of the VK STEEL program. For this reason emphasis has been given to the structure definition procedure (model design), while to avoid an information overload the validation of the elements calculations is not extensively commented.

In the example what is explained are the procedure of the design of a simple frame structure, its calculation, the members efficiency check according to the requirements of Eurocode 3 as well as the check and design of one of its nodes.

After the completion of this example you will have

- A quick familiarization with the most important commands and functions of this program
- A concise view of the work flow in the program
- A project example that you can use in the future, as reference

Read the next paragraphs of this introduction. Their aim is to inform you on the project you intend to create, as well as the ways to look for help about operating matters, that are being covered in another part of the user manual, or the on line help system.

Specifically,

- The next paragraph, STRUCTURE OF THE EXAMPLE, presents, graphically, the main steps that will be explained in the next chapters.
- The LOOKING FOR HELP paragraph guides you so that you can take advantage as much as possible, of the help system, as you are working on the example
- The ABBREVIATION paragraph includes a table with the printing conventions of this example. This table is an essential reference help for the understanding of the abbreviated words that are used for the sentences.

NOTE: Within the INTRODUCTION Chapter of the complete user manual you will find, among others, a short presentation of the working environment and the features of the program.

STRUCTURE OF THE EXAMPLE

The procedure is structured in ten steps, organized in two units. The first unit (steps 1-7) include the actions you can do with the main part of the VK STEEL program, MEMBERS. This unit corresponds to the largest part of your work in the program, since after the completion of step 7, you will have solved the structure and have secured the efficiency of its members, according always to the requirements of the selected regulation (in our example we have selected, as Members Regulation, Eurocode 3 and as Aseismic Code, NEAK).

The second unit, (steps 8-10) is mainly about nodes design, foundation check and use of the program's results (Printings and Drawings).

The program structure is the following:

FIRST UNIT

1. Step 1 Introduction – Creating the project folder
2. Step 2 Preliminary work – Actions that are required before the description
3. Step 3 Defining the project parameters
4. Step 4 Defining the loads and the load combinations
5. Step 5 Structure description. Modifications and corrections
6. Step 6 Diagnostic checks and calculation
7. Step 7 Members check

8. SECOND UNIT

9. Step 8 Foundation Design
10. Step 9 Connection Design
11. Step 10 Results

LOOKING FOR HELP

This example has been designed in such way that it guides the beginner user, quickly and in a concise way in the completion of a project. For that end the references to alternative ways to the same function or more detailed explanations have been omitted. However, if you need any clarification you will need to read the printed user manual where the relevant subject is explained in more detail. For example, in STEP 3 Defining the project parameters only the parameters determination procedure is mentioned, which is needed in the project. Details on the rest of the parameter can be found in paragraph MATERIAL LIBRARY of chapter THE "PARAMETERS" MENU.

An other fruitful way to look for more details is through the electronic online help system that accompanies the program.

For the same subject (the "materials" and their parameters) look for details following the next procedures.

Start up the program

Press the [F1] key, or execute the command Help>Index (in other words select the Index command which you will find in the Help menu). The electronic help window of VK STEEL will appear. At the left side of the window you can see the tabs Contents and Search. At the center the larger window features the texts of each subject as well as the Hyperlinks between the pages.

1. **FIRST WAY** Select with left click the Contents tab. The list with the contents of help will appear, in hierarchical order.
 - a. Select, with left click the subject THE PARAMETERS MENU.

b. On the page, at the central window, left click on the hyperlink "Material Library". In the same way you can read all the units that comprise the Material Library subject, by choosing the relevant hyperlink.

2. **SECOND WAY.** Select the Search tab. The word/phrase search engine will appear, which is part of the help system.

a. At the field {Type in the keyword to find} type the word {Materials}

b. Select with left click the [List Topics] button. All the page titles that include the word Materials will appear.

c. To end up in the same page, as in the first way, select the title Materials and press the button [Display Topics]. Experiment with the other titles.

ABBREVIATIONS

No	Conventions	What it is for	Comments – explanations
1	File>New	Program command	It means that you must execute the New command, which you will find under the File menu of VK STEEL
2	{This Project has not been saved. Are you sure?}	Message Content. Text you should type	Usually the execution of a command is followed by the appearance of a message
3	[CLOSE]	Sign on a button	Other examples: [OK], [CANCEL]
4	<DIN>	Sign on a list or a field	Select one of the contents of a list, after you have opened it, by clicking on the word with the left button of the mouse
5	LC	Left Click	Abbreviation of the phrase: Click with the left button of the mouse
6	RC	Right Click	Abbreviation of the phrase: Click with the right button of the mouse
7	LDC	Left Double Click	Abbreviation of the phrase: Click twice with the left button of the mouse
8	C:\vk\windows\steel	Place of the steel folder	It means: The steel folder is in the windows file, which is in the VK file of hard drive c
9	C:\vk\windows\steel\Winsteel.exe	Place of the Winsteel.exe file	It means: The Winsteel.exe file is in the ... etc
10	C:\vk\windows\steel\...	Contents of the steel folder	It means: All the files and all the folders that are in the folder steel, which is ... etc

FIRST UNIT

STEP 1 – CREATION OF THE PROJECT FOLDER

Working with VK STEEL starts with the creation of a new project and the appropriate adjustment of the program environment.

In the first step you will start up the program and create a New Project. Every new project creates a directory in C:\vk\windows\steel\meletes\... as well as a first series of essential files. The place where the program looks for the saved projects is determined and therefore you should not change it, until you have fully completed your work. If you move, for any reason, the project folder to another part of the disk, this project will not be available anymore, on the “PROJECT” dialog frame.


- Follow the procedure

Flow


User Actions

Program Start Up

Start the STEEL 2001 application doing one of the following,

1. Select the WinSTEEL command from the typical Windows START>PROGRAMS> WinSTEEL menu.
2. LDC the icon that you will find on Windows desktop. (If this icon does not exist, read the notice bellow about the ways to make an , on the Windows desktop)

New Project

3. Select the command FILE>NEW or LC the . The dialog frame {NEW PROJECT} will appear
4. With LC select the {Steel} tab and make sure that there is no other project with the same number you are about to use.
5. Select again the {New} tab and LC the {Project} icon. Press [OK]
6. On the {Steel New Project} dialog frame type the number 100 on the field {Project}, and the text {Simple frame structure} on the “Information” field.
7. Press [OK]. The program has created a directoy named {100} where all files of the 100 project will be stored. It has also formed appropriately the project window and displays the toolbars and the program menu bar.
8. RC on any icon. The TOOL BARS command will appear. Make sure that all check boxes will be checked. If any of them is not checked, then LC on the check box to select it.

