Volume

1

CIVILDESIGN INC.

VisualDesign Software– Version 5.9

April 2006

Complete User's Manual

VISUALDESIGN™ SOFTWARE

Disclaimer

The software described in this document is furnished under a license agreement or nondisclosure agreement. The software may be used or copied only in accordance with the terms of those agreements. Although we have taken all precautionary steps to ensure the reliability of VisualDesign™ software, as well as the accuracy of data given in this manual, it must be understood that neither the authors, nor CivilDesign, nor distributors can be held responsible, in any way whatsoever, for inaccurate or improper use of the material. Users must explicitly understand the assumptions of the program and must independently verify the results.

Any comment about the use of **VisualDesign™** software or
about the present documentation shall be sent to

CivilDesign inc.
61, St. Charles St. W., Suite 50
Longueuil (Québec)
J4H 1C5
Phone: (450) 674-0657
Tool Free Number: 1-800-724-5678
Fax: (450) 674-0665
or
E-mail: support@civild.com

www.civild.com

Chapter

1

BASIC PRINCIPLES

TABLE OF CONTENTS

Chapter 1 Basic Principles

VisualDesign

Overview

VisualDesign™ is a 3D Structural Analysis & Design software that allows you to analyze and design all types of structures. It includes finite elements, floors, plates, and composite sections. Automatic generation of 3D mesh, trusses, ice and wind loads and load combinations is also available. The program can support an unlimited number of nodes, loads, load combinations, and envelopes.

The program is user-friendly developed on a Windows interface. You will find it easy to use, even for the first time. It has a Multi Document Interface (MDI) that allows the opening of several projects simultaneously from the same application. You may even display different documents on the screen at the same time. To choose a specific document, click on its title bar or select it from the Window menu.

VisualDesign™ provides a unique cyclic **Steel Design** using results of Dynamic, Spectral, Time History, Moving Loads and Non-linear Static analyses prior to selecting shapes and prior to starting the Foundation Design. The design process is iterative. Furthermore, American, Canadian, or Parametric standards are supported in this program. Users may easily switch from Imperial to Metric system of units.

Results may be obtained graphically (forces, diagrams, and even animation of vibration modes) or by the way of printable spreadsheets. P-Delta effects are included in the static analysis, as well as inelastic and torsional effects in **Spectral** analysis. The **Dynamic** analysis offers an integrated database of 100 accelerograms. **Non-linear Time History** analysis includes elastoplastic hysteretic models, hysteretic loops, friction dampers, plastic hinge, and displacement, support and stress time history graphs.

The **Moving Loads** analysis provides a real 3D analysis including editable scenarios for trucks, pre-defined and/or user-defined moving loads, envelopes for Finite Elements, members and reactions and associated min/max values.

The **Foundation Design** module includes the design of shallow and deep foundations. Reinforcement is supplied in the footing design results. It can be used with the static analysis only. However, for those owning any **Design Module**, the cyclic design will also include the foundation design results in the design process.

The **Reinforced Concrete Design** module and **Prestressed Concrete** module are also available. The first one designs and places rebars in beams, columns, shear walls and 2-way slabs. Resistance and forces diagrams for beams, columns and shear walls are displayed above the elevation view of a continuous system. This drawing shows all rebar placement details. You are allowed to edit (move, delete, stretch) rebars on the displayed drawing and check the resistance diagrams which are automatically recalculated as you edit rebars! The **Prestressed Concrete** module designs concrete elements and checks the capacity of prestressed concrete beams according to the prestressing cables that you placed into the beam with the help of a tool called "Cable Layout Models". It also includes several graphical results, loss of prestress in cables and stresses results for each construction steps.

VisualDesign™ is multilingual (English, Spanish, and French). You can switch from one language to another, simply by clicking on "Display Language" in the **File** menu.

This program allows interfacing with AutoCAD through DXF format. The **Reinforced Concrete Design Module** and **Prestressed Concrete Module** allow you to export rebar placement details in element elevation view through the **Export DXF** function of **File** menu.

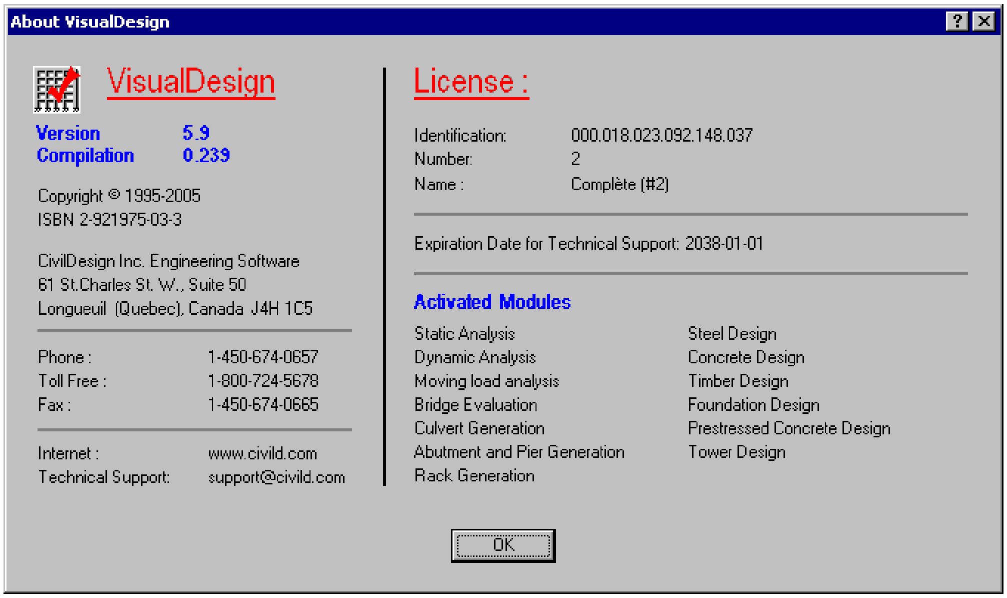
When you need assistance or want more information, simply click the **Help** menu or point the arrow on one of the menu functions or buttons on the toolbar. You can also press the **F1** key to learn more about posted dialog boxes or spreadsheets.