SHORTCUT CREATION on the Windows desktop.

If a program icon has not been automatically created you can very easily create it by following the procedure bellow:

1. RC anywhere on the desktop, as long as there is no other shortcut at this place.
2. On the pop up menu that will appear select the "New" command and LC on the command "Shortcut". The "CREATE SHORTCUT" dialog frame will appear.
3. Press the button [BROWSE]. The familiar dialog frame for files browsing will appear. Look for the file Winsteel.exe, in the program folder. If during set up you didn't change the default place of the STEEL folder, then you can look for the file at c:\vk\windows\steel\WinSTEEL.exe
4. Press the [NEXT] button.
5. Press the [FINISH] button.

MOVE – COPY – COMPRESS A VK STEEL PROJECT

To move a project of the program from the directory they were saved all you have to do is move the folder with the name of the project. The same applies for copying a VK STEEL project. The compression of the project is a simple procedure as well, although it requires the existence of a relevant application. Many programs or compression algorithms allow the compression of files but not of directories. A VK STEEL project, though, does not comprise of only one file but of a group of files. This means that for the right compression you have to create first the compressed file (e.g. if you use the ZIP algorithm through an application, such as WinZip®, the 100.zip file) and you have to include all the files that are included in the 100 folder of the VK STEEL project. This can be achieved with the help of the command Add to archive of the compression program.

STEP 2 PRELIMINARY WORK – REQUIRED TASKS BEFORE THE DESCRIPTION

Before we move on to the structure definition, in the program, we have to mention some comments about the tasks that are required to be done first.

INTRODUCTION

The VK STEEL program is designed in such way that it allows any corrections that you may need to do during the model definition phase. However the best way to start your work is if you have from the start a concise view of your data and you project's requirements.

- More specifically you should
 1. Have a clear view of the geometry of the structure and also of the section categories that will be used. In other words, you should bear in mind any architectural or other constraints and have checked the extent in which the structure you are about to define meets them or not. In regard to the sections, the correct choice of category will save much of your precious time.
 2. Have done some kind of preliminary work in regards to the load. In VK STEEL you can input two sorts of load,
 - a. distributed along the member, and
 - b. concentrated , forced on the nodes. Both the distributed and concentrated on the members and nodes respectively, will have to be given in units that are according to the regulation that will be used for the design of the members. If, for example, Eurocode 3 (EC 3) is being used then the distributed load should be given in kN/m and the concentrated in kN. For the time being the program is not able to process the loads and therefore the modification of the load from kN/m², as they emerge from the regulations, to kN/m as they have to be input distributed on the members, will have to be done by the user.

In this unit we will give a description of the problem that we are about to describe and solve. Some details will be given about the project data and some drawings and images in which you will be able to get as reference any time you have a query during the definition of the structure.

NOTE: The regulations that will be used for the structural analysis and design of he structure are NEAK and Eurocode therefore to define the loads the relevant provisions of Eurocode 1 have been taken into account.

PRELIMINARY MODEL DEFINITION DATA OF THE EXAMPLE

Introduction

The structure we are about to define is a simple space frame that could represent a shed.

Geometry

The structure comprises of two parallel frames that are connected with secondary beams. Each frame comprises of two columns of height 4 m and a pair of leaning beams with an 18.43 degrees angle from the horizontal axis.

The frames are connected by parallel secondary beams that, as it is conceded, take part in the structural analysis.

To avoid any unwanted relevant distortions, a pair of bracings has been placed in each leaning plane, comprising of members, which it is conceded for that they take only tension. Those members will be hollow beam of rectangular section and be connected to the main beams through brace struts.

The parallel secondary beams system comprises the main mechanism of load transfer from the cover (of steel leaves of trapeze section with expectation – in the load – of future insulation), of wind pressure and sub-pressure by snow and moving loads (temporary load, as is provided by Eurocode 3 for frames with an angle less than 20 degrees).

The form of the structure can be seen in the series of successive drawings and images.

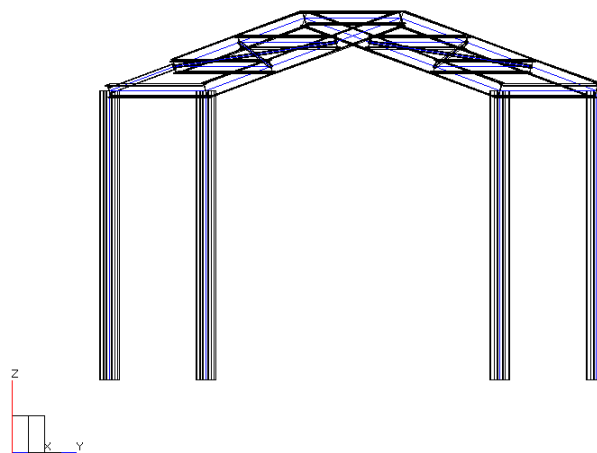


Image 1. Three dimensional model of the structure after its definition

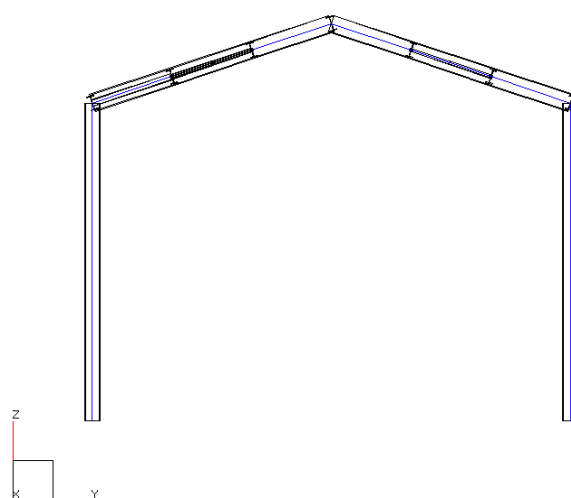


Image 2. View of the first frame

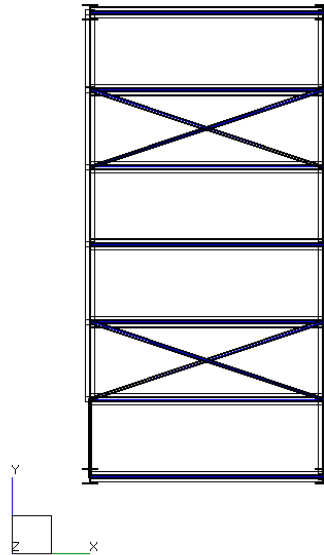


Image 3. Ground Plan (These images have been created with the program VK STEEL through the command Export To> Image, combined with the RENDER command)

Design Codes

The regulations that have been followed are:

- For loads: Eurocode 1
- Aseismic: NEAK
- For the members: Eurocode 3

When it comes to other assumptions from the aseismic code the following assumptions have been taken:

1. The structure is in Athens
2. The Importance of the building is $\Sigma 2$
3. The soil is class B
4. The building although a frame, will be designed in elastic area (section class 3) with earthquake behavior factor $q = 1.5$
5. The foundation coefficient is $\theta = 1.0$
6. $\zeta = 3.0 \%$

According to the assumptions above the components of the earthquake coefficient in the YX plane are $R_d(T)/g_{x-x} = R_d(T)/g = 0.31$ while the activity of the vertical component of the earthquake is considered negligible.

Loads

For the load of the structure the following assumptions have been made:

- **DEAD** Dead load strains the secondary beams as distributed load. The main beams are being strained due to dead load only on the secondary beams base points

- **WIND & SUB-PRESSURE** The wind's load, for the needs of the example has been reduced to only one direction with angle $\alpha = 0$. In other words the vector of the resultant of the load (that corresponds to wind pressure) has the direction of the positive Y-axis. Based on this assumption the value and direction of the distributed load has been calculated on the columns and secondary beams for the case of positive pressure (LC 4: Wind +Y) and sub-pressure (LC 6: Wind -Y). Because the frame does not have any draperies (or other surfaces) on a level that is vertical to the X-X axis the load of the wind in this direction has not been taken into account while, to avoid any complexity, the relevant Eurocode 1 prevision has been ignored.
- **SNOW** For snow load the simplest case of different load has been taken, on the left and right leaning surfaces of the frame. Like in the case of wind, the load strains the secondary beams as a uniformly distributed along them. Snow load is being developed in two Load Cases: LC 8: Snow 1 and LC 9: Snow 2.
- **LIVE** According to Eurocode 1, even in cases of non-passable roofs, such as the case in this example, the live load for the case of repair should be taken into account. This load that will also be known as [...] is taken based on the weight of the workers that will make the maintenance work.

The resulting load due to frame imperfections has been ignored as negligible.

Connections

- **COLUMNS** The columns are considered to be fixed on flexible footing of cube shape, connected with inflexible beams. The support node is considered to be fixed and the assumption that there is a suitable mechanism that transfers the torsional moment to the foundation system. To make more strict the overturning check of the footing, the participation percentage of the beams (their inflexibility) of torsional moment receipt has been reduced. In terms of the frames, the nodes that connect beams to columns are considered fixed.
- **BEAMS** The beams have been considered as fixed in terms of their connections to the columns as well as each other.
- **SECONDARY BEAMS** The secondary beams have been considered as pinned in both sides.
- **ROOF BRACES** We have already talked about the crosswise pairs of roof braces. We should say here that they will be defined with a buckling length that is twice the actual one to make sure that the relevant check will succeed. In practice the pair of roof braces is set up ready on the frame and consists of four members which are connected in their meeting point with a brace strut. In the program we will define those members from the node of one frame to the one on the diametrical opposite of the other frame (in other words we will define two members of each pair with length twice the actual one).

STEP 3 DEFINING THE PROJECT PARAMETERS.

On the 3rd step of the project you will select the aseismic code, the members check code and define all their relevant parameters. You will also create the Project Library having made the appropriate choices from the General Section Library that accompanies the VK STEEL program. The Project Library sections will be used later during members insertion (STEP 5).

The whole parameters defining procedure will take place through the "PROJECT PARAMETERS" dialog box.

- Follow the procedure

PROGRAM FLOW	USER ACTIONS
Code Parameters defining	<ol style="list-style-type: none">9. Select the command Parameters>Material Library. The {PROJECT PARAMETERS} dialog box will appear, which includes all the essential code parameters, which are organized in successive tabs.10. Pop-up (if it is not already visible) the {CODES} tab with LC on its title.11. On the lists {Design Codes} and {Aseismic Code} select <Eurocode (EC3)> and <NEAK>12. Select the <Equivalent Static Analysis Eq 3.7> for the lateral seismic forces distribution13. On <LC Factor (Live)> type {0,30}14. On <Termal Loading Value> set to zero {0}15. Make sure that the check box <Set as Default> is not checked. <p>On the right side of the tab make the following changes:</p> <ol style="list-style-type: none">a. Choose <Zone II> for seismic risk zone.b. Choose <S2> for category of importance.c. Choose for soil class.d. Without clicking any of the {Influence Factor of Foundation} options, type <1,5> for the <q> factor value.e. Type <0,15> in the <Tx> and <Ty> fields

- f. Type <0,15> and <0,6> in the <T1> and <T2> fields, respectively
- g. Type <1,5> and <2,5> in the <Bd(0)> and <Bd(T1)> fields, respectively
- h. Type <0,375> in the <Bdmin> field
- i. Type <3> in the <Damp.Ratio> field
- j. When you are finished with all necessary changes the <Code> tab should look similar to the one illustrated in the following figure.

NOTE: Defining $Rd(T)/Bd(T)$, $Rd(T)/g$ X-X and $Rd(T)/g$ Y-Y is done automatically so it is not necessary to type anything on those fields.