Available Modules:

Static Analysis (Base module)
Dynamic Analysis Module
Foundation Design Module
Moving Loads Analysis Module
Steel Design Module
Reinforced Concrete Design Module
Prestressed Concrete Module
Bridge Evaluation Module
Tower Design Module
Generation of Culverts
Generation of Abutments, Piers & Retaining Walls
Timber Design Module

About VisualDesign™

Select **About VisualDesign** from **Help** menu to know the version and compilation number and activated modules for the license you are using. The expiration date of your maintenance agreement is supplied and it gives information about technical support.



Version and Compilation

On-line Help: Version 5.9, compilation 240.

To know the version and compilation you are currently using, select **About VisualDesign** in the **Help** menu.

See also

About VisualDesign
Updating VisualDesign
About the protection key
Technical Support

Updating VisualDesign

If your maintenance agreement is still active, you have the right to download VisualDesign latest compilation (update) through our Web site www.civild.com. Make sure that you have the name of your Client directory and password. Ask your network administrator about it. Should you have any question, please contact us at this address: installation@civild.com.

Maintenance Agreement

To renew your maintenance agreement, add new licenses or modify existing ones, please contact us at the following address: licence@civild.com

Minimum Equipment Required

- Pentium 100MHz or higher
- 32MB RAM
- 100MB available disk space
- SVGA screen with a minimum resolution of 800 x 600 with 65536 colours

VisualDesign's Limitations and Capacity

VisualDesign™ analyzes structures that induce small deflections and average displacements.

There is no limitation on the size of the project. It only depends on the computer available memory (RAM and Hard disk).

See also

Analysis and Cyclic Design Limitations
Limitations of the Reinforced Concrete Design module
Limitations of the Foundation Design module
Bolted Connection Design Limitations

VisualDesign's Modules and Integrated Standards and Codes

Modules	Canadian Standards	American Standards
Base Module (Static Analysis)	N/a	N/a
Dynamic analysis (spectrum)	CNBC 1995 CNBC 2005 CSA/CAN S6-00	UBC 1994 UBC 1997
Foundation Design	CNBC 1995 CFEM 1992 CSA/CAN A23.3-95 CSA/CAN S6-00	US Army Corps of Engineers 1994
Steel Design	CSA/CAN S16-01 CSA/CAN S6-00	ASCE-7 SD ASCE-7 ASD AASHTO LRFD-98

Modules	Canadian Standards	American Standards
Moving Load Analysis	CSA/CAN S6-00 CSA/CAN S6-88 Customizable	AASHTO LRFD-98
Reinforced Concrete Design	CSA/CAN A23.3-95 CSA/CAN-A23.3-04 CAN/CSA-S806-02 (Fibre reinforced polymer bars) CSA/CAN S6-00	AASHTO LRFD-98
Prestressed Concrete	CSA/CAN S6-00 CSA/CAN A23.3-95	AASHTO LRFD-98
Bridge Evaluation	CSA/CAN S6-00	
Tower Design	CSA/CAN S37-01	ASCE-10-97*
Timber Design	CSA/CAN O86-01	

* Refer to **ASCE-10-97** tab of **Project Configuration** dialog box.

** *AASHTO LRFD Bridge Design Specifications* (1998): For the moment, shrinkage and creep models are based on Code S6-00 (equivalent to CEB-78). Other models will be added in a later version.

Starting VisualDesign™

Double-click on VisualDesign™ icon or start automatically by double-clicking on any VisualDesign™ application (.vd1 or .vdz extensions).

About the Protection Key

You may install VisualDesign™ on more than one workstation. However, in order to work with the software, you must also install a protection key.

If you use the same key on more than one workstation, make sure that you shut down the computer before removing or installing the key. The key may be damaged if you transfer it from one station to another without first shutting them down.

Remember that the authorization numbers of a license and its protection key are interdependent.

Using VisualDesign

This topic includes ten essential steps that you must well understand before beginning a project with VisualDesign.

1. Project Configuration

Select this dialog box in the **File** menu. According to the type of analysis that you are planning to do, complete the appropriate tab and modify the default parameters, if needed.

Always consult the **Preferences** tab and the **Analysis** tab.

See Project Configuration

2. Activation Modes and Properties Function

With VisualDesign™ you may work under seven activation modes, namely, "Structure", "Load Case", "Load Combination", "Envelope", "Vibration Mode", "Design Results" and "Rebar Placement" mode. Before asking information about any specific element through the **Properties** function of the **Edit** menu, make sure that you have activated the right mode, since the displayed dialog box is specific to an activation mode.

For example, if you click twice on a node while you are in the "Structure" activation mode, you will get the **Node Characteristics** dialog box. If you click twice on the same node while you are working with the "Load Case" activation mode, you will get the **Loads on Node** dialog box. If you proceed the same way in the "Load Combination" activation mode, you will get the **Nodes Displacements** dialog box. Finally, if you double click on a member while in the "Envelope" activation mode, you will get the envelope diagram for this member.

See Activation Modes

3a. Selection Mode

VisualDesign™ allows one of two selection window modes: the **Extended Window** and the **Restricted Window**. By default, one of these modes is activated when the "Add" mode is not. Both windows allow you to select an element or group of elements. The entire element is selected whichever mode you used.

The difference between the two selection windows appears when you use the pointer to plot a window around an element or a group of elements.

Whichever mode you chose, you must always specify the type (member, node, plate etc.) of element that you want to select. The list of available elements appears both in the **Edit** Menu, under the "Elements" heading, and on the Elements toolbar.

See also

Extended Window

Restricted Window

3b. Multiple Selection

We call it a multiple selection when many elements are selected. To do a multiple selection of elements of the same type, press down the **Ctrl** key while you click on elements. Then, press the **Properties** icon. You will notice that some fields are blank in the dialog box. Values that will be entered in blank fields will be applied to all selected elements.