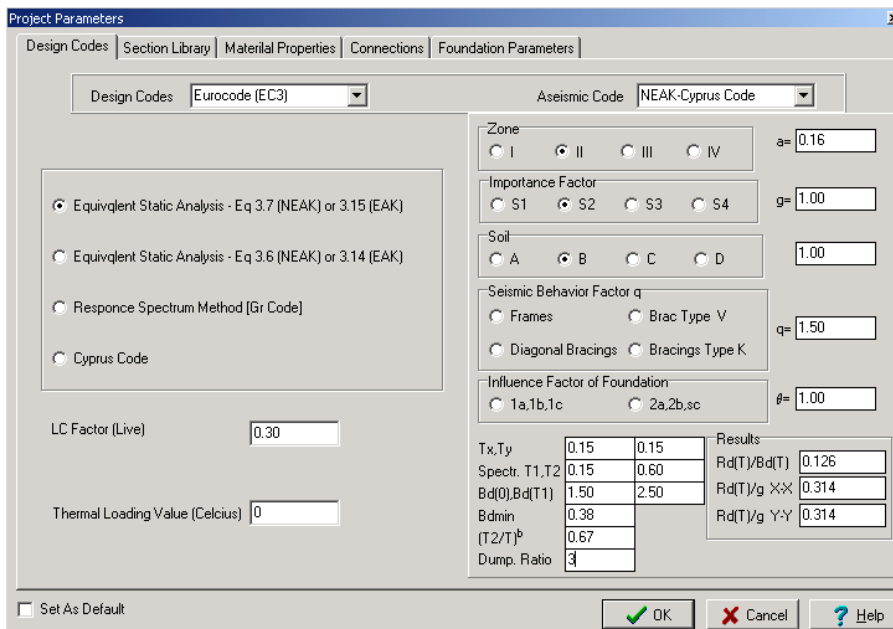


Image 4 The "Design Codes" tab after defining the parameters.

PROGRAM FLOW

Sections Selection- Project Library Creation

USER ACTIONS

16. The next step is to select the sections that will be used for the structure members you are about to define. With LC select the {Section Library} tab. The tables that comprise your tab provide you with access to the General Sections Library, which will enable you to create the Project Library, in other words a list of the sections that will be used in your project.
17. Activate – if it is not already active – the check box next to <Fe360> and make sure that all other steel qualities are not checked. All sections you will select, from now on, will be of Fe360 steel quality.

18. Select with LC on the left side of the dialog box the IPB sections. The table in the middle will show the list of all available IPB sections along with their properties
19. With LC select the IPB200 section to select it.
20. Without moving your cursor away from the selected row, with RC pop-up the menu with the commands to insert the selected sections to the Project Library.
21. With LC select the Insert Selected Sections command. The IPB 200 section has been added to Project Library.

USER ACTIONS

22. Repeat process 18 –23 for the following sections,
 - IPE 80
 - IPE 180
 - IPE 220
 - KT 40x40x2,6 (Square Hollow Section)
23. The Project Library creating process has been completed. Notice that the arrow next to all section types that have been selected has turned to black. Make sure that <Set as Default> (down on the left) is not checked and proceed to the next task, in the {PROJECT PARAMETERS} dialog box, which is the steel quality parameters check.

PROGRAM FLOW

NOTICE It is understandable that the selection of the Sections is dictated by the nature and function of the structure. As it was pointed in Step 2 it is essential that you have done some preliminary calculations, so that you avoid multiple tests for the selection of the appropriate sections for the structure in question and its load conditions. For this example we have selected a IPB200 section for the column which is more suitable for combined strain in axial load and biaxial pressure, while for the beams are mainly pressed elements (main and secondary) we have chosen IPE80, IPE180 and IPE220 sections. Finally, for the roof braces that act only in tension we have chosen square hollow sections.

PROGRAM FLOW

Steel Quality Parameters Check

USER ACTIONS

24. Chose with LC the {Material Properties} tab. At this point the program allows the modification of the steel quality parameters
25. Without making any changes move on to the Connections parameters

- | | |
|------------------------------------|---|
| Connections Parameter Check | <p>26. Select with LC the {CONNECTIONS} tab. At this point the program allows the modification of the welds and/or bolts that will be later used during nodes design.</p> <p>27. Without making any changes move on to the Foundation parameters</p> |
| Defining the Foundation Parameters | <p>28. Select with LC the { Foundation PARAMETERS} tab. At this point the program allows access to the foundation parameters, which are organized in sub-groups.</p> <p>29. Select from this list the <[SOIL]> sub-group. With two successive LC in every field you can make the necessary changes. Make those changes</p> <ul style="list-style-type: none"> a. On <Moduls of Subgrade Reaction (Ks)> type {40000} b. On <Young Moduls of Soil> type {11000} |

USER ACTIONS

30. Select from the list the <[MODELING]> sub-group and make the following changes:
- a. On <Column Participation Percentage> type {20}
 - b. On < Participation Percentage in Tie Beam > type {60}
 - c. On <Footing type> type {0}
31. Having completed the defining of the essential parameters and after you have made sure that <Set as Default> is not checked, press [OK]. Your program will save your selections at the parameters file of this project.

PROGRAM FLOW

STEP 4 DEFINING THE LOADS AND LOAD COMBINATIONS

On the forth step you will fill in the load combinations table that will be used for the classification of the stress values during the members efficiency test.

Additionally you will separate the given load into member load and nodal load.

Follow the procedure

PROGRAM FLOW	USER ACTIONS
Safety factors selection and Load Combination defining	<p>32. Select with LC the PARAMETERS>LOAD COMBINATIONS command. The Load Combinations table will open.</p> <p>33. On this table, the columns represent the Load Cases while the rows represent the load combinations. Each load combination, to be valid, should include at least one load case. A load case takes part in a load combination if the table cell where the combination with load case in question meet has a value that is not zero. Those values that correspond to the Safety Factors are defined each time by the user depending on the project's special requirements. Generally, they are given by the codes.</p>

- USER ACTIONS**
34. We should point here that the factors values are chosen for this project only and are not typical.
35. Type the values of the safety factors, as seen on Table 2 bellow

PROGRAM FLOW

NOTE: We should point that we fill in the load safety factor according to the regulations only in the first three and eleventh columns. In all other cases the title (e.g. Wind + X) is just indicative and you could use the column for live loads.

	Dead Loads	Seismic Y	Seismic X	Wind +Y	Wind +X	Wind -Y	Wind -X	Snow (1)	Snow (2)	Other Loads	Live Loads
1	1,35										1,50
2	1,35			1,50		0,90					0,90
3	1,35			0,90		1,50					0,90
4	1,35							1,50	0,90		0,90
5	1,35							0,90	1,50		0,90
6	1,35			1,50		0,90		0,90	0,90		0,90
7	1,35			0,90		1,50		0,90	0,90		0,90
8	1,00	1,00	0,30								0,30
9	1,00	0,30	1,00								0,30
10	1,00	1,00	0,30		1,00		1,00				1,00
11	1,00	0,30	1,00		1,00		1,00				1,00

Image 5 Load combinations form and the values of the safety factors for the example project.

PROGRAM FLOW

USER ACTIONS

36. At this point you should check the loads in relation to the load combinations you have just defined. We recommend you to compare the table above to all we have mentioned about load assumptions at STEP 2 and at the "Loads" unit.