To de-select elements in a multiple selection, just click on the highlighted element that you wish to de-select.

3c. Mixed Selections and the Mask Function

A mixed selection is made up of elements of different types (members, floors, plates…). This type of selection is very useful to isolate a part of a structure. You will be allowed to select a bay or a story with its selected members, plates, floors, etc.

To do such a selection, activate a type of element on Elements toolbar, select the **Restricted** window and draw a window around the part of the structure that you want to isolate. Elements of this type will be selected. Then, activate another type of element on Elements toolbar and draw another window to select the other type of elements. Repeat this process to select other type of elements. Click on the **Mask** function.

3d. Correspondence between Selected Elements on your Screen and Spreadsheet Contents

If you select a spreadsheet and there are selected elements on your screen, the spreadsheet will only include the selected elements. If no elements are selected, the spreadsheet will include all elements in your project.

This function applies to all activation modes except "Vibration Mode".

4. View Options

Use VisualDesign view options (**View** menu) because they are essential when modeling a structure and to consult results.

See View Options

5. 3D Modeling (By default in VisualDesign)

For a 2D model, you must put a restraint on Mx (torsion) to at least one or two support nodes, depending on the type of structure. Otherwise, the model will be unstable and you will get a warning message that there is a null pivot in the stiffness matrix.

See also

Null Pivot in the Stiffness Matrix

How to model a structure
Modeling Strategy
Practical Example - 2D Frame

6. Definition of Load Case Titles

VisualDesign must know the type of loads that are going to be applied to the structure to accurately generate load combinations and load factors according to the building code.

The spreadsheet **Loads Definition** is located in **Loads** menu at heading **Load cases** / **Definition**. The purpose of this spreadsheet is to define the title of each load case and to select the type of load because the Load Combination Generation Wizard needs to know which type of load applies.

Once in the spreadsheet, you must insert the number of lines that correspond to the number of loads that you want to apply on the structure such as live, snow, wind, additional dead load, dynamic, etc. The standard dead load is automatically calculated and inserted in the spreadsheet. It cannot be removed.

For each load, specify its type by double-clicking in the "Type" column. Choose among the drop-down list box: Dead [1] Standard, Dead [2] Wearing surface, Dead [3] Pouring of Concrete, Dead [4]-Backfill, Dead [5]-Hydrostatic, Dead [6]-Passive, Dead [7]-Guy, Live, Snow, Wind, Seismic, Interaction, Dynamic, Temperature, Load on roof (LRFD), Rain (LRFD), Auto ice and Auto wind.

See Load Definitions Spreadsheet

7a. Using Load Combination Generation Wizard

Use this tool (**Loads** menu, heading **Load Combinations**) to automatically generate all load combinations according to the selected code. You can also include different envelopes such as spectral (**E**), moving load (**Lmi**), linear time history (**Et**) and non-linear time history (**Etnl**) envelopes. Use this tool to generate resistance and deflection envelopes also.

You are allowed to disable load combinations if you do not want to generate them.

When that the generation is completed, the **Load Combinations** spreadsheet is automatically open. Look at the load combinations statuses. Disable the option "Required" for load combinations that you do not want to analyse right now. Look at load factors in the **Load Factors** tab.

See Load Combination Generation Wizard

7b. Definition of Load Combinations and Load Factors

If you do not want to use the **Load Combination Generation Wizard,** you must select and complete the **Load Combinations** spreadsheet. It is located in the **Loads** menu at heading **Load Combinations**/ **Definition**.

The purpose of the first tab is to give a title to each load combination and to choose its status. The load combination status is very important. The "Stage" column is used for prestressed concrete and composite beam that have construction stages.

The second tab, titled **Load Factors**, is very important and is often forgotten by new users. Insert lines in the right part of this dialog box and select all load cases that are part of each load combination. To do that, you must highlight the load combination in the left part of the dialog box and then, in the right part, insert the number of lines corresponding to the number of loads that is part of the load combination. At each inserted line, you must double-click in the "Load Cases" column and choose a type of load in the drop-down list box. Also, enter the load factors that you want to apply to each load. Available loads are those defined in the **Load definition** spreadsheet. You must do that for each load combination.

See Load Combination Definition spreadsheet and Load Factors

8. Apply Loads on the Structure

You must activate the Load Case mode on the Activation toolbar. Then, select a load title in the drop-down list box. After, go to Elements toolbar and activate the appropriate type of element that you want to load. Select those elements on the screen and click on the **Properties** icon. The **Load** dialog box will appear on the screen. Enter loads.

Select another load title in the drop-down list box and repeat the process.

If you want to modify the loads in a quick manner, select the corresponding load spreadsheet in the **Loads/Load cases** menu, select the whole column or a part of it and right click to call up the contextual menu. Choose one of available functions. Do not forget that spreadsheet content corresponds to selected elements on screen.

9. Analysis

Refer to topic Analysis Procedures (**Help ?** menu) to guide you through all the required steps. Procedures are available for all types of analyses.

10a. Numerical Results:

According to the type of results that you want to look at, activate the appropriate activation mode (Load Combination, Envelope, Vibration mode, or Rebar Placement) and choose a title in the drop-down list box. Then, double-click on an element or, if you have selected many, click on the **Properties** icon. A results spreadsheet will automatically appear on your screen. The "Design Results" mode gives you direct access to this spreadsheet.

You can also select results spreadsheets through the **Results** menu, at headings **Load Combinations** and **Envelopes**.

10b. Graphical Results:

Select the **Results** tab of **View Options** and select the diagram (forces, resistances or deflection) that you want to look at. Activate the legend for results.

If you ran a dynamic and spectral analysis, activate the "Vibration mode" on Activation toolbar and choose a vibration mode in the drop-down list box. You will see the deflection of the whole structure under this mode. To animate this vibration mode, select the **Animation** function.

See Animation

10c. Design Results:

If you ran a steel design, the "Design Results" activation mode will be automatically activated at the end of the design. Click on the **Properties** icon to call up the **Design Results** spreadsheet. In the lower part of the spreadsheet, three buttons are available to help you look at results. First, select a line (which corresponds to a member) in the spreadsheet and click on one of the button: a print preview of the member Design brief, the printing of this Design brief or call up the forces and deflections spreadsheet.

See also
Design Results Spreadsheet
Design Brief

Technical Support

Technical Support

Contact us if you have any question about the utilization of the program, or its installation procedure and upgrade, and feel free to send us your commentaries.

We are open from 8h30am to 5hpm, Monday to Friday.

Use the following e-mail address:

Technical Support: support@civild.com

Installation Problems: installation@civild.com

Sales: licence@civild.com

Using Help

Online Help is a reference tool that gives you a systematic protocol. You can ask for help by using one of the following procedures:

- By selecting the **Online Help** command from **Help** menu.
- By selecting the Help icon  from the Standard toolbar and by clicking on any sub-menu included in any VisualDesign™ menus. The Online Help will automatically opened at the item that you selected to give you the information you asked for.
- By pressing the **F1** key. This function will opened the online help at the topic corresponding to the dialog box, spreadsheet or contextual menu you were in when you pressed down this key. This function is the mostly used by users.

The Online Help table of contents appears at the left part of the window once that the **Online Help** dialog box is opened. It includes all online help topics that you may find through book chapters. Each chapter corresponds to a module.

To make your search quicker, select the **Index** tab and enter a key word in the appropriate field.

See also
Displaying a Help Window
Help Command of Standard Toolbar
Online Help Command from Help menu
Printing a Help Section

Frequently Asked Questions (FAQ)

Q1: "Yesterday, I applied loads on my structural model but this morning, when I opened VisualDesign, I could not see them."

A. If loads were applied on floors, you will not see them unless you activate the floor outline in the View tab of View Options dialog box.

Q2: "After a spectral analysis, I selected the Seismic Directions spreadsheet and the value of "V" is 0.00. Why?"

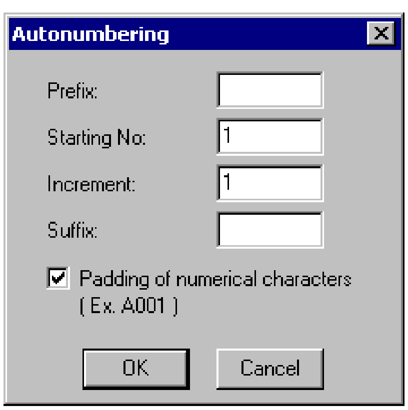
A. The value of V corresponds to the chosen minimum lateral seismic force that is used to calibrate forces according to Vmin, if calibration is required (Seismic tab of Project Configuration). When calibration is not required, VisualDesign does not display the value of V.

Q3: "When analyzing my structure, a warning message appears in the Analysis dialog box: "Warning! Load combination DL1 has not reached a desirable level of precision!" What does it means?"

A. This message appears during a non-linear analysis when the last iteration cycle is reached and when the variation of member axial force exceeds the convergence criterion as specified in the **Analysis** tab of **Project Configuration**. If it happens, increase the number of iterations or maximum variation on axial force.

Q4: "I did a multiple selection of floors and did some modifications but it seems that not all the selected floors have been modified. What is wrong? "

Sometimes, two floors have the same number (same name) because one of them has been copied from another project or else. When selecting multiple floors for a loading, only one of these two floors will be selected because they have the same number. To solve this problem, call up the Floors spreadsheet and select the "Number" column. Re-number all floor numbers (to make sure they are not used twice) by right clicking and choosing the **Auto numbering** function in the spreadsheet' contextual menu. Always verify that members, plates, nodes, etc. numbers are not doubled.



Q5: "What are the types of extension files created by VisualDesign?"

See Type of Files generated by VisualDesign

Q6: "I cannot print my results. Why?"

A. Users that have printing problems with a HP-LaserJet printer must make sure that it is not using a PCL6 driver because it can cause a failure in the system. To solve this problem, go to Windows *Start* menu and select *Settings* and then, *Printer*. Right click and choose function *Properties*. Change the PCL6 driver by another one.

Q7: "The program failed at its opening."

A. Many reasons can explain this. The first thing is to consult the xxx.log file located in directory C:\VisualDesign\User. xxx represents the end of network card physical address, or key number, if you have one.

This file will help to locate the problem. The most frequent causes are:

- One of these files or directories are read only:
  VDBase.mdb in C:\VisualDesign\Sections
  Licence.mdb in C:\VisualDesign\User
  Commun.mdb in C:\VisualDesign\Commun
- Some DLL files do not come from the right version of the software.
- There is an error in the registers.

Q8: " VisualDesign displays an error message following the registration of my workstation."

A. Following the registration of a workstation, the *licence.mdb* file must be validated. You must e-mail us the file to this address: licence@civild.com. As long as this validation is not done at our office, the software will not work on this new workstation and will display the error message.

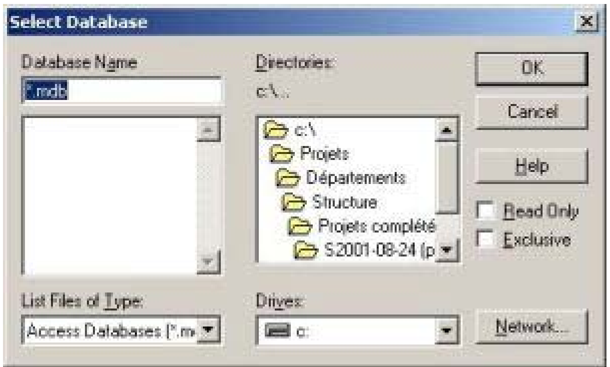
Q9: " Following an upgrade of a registered installation, the program asked me to enter an authorization number for the Evaluation copy."

A. The installation of a registered version is done in two steps:

- The execution of the **SetupVDxx-yyy.exe** file (about 20Mo)
- The replacement of your customized *licence.mdb* file by the new one (VDxx-yyy.exe file is smaller than 100ko).