37. The loads values that will be defined as distributed along the member, have resulted from the calculation of the weight surface in each member according to its position and angle (e.g. Outside - Inside members with an angle of the axes of the local coordinates system turned towards the load implementation level)

38. LC on [OK], to close the window.

STEP 5 STRUCTURE DEFINING – MODIFICATIONS AND CORRECTIONS.

On the fifth step you will define the elements that comprise the structure and you will modify their properties. The data insertion process is the most time consuming stage of your work. For this reason the program has many alternative ways for the most necessary parts of your work so that the formation of the structure can be simplified. In our case we have tried to describe the most possible ways. The more you use the program the more you will see that depending on the individual requirements of each structure some of the alternative ways for the same work simplify the process more than others. In other words, it is not necessary to repeat to the last word this procedure in your next project.

- Follow the procedure,

PROGRAM FLOW

Columns Insertion

USER ACTIONS

39. If you do not see the "MEMBERS PROPERTIES" toolbar make it visible. The toolbar should look like the on in Image 6

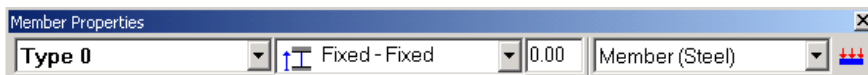



Image 6 "MEMBERS PROPERTIES" toolbar

40. Select the IPB200(Fe360) section from the <Project Library> list
41. Select the <Fixed-Fixed> option from the <Constrains> list
42. Type <90> at the EULER field
43. Select <column> in the <Memmers Groups> list


USER ACTIONS

PROGRAM FLOW

44. LC on the  icon. The {Elements Loads} dialog box appears.
 - a. LC the <Wind +Y> option
 - b. Select the <Local Z> axis as load axis
 - c. Set the <Q> value to <1,5>
 - d. Press [Update] and close the dialog box by pressing [OK]
45. RC anywhere in the project window. On the pop-up menu select WIZARD. You can select the same command from the Edit menu. The {New Member} dialog box will appear.

46. In the <Start Node> dialog box, fill in the bottom node of the column co-ordinates as follows, {1,0}, {1,0}, and {1,0}. In the <End Node> dialog box, fill in the top node of the column co-ordinates as follows, {1,0}, {1,0} and {5,0}.
Press the [Add] button and then press [Close].

USER ACTIONS

47. Click on the  icon again. The {Elements Loads} dialog box appears again. Before you do anything else, uncheck the <Wind + Y> check box.
- LC the <Wind -Y> option
 - Select the <Local Z> axis
 - Set the <Q> value to <1,0>
 - Press [Update] and close the dialog box by pressing [OK]

PROGRAM FLOW

48. Select the EDIT>WIZARD command

Like in the previous column, on the values fields type the following: SN: {1,0}, {7,0}, {1,0} and EN: {1,0}, {7,0}, {5,0}.
Press the [Add] button and then press [Close].

Defining Beams

49. At the {Members Properties} toolbar, make the following changes,
- Select the IPE220(Fe360) section from the <Project Library> list.
 - Select the <Fixed-Fixed> option from the <Constrains> list.
 - Type <0> at the EULER field
 - Select <beam> in the <Memmbers Groups> list

USER ACTIONS

PROGRAM FLOW

50. (The beams do not bear any other load apart from the secondary beams reaction, so we will not define any

load during their defining)
Select EDIT>WIZARD, so that the members "New Element" dialog box.
Uncheck the <Wind - Y> check box.

- a. LC to the [Pick] button. Select node 2 (the upper node of the left column).
 - b. RC anywhere in the drawing window. Select the SHOW DIALOG command.
 - c. In the <End Node> fields, type {1,0}, {4,0}, {6,0}
 - d. LC on <Number of multiple members to insert>. In the field below type {3}.
51. Press the [Add] button and then press [Close]. The program has created a beam and divided it to three equal parts.
52. Select EDIT>WIZARD, so that the "New Element" dialog box appears.
- a. Click on the [Pick] button. Select the nodes indexed 7 and 4.
 - b. At the < Number of multiple members to insert > field type <3>

PROGRAM FLOW

USER ACTIONS

53. Press the [Add] button and then press [Close].

NOTICE If you cannot see the numbering of members or nodes, select the command Parameters>Options>Visible. The dialog box "Visible Items" will appear. Check the options Member Names, Node Names. If the nodes themselves are not visible then you can check the Nodes option

Defining the support property of the bottom column nodes

54. You will now set the start nodes of both columns to support joints. Select the command EDIT>SELECT NODES
55. Press the [Ctrl] key and keep it pressed. Select with LC the nodes 1 and 3 (the bottom nodes of the columns). The selected nodes will change their color.
56. Having selected the two nodes RC and select the SUPPORT command. The two bottom nodes of the columns will be turned into support nodes.

57. Select the command EDIT>UNSELECT NODES, to get off the Select Nodes environment.

Define the second frame

58. You have just completed the definition of the first frame. The second frame will be defined by creating a copy of the first in a distance in the x-x direction. The copy apart from the members and their geometry will also have all information relevant to the loads.

USER ACTIONS

59. Select the EDIT>SELECT MEMBERS WINDOW command

PROGRAM FLOW


60. LC somewhere on the left of above the structure. Release the button and move the cursor to the right and bottom, so that the window includes the entire structure. LC again. All structure members have been selected.

61. Select EDIT>COPY

62. Select EDIT>PASTE. In the appearing dialog box, type <3> to the <dx> field. The Program has copied the first frame with the loads on a distance of 3 m and created the second with its members parallel to the members of the first ones.

63. Select the command EDIT> UNSELECT MEMBERS to get off the Select Members environment


Change The Structure View Point

64. LC on the  icon. LC anywhere near the top node of the structure (the beams meeting point). Release the button and drag slowly the cursor to the right and bottom. Stop when you feel think that the viewpoint is satisfactory.


PROGRAM FLOW

USER ACTIONS

Define Secondary Beams

65. At the {Members Properties} Toolbar, make the following changes,
 - a. Select the IPE 180(Fe360) section from the <Project Library> list.
 - b. Select the <Fixed-Fixed> option from the <Constrains> list.
 - c. Type <18,43> at the EULER field.
 - d. Select <secondary beams> in the <Members Groups> list.
 - e. Loads (LC on the  icon): <Dead>: <Global Z> with <Q> = {0,250} kN/m. <Wind + Y>: <Local Z> with <Q> = {0,70} kN/m. Check the <Snow 1>: <Global Z> with <Q> = {0,05} kN/m. Uncheck all other options.