If you tried to open VisualDesign without having gone through the second step, the program will enter in the Evaluation mode, by default.

Q10: "At the opening of the program, a "Select Database" dialog box was displayed on the screen. What should I do?"



A. In this case, it is very likely that the path to the file (including the name of file) is too long (more than 128 characters). You can do one of the operations describe below:

- Move your project in a smaller path;
- Rename your project with a shorter name;
- Use Windows command SUBST to assign a disk letter to a particular directory.

Example using the command SUBST:

To assign the disk letter "S" to directory "C:\Projects \ Departments\ Structure \Completed Projects\S2001-08-24 (joists)\ Modelling \VisualDesignVersion52-184":

- Go to Windows *Start* menu, click the Run command and enter SUBST S: " C:\Projects\Departments\Structure\Completed Projects\S2001-08-24 (joists)\Modelling\VisualDesignVersion52-184" (Quotation marks are important in this case because of blanks.).
- You will have access to the files included in this directory by selecting the disk (directory) S:FileName.vd1

Customized Options

CivilDesign is happy to put its programming expertise, intellectual abilities and innovative approach at your service by creating customized applications to optimize your operations and increase the reliability of results.

We can develop an automated customized structure just for you to make your design quicker than ever! We can also upgrade and improve your house made programs into Windows environment!

Save time and money: Contact CivilDesign today!

Phone: (450) 674-0657

Toll Free 1-800-724-5678

E-mail: custom@civild.com

Tips

Null Pivot in the Stiffness Matrix

During an analysis, this message indicates that a null pivot was detected in the stiffness matrix. It is a strong indication that there is an instability problem within the model. The node number that is given with the message is a hint to locate the instability source.

Don't forget that VisualDesign™ is a 3D software by default. If you modeled a 2D structure, you must restraint the third direction and the rotation degree of 2D members to avoid the rotation around itself.

A 3D structure that is stable (having analysed it without any problem) may become unstable if lateral forces are applied and if you ran a non-linear analysis. It means that instability was not detected under gravity loads. Consequently, we recommend that you always check the structure behaviour under lateral loads.

Always check the structure deflection, if possible. An extreme displacement is an important indication that a mechanism is present in the structure. Strengthen the structure where the mechanism took place.

If you have gone through many tries but haven't resolve the problem, here are a couple of suggestions to make sure that the model is stable:

- Create a unit load case on each node of the model. Loads should be applied in the three orthogonal directions (x, y, z, Mx, My and Mz).
    - Create a load combination that corresponds to this load case.
    - Configure your project for a static analysis (**File** / **Project Configuration** / Analysis tab / "Type of analysis").
    - Run a static analysis. If you obtain the null pivot message, the model is still unstable.
    - Observe the model deflections to detect instabilities that could be present. Huge deflections are a sign of instability, inferior structural stiffness or excessive loads.
- If there are some Built-Up sections in your project, change all these sections for symmetrical shapes or pre-defined shapes such as HSS, to make sure that instability is not caused by data entry.

Correspondence between a Graphical Selection of Elements and Spreadsheet's Contents

There is a correspondence between selected elements and spreadsheets' contents.

Example: Graphically select a few members on the screen and call up the **Members** spreadsheet in the **Structure** menu. The members included in the spreadsheets are those you selected on the screen. Now, select a few lines in the spreadsheet and press OK. The highlighted members on your screen are those you selected in the spreadsheet.

Steel Design with no Loading

If no loading is defined and a steel design is carried on, the design results will be "Not evaluated" for each member.

"Mask" Function

When using the **Mask** function, only the selected elements are displayed. If you masked a structure after having selected floors only, members will not be displayed. You need to select floors AND members in the same part of the structure before using the **Mask** function. (Use the [Ctrl] key to add or withdraw objects in your selection).

Mask a Part of the Structure to Have a Better Look at It

Functions "Mask" and "Unmask" are very useful to cut off a part of the structure.

Example: To look at results of a 3D frame, choose a plane view, select the frame members using the **Restricted** window  then, press down the "Mask" icon . Go to a perpendicular view to the frame plane. Press down the "Unmask" icon  to display the whole structure again.

Selection of Elements in a 2D Model

In a 3D structure, it is often more secure to select elements in a 3D view instead of a 2D view. The following example shows why.

Figures A and B represent the same structure but in different views. The first one is in the XY plane and the second is the isometric view. With the mouse, select member m1 in a 2D view (figure A). Look at figure B: we chose bracing c1! The same thing can happen when selecting other elements (nodes, floors and plates).

FIGURE A

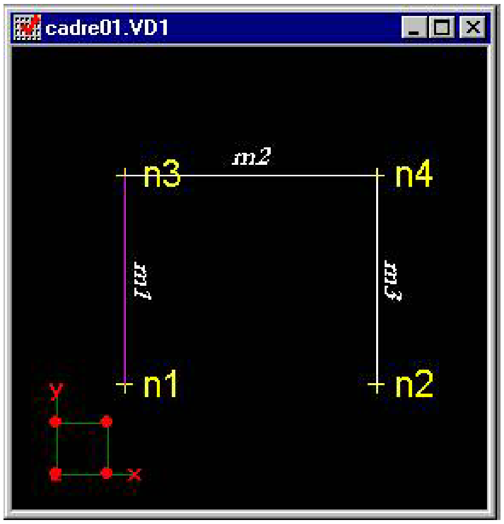
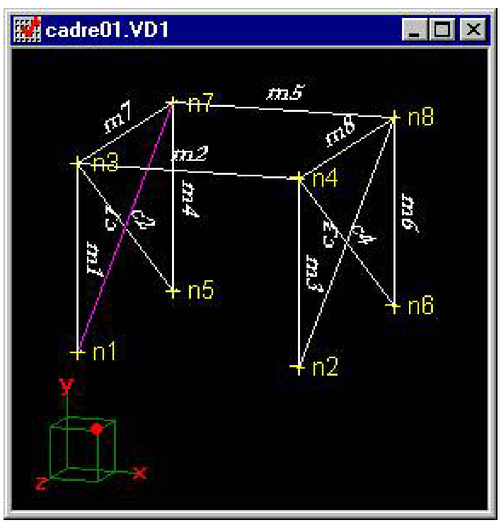


FIGURE B



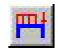
Addition or Removal of Elements in a Selected Group of Elements on Screen

Use the keyboard [Ctrl] key to add or to withdraw one or more elements from the selection. Ex.: Select a member, and while you press down the [Ctrl] key, select other members to add them to the selection.

Applying Common Modifications to a Group of Selected Elements

If you want to quickly apply common modifications to many elements, select them on the screen and press down icon Properties . The **Characteristics** dialog box (Structure mode) or **Loads** dialog box (Load Case mode) will appear. Enter common parameters in the blank fields and press OK.

Graphically Delete a Load Applied on an Element

The [Delete] key is useful to delete a load applied on an element or on part of the structure. Only the load is deleted, not the element. However, the "Load Case" activation mode  must be activated beforehand.

Analysis with Cable Elements (Pre-tensioned Members)

To analyse cables (pre-tensioned members), make sure that they are properly split up because the non-linear calculation can create a loss in the numerical precision. It may cause discontinuities in the cable deflection diagram it has not been split up the right way.

Eliminate Window Animation, Menus, and Lists on Win 98

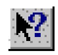
Users with Windows 98 will increase the performance of VisualDesign when analyzing, if they eliminate window animation, menus and lists. This animation option appeared with Windows 98 and it is harmful to the display of Analysis status in the Analysis dialog box.

To eliminate window animation, go to Windows 98 Start menu, click on Settings, Control Panel, Display and Effects tab. Uncheck the box "Animate window, menus and lists".

Using On-line Help

Using Help

Online Help is a reference tool that gives you a systematic protocol. You can ask for help by using one of the following procedures:

- By selecting the **Online Help** command from **Help** menu.
- By selecting the Help icon  from the Standard toolbar and by clicking on any sub-menu included in any VisualDesign™ menus. The Online Help will automatically opened at the item that you selected to give you the information you asked for.
- By pressing the **F1** key. This function will opened the online help at the topic corresponding to the dialog box, spreadsheet or contextual menu you were in when you pressed down this key. This function is the mostly used by users.

The Online Help table of contents appears at the left part of the window once that the **Online Help** dialog box is opened. It includes all online help topics that you may find through book chapters. Each chapter corresponds to a module.

To make your search quicker, select the **Index** tab and enter a key word in the appropriate field.

See also

Displaying a Help Window

Help Command of Standard Toolbar

Online Help Command from Help menu

Printing a Help Section

Online Help Command from Help menu

- To access to On-line Help, select **On-Line Help** from **Help** menu. You will then have access to help summary.
- To have access to one of the summary sections, click once on this section.
- To have access to keywords, select the **Index** tab.

For more information on the way to surf on On-line help, select the **Using Help** command from the **Help** menu.

Contextual Help of Standard Toolbar

The "Contextual Help" button of Standard toolbar

Use the **Help** command to get help on any section of VisualDesign™. When you select the "Help" button from the Standard toolbar, the pointer appears as an arrow and a question mark. Click anywhere in the VisualDesign™ window, for example on another button of the toolbar. The help section for the element on which you have clicked will then appear.

Analyses Procedures

Analyses procedures (step by step operations) are available for all type of analyses that can be done with VisualDesign. Reach quickly these procedures by selecting the command **Analyses Procedures** in VisualDesign **Help** menu.

Printing a Help Window

To print a help section, select **File Print**. To print a help procedure, select "Print" from the help procedure window.

Display a Help Window while in Working Session

There is two types of help windows: the main window and the procedure window. The main window displays general information, while the procedure window shows the systematic procedure in order to carry out each function

- To keep the Main help window on screen while in a working session:
  - Select "Help always Visible" from "Options" menu of Help window.
- To keep the Procedure help window on screen while in a working session:
  - Select "Visible".

Should the help window be in your way while in working session, click inside the document. The help window will go into the background. To recall the main help window to the foreground, push [Alt]+[Tab] keys.

Databases and Interface

Interface

VisualDesign™ has Multi Document Interface (MDI) capabilities, which allows working on several projects simultaneously from the same application. Each project will have its own window.

To work on a specific document, click on its title bar or select it from the **Window** menu. Do not click in the document window because it might lead to the selection or de-selection of an element.

VisualDesign Databases

VDBase.mdb: Shapes, Materials, Bolts, Reinforcement, Cables, Studs, Steel decks, and Soils data.

Mobiles.mdb: Mobiles (Trucks)

Seismes.mdb: Accelerograms

See also
Merge to current database
Change current database
VDBase.mdb Database
Saving Styles and Colours
Archiving Common Objects used in your Project within your .vd1 file

Commum.mdb Database

All information concerning a project configuration, except for data contained in *Commun.mdb* database, create a relational database saved in a *Project.vd1 or .vdz* file.

The following tables are contained in the *Commun.mdb* file:

- Table for physical attributes of line types (colour, style, thickness) and symbols (amplification factor, filling option);
- Table of items included in list boxes (combo box);
- Table for unit factors;
- Table for font selection to print numbers;
- Table including texts for messages with different identification numbers (ID);
- Table for spreadsheets;

- Table of menus;
- Table of displayed language.

ID Numbers

ID numbers of personalized objects (Common menu) are random and superior to 100000. This prevents ID conflict when importing another database.

Saving Styles and Colours

Styles and colours are saved in *Commun.mdb* file. So are messages and units. Styles and colours can be modified using the **View Options** command from the **View** menu.

You must also know that if you modify the colour of elements or certain units while working on a network, you will pass on this modification to all other users.

Furthermore, all styles or colours modifications are automatically registered in the database even if you should decide not to save such modifications.

See also

Database Commun.mdb

VDBase.mdb Database

VisualDesign **VDBase.mdb** database includes all the objects that are composing the **Common** menu: Shapes, Materials, Bolts, Reinforcement, Cables, Studs, Steel decks, and Soils data.