Member Load Correction


66. LC on the  icon. Make consecutive LC on the following nodes, {2,11}, {5,14}, {6,15}, {7,16}
67. Because the secondary braces in the middle bear a load that is greater than the ones in the sides we will correct the loads.
68. Select the command EDIT>SELECT MEMBERS. While keeping the [Ctrl] key pressed, select members 18, 19, 20.

USER ACTIONS

PROGRAM FLOW

Define the rest of the secondary beams

69. LC on {MEMBERS LOADS} on the {SECTION PROPERTIES} frame (on the right). On the field <Load Cases> select <Dead>. Type {0.5} kN/m in the <Q> field. Select <Wind + Y> and type {1.2} kN/m. Select <Snow 1> and type {0.1} kN/m.
70. Select the command EDIT>UNSELECT MEMBERS
71. At the {Members Properties} toolbar type <-18,43> at the EULER field
72. LC the  icon, to see the {ELEMENTS LOADS} dialog frame again. Uncheck all checked loads. Press [Update] and [OK].

73. Select again the  icon. Make consecutive LC on the following nodes, {8,17}, {9,18}
74. We will define the members loads with the change method.
- Select the command EDIT>SELECT MEMBERS and select members 21, 22.
 - In the frame {SECTION PROPERTIES} make the {Members Loads} tab visible


USER ACTIONS

- Select <Dead> and type {0.5} kN/m at <Q> on the <Global Z> Axis.
- Select <Wind - Y> and type {-0.9} Kn/m at <Q> on the <Local Z> axis.
- Select <Snow 2> and type {0.120} for <Q> on the <Global Z> axis.

75. Select the command EDIT> UNSELECT MEMBERS

PROGRAM FLOW

Secondary Beams Load Correction

76. Select the  icon again. Make consecutive LC on the following nodes, {4,13}
77. We will define the members loads with the change method.
- Select the command EDIT>SELECT MEMBERS and select members 23.
 - In the frame {SECTION PROPERTIES} make the {Members Loads} tab visible.
 - Select <Dead> and type {0.25} kN/m at <Q> on the <Global Z> Axis.
 - Select <Wind - Y> and type {-0.45} kN/m at <Q> on the <Local Z> axis.
 - Select <Snow 2> and type {0.06} for <Q> on the <Global Z> axis.

USER ACTIONS

78. Select the command EDIT>UNSELECT MEMBERS

PROGRAM FLOW

Define loads through filters

79. While the structure definition has been almost completed we have not defined any live loads on the secondary beams. We will define them with the Member Selection through filters method.

80. Select the command EDIT>SELECT USING FILTERS. The dialog box of Image 7 will appear.

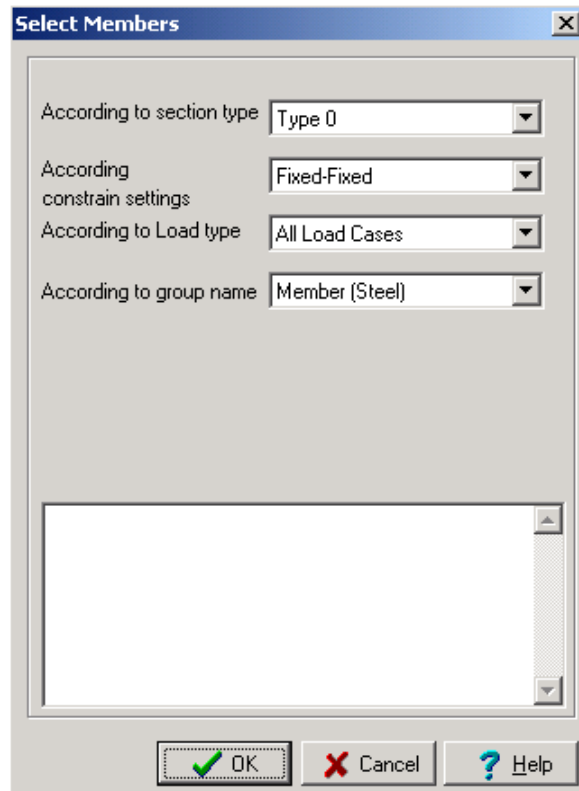


Image 7 "SELECT USING FILTERS " dialog box

PROGRAM FLOW

Define Roof Braces

USER ACTIONS

81. In the field <According to the Group> select <Secondary Beam (Steel)> and press [OK]. The program will automatically select all the members that have been defined as secondary beams.

82. On the {Members Loads} tab select <Live> and type {1.0} kN/m at <Q>.

83. Change this value for the secondary beams to {0.5} kN/m in the way that was described in steps 69/70.

84. At the {Members Properties} toolbar make the following changes,
a. Select the <KT 40x40x2,6 (Fe360)> section from the <Project Library> list
b. Select the <Pinned-Pinned> option from the <Constrains> list.
c. Type <18,43> at the EULER field.
d. Select <bracing> in the <Members Groups> list.

85. Followin the procedure at 52 define the members between the nodes {5,15} and {6,14}
86. At the {Members Properties} toolbar type <-18,43> at the EULER field and repeat the procedure in step 52 for nodes {8,18} and {9,17}
87. You have just completed the model definition process.. Select FILE>SAVE to save your work.

STEP 6 DIAGNOSTIC CHECKS STRUCTURAL ANALYSIS

Having completed the structure definition you should check its integrity and move on to the Space Frame structural analysis. The VK STEEL program provides a set of diagnostic checks to warn you about mistakes with a relevant message.

On STEP 6 we will describe those two actions: Diagnostic checks and Space Frame structural analysis

Follow the procedure

WORK FLOW	USER ACTIONS
Topology Check	88. Select the command ANALYSIS>TOPOLOGY CHECK. The frame of Image 8 will appear

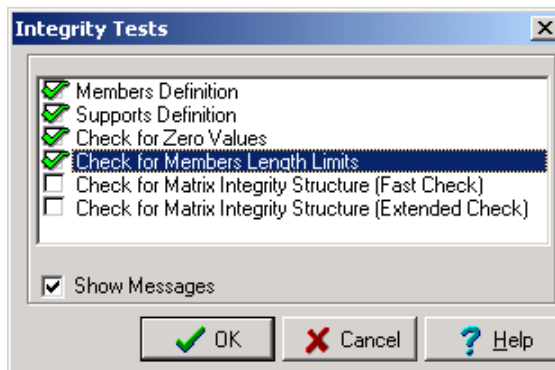


Image 8 "Integrity Tests " dialog box

89. On the dialog box check all options and press [OK]
90. There should not be any errors. If there is one then follow again all steps in STEP 5

WORK FLOW	USER ACTIONS
-----------	--------------

Structural Analysis

91. If nothing went wrong in the Topology Tests then you can proceed to the Space Frame structural analysis. Select the ANALYSIS>STRUCTURAL ANALYSIS command. After a few seconds a dialog frame will appear like the one in Image 9.