These pre-defined data cannot be edited but users can create their own by inserting lines at the end of spreadsheets.

Users are allowed to choose another database at the opening of a VisualDesign project or to merge the current database with another one. Go to **File** menu in VisualDesign *Start* window and select either to merge the current VDBase.mdb database with another VDBase.mdb database or to change the current database by another one.

See also

Change or Merge the current database with another
Database Commum.mdb
Archiving common objects within your .vd1 or .vdz file
Importing Common Objects to your Database

Archiving Common Objects within your .vd1 File

All objects that are included in VDBase.mdb (shapes, materials and soils), Mobiles.mdb and Seisme.mdb databases are automatically saved, by default, within your project (.vd1 file).

This option can be disabled in the **Preferences** tab of **Project Configuration**, at section "Archiving Common Objects".

Importing Common Objects to your Database

When you open a .vd1 or .vdz file that comes from another computer (from outside your network), it may contain "public" personalized objects that are not part of your VDBase.mdb file. VisualDesign will read this file and detect these elements.

To inform users and to facilitate the understanding if what is going on, a spreadsheet will appear on your screen when the reading will be done. VisualDesign will let you choose a process for each detected object: either import the new object to your database or give a new name to the identical object that it found while reading the file, or cancel the importation.

If you accept the importation of an object that has an identical name in your database, you can enter a new name for the imported element and VisualDesign will modify the input and use this name if there is no conflict.

This table describes the columns that are part of the spreadsheet:

Column	Description	Editing
Type	Type of personalized common object that has been detected at the reading of the file and which is not part of the user database.	No
Number	Number or name of the detected common object.	No
Equivalent Object	If VisualDesign detected an object in your database that possesses identical parameters, the name of this equivalent object will be written in this column.	No
Action	VisualDesign can choose between two actions when the reading is completed: *Import* or *Use equivalent*. If no equivalent object has been found, the *Import* option will be automatically selected. If an equivalent object has been found, the option *Use equivalent* will be selected.	Yes/No

Column	Description	Editing
New Number	Enter a new number/name to describe the imported object if an identical name has been detected in your database. If you do not enter a name, VisualDesign will rename this object by adding the term "%a" at the end of existing name and will use it for this project file.	Single click

Change or Merge the Current Database

Users are allowed to choose another database at the opening of a VisualDesign project or to merge the current database with another one. Three databases can be changed or merged with another similar one: VDBase.mdb, Mobiles.mdb and Seismes.mdb.

Stay in VisualDesign *Start* window and go to **File** menu. You have the choice to either merge the current database with another one or change it by another one without loosing any information in the original database, including personalized objects. Then, select the database among the list in the **File** menu.

At the next opening of the software, databases that have been specified in registries, at the installation of the software, will be read by default

Type of Files Generated by VisualDesign

Here is a description of file extensions that may be created by the software.

vdz: File that is a renamed ZIP file and includes the VisualDesign project file .vd1 (input) and results file .vr1 (output) files. This zipped file is updated at the closing of a project and this option is available since version 5.4.

vd1: VisualDesign project file. It is a renamed Microsoft Access (mdb) document. This file can be open with the Access *Open* command in order to consult its data and to generate particular reports.

vr1: VisualDesign results file. It is also a renamed Microsoft Access (mdb) document and can be open with the Access *Open* command in order to consult its data and to generate particular reports.

ldb: Temporary file created by Microsoft Access while a mdb, vd1 or vr1 file is open. It does not include any recoverable information about the project except the name of user(s) whom opened the project.

vdk, vrk: vd1 and vr1 backup files created at the opening of a vdz file of the same name, in case the program fails at its opening. These files are deleted if the opening process is normally done.

vzk: vdz backup file that is created if the vdz file is open. It will be deleted if the closing process is normally done.

log: Log file created at the stating of VisualDesign (present in directory C:\VisualDesign\User). The file name is the last eight characters of the MAC address network card, in hexadecimal. If a *Sentinel* key is used, the file name is the last eight characters of the key number.

err: Renamed vd1 file when a writing error has occurred and is not recoverable.

vpk: Temporary file created for the compaction (DAO compact database) of vd1 or vr1 file. This file is deleted if the compaction process is normally done.

vbk: Backup file created when converting a project that was created with a previous version. This file contains the unconverted previous version of the document and it must be renamed .vd1 in order to be used. This file is not deleted.

001, 002, 00x, etc.: Temporary file created at the saving of a project. It is deleted if the saving process is normally done.

VDBase.bak, .1, .2, .x: VDBase.mdb files coming from a previous version. They are located in directory C:\VisualDesign\Sections.

Mobile.bak, .1, .2, .x: Mobile.mdb file coming from a previous version. It is located in directory C:\VisualDesign\Sections.

Display Language

Display Language

VisualDesign™ is multilingual – French, English and Spanish. When you start the application for the first time, you must select the language you wish to operate the software.

Thereafter, the application displays, by default, the language selected during the previous session.

It is possible to switch from one language to another during one session by selecting **Display Language** from **File** menu.

Modifying the Displayed Language

- Select **Display Language** from **File** menu.
- Select one of the available languages from the pull-down menu of the dialog box.
- Press OK

VisualDesign Main Window

Menus

File Menu
Edit Menu
View Menu
Common Menu
Structure Menu
Loads Menu
Analysis Menu
Results Menu
Window Menu
Help Menu

File Menu

The **File** menu contains all necessary commands to manage the files created with VisualDesign™, to print them and to choose the language of display too.

With the **File** menu functions, you can create new documents, save your work, open existing documents or open one of the last documents used and quit the application. They also allow you to display your structure in graphic or spreadsheet form and view it before printing.

It is also from the **File** menu that you will select the parameters of your project and switch from one language to another.