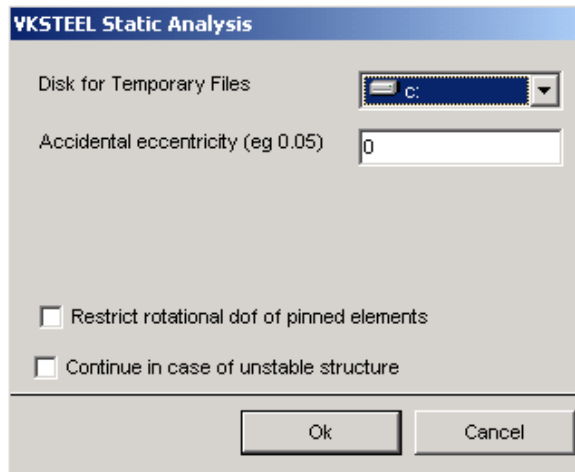


Image 9 "Analisis " dialog box

Analysis Report

92. Make sure that the Analysis drive (disk) is the same with the one you have installed your program and the value of <Accidental eccentricity> is {zero}. Press [OK]
93. When the structural analysis has been completed the analysis report will appear on the screen. It should look like the one in Image 10

Results Test							
L.C.	Sum Fx		Sum Fy		Sum Fz		
	DATAF	ANALYSIS	DATAF	ANALYSIS	DATAF	ANALYSIS	
1	0.00	0.01	0.00	-0.02	-25.44	25.43	
2	0.00	0.01	-8.65	8.56	0.00	0.00	
3	0.00	0.00	0.00	0.00	0.00	0.00	
4	0.00	0.00	0.00	0.00	0.00	0.00	
5	0.00	0.00	0.00	0.00	0.00	0.00	
6	0.00	0.00	0.00	0.00	0.00	0.00	
7	0.00	0.00	0.00	0.00	0.00	0.00	
8	0.00	0.00	0.00	0.00	0.00	0.00	
9	0.00	0.00	0.00	0.00	0.00	0.00	
10	0.00	0.00	0.00	0.00	0.00	0.00	
11	0.00	0.00	0.00	0.00	0.00	0.00	

Ok

Image 10 The "Result Test" Screen

STEP 7 MEMBERS CHECK

After the structural analysis the relevant node displacements, and therefore the member deformations, have been calculated. In other words we have found all stress magnitude everywhere where the program will run the checks.

During the checks procedure the procedure the program factorises the stress values and according to their position (e.g. beams, columns etc) will run all Eurocode 3 required efficiency tests. In case of failure the check stops and the relevant failure message will appear in a dialogue frame, citing the member and failure type.

For this example we will change a part of a beam so that it fails.

Follow the procedure

WORK FLOW	USER ACTIONS
Section change for a beam (for this example only)	94. Select the command EDIT>SELECT MEMBER 95. SELECT beam 3 96. In the "Geometry" tab of the {Section Properties} frame from the Section Type list select <IPE80(Fe360)> and click on the field right bellow. The section of the beam 3 has changed from <IPE(Fe360)> to <IPE80(Fe360)> 97. Repeat the procedure we have described in STEP 6
Members Check	98. Select the command ANALYSIS>MEMBERS CHECK. The dialogue frame in Image 11 will appear

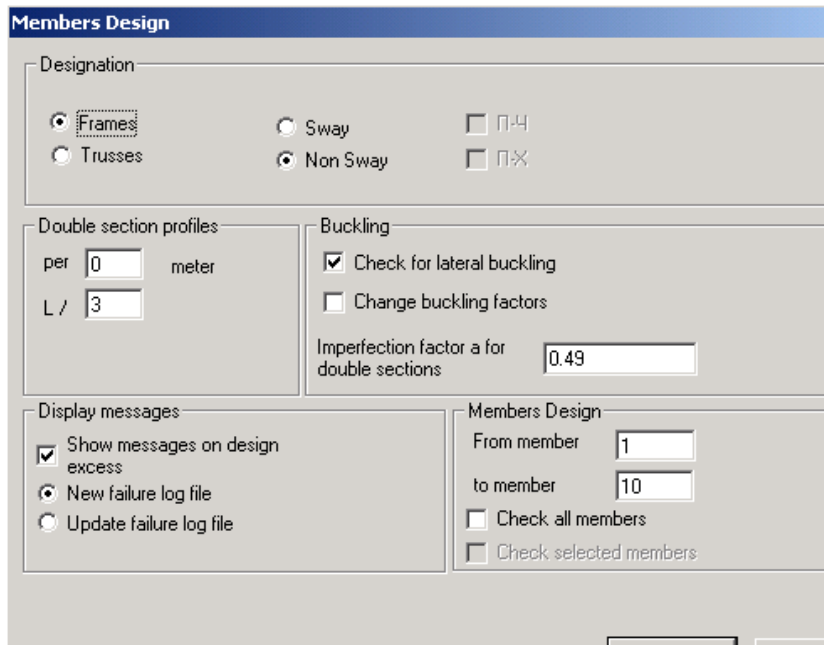


Image 11 The "Members Design" Dialog box

99. Make the following selections,
 - a. On the Designation unity select <Frames> and <Non Sway>
 - b. On the Display Messages unity select <Show messages on design excess> and <New failure log File>
 - c. Make sure the <Check all members> check box is checked
100. Press [CONTINUE]
101. After a while the program will stop and the message of image 12 will appear

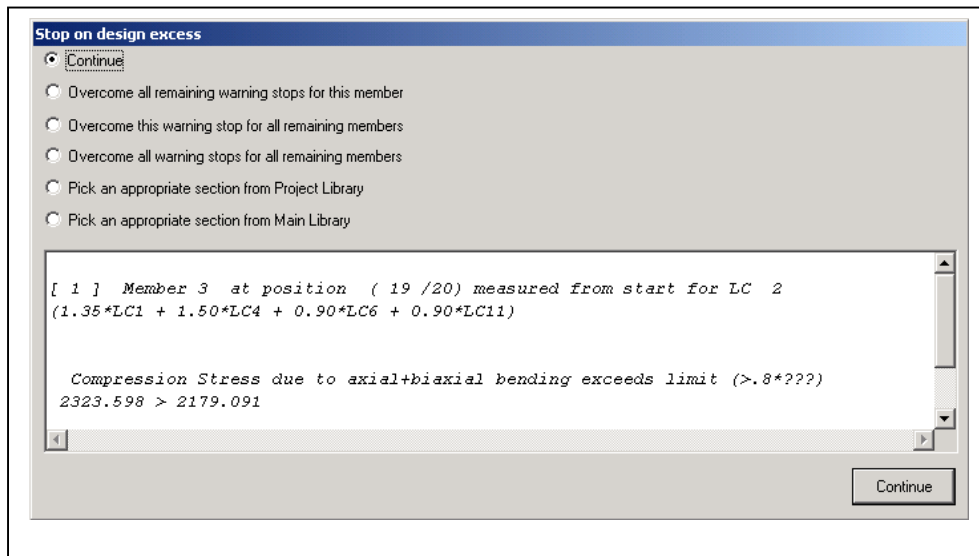


Image 12 The "Stop in Excess" window

102. The message says which member fails, where it failed the load combination during which the message appeared and the reason
103. Press [CONTINUE]
104. This time the message of Image 13 will appear. Notice that while we are still on the same member the failure is due to another reason.
105. This time select <Overcome all warning stops for all remaining members>
106. Press [CONTINUE] again. The program should not stop until the end

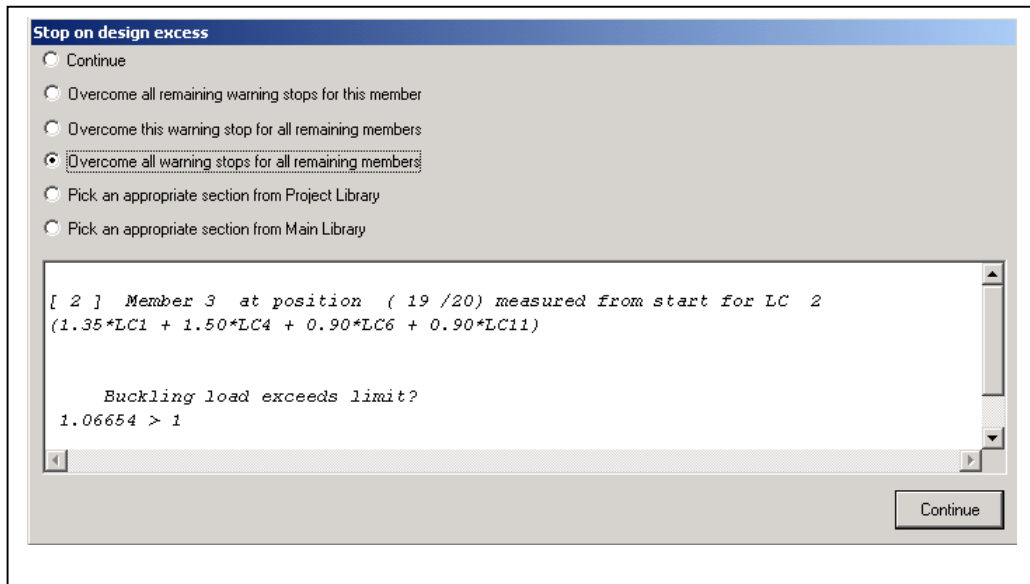


Image 13 The {Stop In Excess} window frame. This time the excess is due to member buckling on the Z-axis, because of Load Combination 2

107. After having selected the [CONTINUE] button, the program will keep on checking all members. Because you have selected <No more interruptions from all diagnosis in all members> the Members Check will not stop even if it meets some excess. This of course, does not mean that the checks do not take place. All excess messages are collected and written in a printable format file, the appearance of which notifies for the check procedure end. The excess message file will look like the one in Image 14

108. Press [CONTINUE]. A new window will appear, the section modification results window. The window displays the contents of the automatic section change file – it should be blank for you, since you did not use that method. Press [CONTINUE] again to complete the procedure.

Correction

109. To restore the beam you have to reset the original beam, as we have previously described and to proceed again to structural analysis and members check.

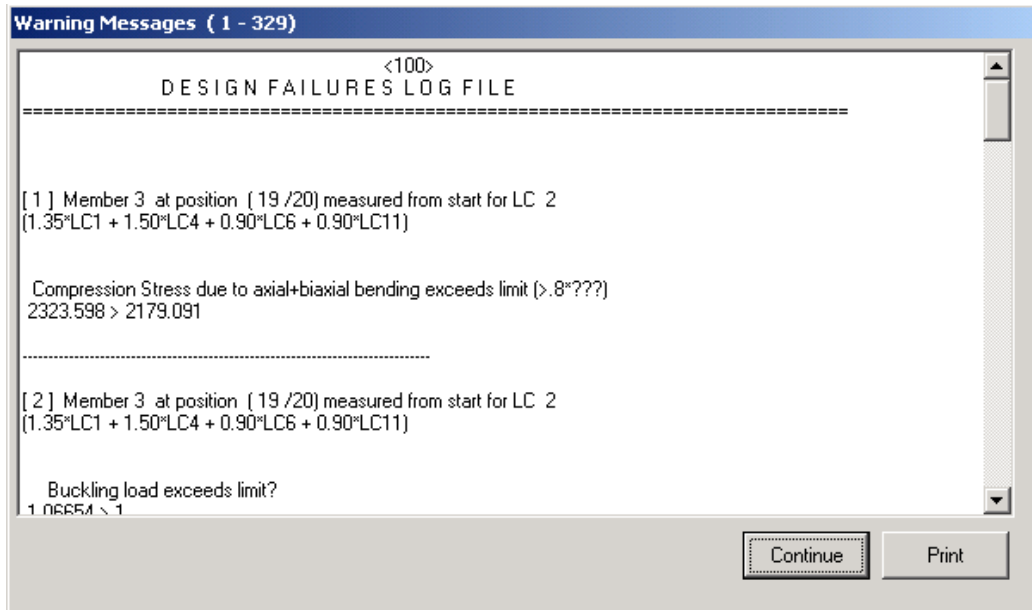


Image 14 The Error Messages file (excess messages) on display window. You can print the file by selecting the [PRINT] button.