See also
New Project
Open a Project
Close the Project
Save
Save As
Compact Document
Import a DXF
Export a DXF
Import or Export a ProSteel File
Import or Export a CadWork File
Export a bitmap or screen selection as a bitmap
Export an image as an EMF or a screen selection as an EMF
Project Configuration
Display Language
Print

Print Preview
Printer Configuration
List of Latest Opened Files
Exit

Edit Menu

The **Edit** Menu contains the functions that will allow the manipulation of the elements of your structure.

You may cancel or repeat the previous entries, select elements and set their properties. You may also add elements, stretch, copy, paste, move, translate or rotate nodes, split up a member, search, quickly delete elements or their loads.

The selected activation mode (Structure, Load Case, Load Combination, Envelope, Vibration Modes, and Design Results) displays a **Properties** dialog box if you click twice on an element. For example, the **Properties** dialog box displayed in active mode "Structure" for a member will allow you to set or modify the characteristics of this member. In active mode "Load Case", you may set or modify the specific loads for this member. In active mode "Load Combination", you will observe the internal strain and deflections diagrams. In activation mode "Envelope", you will get the envelope results of the selected member.

See also

Undo
Redo
Activate Selection Window
Activate Elements
Activate Mode
Properties
Default Spreadsheet
Configure Copy/Paste
Paste
Add element
Enlarge
Move
Translation
Rotate
Delete
Find
Split
Select

View Menu

View Options
Camera
Zoom Window
Global Zoom
Zoom +
Zoom -
Static Pan
Dynamic Pan
Previous Camera
Next Camera
Previous View
Next View
Animation
Mask
Unmask
Perspective view
Increase the Font size
Reduce the Font size
Diagrams
Tools Toolbars
Status Bar

Common Menu

The **Common** menu is the reflection of VisualDesign™ database called *VDBase.mdb*, which contains basic information about materials, shapes, reinforcement, cables, studs, bolts, steel decks, soils, and mobiles.

This database may be customized if the user adds new shapes, materials, etc. in the appropriate spreadsheet. It can also be shared between users working on a network version of VisualDesign™. See topics Commum.mdb Databases and VDBase.mdb Database for important information.

Mobile data (trucks) are saved in the *Mobile.mdb* database.

These basic data cannot be edited, except the shape availability and the rebar bending shape's number and aliases. However, all these spreadsheets can contain personalized data when a line is added at the end of the spreadsheet.

To create a new object (material, shape, rebar, cable, stud, steel deck, bolt, soil or mobile), add a line at the end of the spreadsheet. You will notice that the ID number is 5000 or more, meaning that it is a personalized object. Change the name and enter new parameters. (Example: copy & paste a type of stud, change the name and its length).

The archiving function is activated by default (**Preferences** tab of **Project Configuration**). Therefore, all personalized objects are saved within your project file (.vd1 or .vdz).

See also

About Shapes spreadsheets

Archiving Common Objects

Importing Common Objects to your Database

Spreadsheets included in the Common menu

MATERIALS

Steel Materials Spreadsheet

Concrete Materials Spreadsheet

Timber Properties Spreadsheet

Aluminium Materials Spreadsheet

SHAPES

I Shapes Spreadsheet I

C Shapes Spreadsheet

HSS Shapes Spreadsheet

L Shapes Spreadsheet

2L Shapes Spreadsheet

T Shapes Spreadsheet

Z Shapes Spreadsheet

WRF Shapes Spreadsheet

V Shapes Spreadsheet

Rectangular Shapes Spreadsheet

Round Shapes Spreadsheet

L (t, w) Sections Spreadsheet

AASHTO Sections Spreadsheet

NEBT Sections Spreadsheet

Cold-Formed Sections Spreadsheet

Built-up Sections Spreadsheet

REINFORCING BARS

Rebar Steel Grades Spreadsheet

Standard Reinforcing Bars Spreadsheet

FRP Reinforcing Bars Spreadsheet

Meshes Spreadsheet
Rebar Bending Shapes Spreadsheet

CABLES

Cable Steel Grades Spreadsheet
Strands Spreadsheet
Post-tensioning Mechanisms Spreadsheet

STUDS

Studs Spreadsheet

STEEL DECKS

Steel Decks Spreadsheet

BOLTS

Bolt Steel Grades Spreadsheet
Bolts Spreadsheet

SOILS

Rocks
Cohesive Soils
Granular Soils

MOVING LOADS

Trucks

Structure menu

TOOLS

Axis Transformation
Checking the Model
Automatic Calculation of Kx, Ky, Kt and Kz
Automatic Calculation of Rigid Extensions
Automatic Calculation of Tributary Areas
Foundation Transformation

GENERATORS

Building Generator
Truss Generator
The Slab & Mesh Generator
Culvert Generator
Abutment, Pier & Retaining Wall Generator
Customized Options

STRUCTURAL MODELING:

Nodes Spreadsheet
Supports Spreadsheet
Spring Supports Spreadsheet
Members Spreadsheet
Plates Spreadsheet
Floors Spreadsheet
Slabs Spreadsheet
Continuous Systems Spreadsheet
Bolted Connections Spreadsheet

SPECIFICATIONS:

Sections' Groups
The Steel Specifications Generator
The Steel Specifications Spreadsheet
The Concrete Specifications Spreadsheet
The Timber Specifications Spreadsheet
The Aluminium Specifications Spreadsheet
Specifications for Shallow Foundations
Specifications for Deep Foundations

GROUPS:

The Steel Design Groups
The Timber Design Groups
Grouping Members
Automatic Grouping (tool)
Group of Plates - Shear Walls
Group of Plates - Surfaces

FOUNDATION:

Stratigraphical Profiles Spreadsheet
Automatic Generation of Foundation Models
Deep Foundation Models Spreadsheet
Shallow Foundation Models Spreadsheet
Foundation Modeling Wizard

Loads Menu

THE LOAD DEFINITION SPREADSHEET:

Load Definition

GENERATION TOOLS:

Automatic Generation of Ice Loads
Automatic Generation of Wind Loads
Load Combination Generation Wizard
Moving Load Case Generation Wizard

LOADS SPREADSHEETS:

Forces on Nodes
Support Settlements
Concentrated Loads on Members
Distributed Loads on Members
Torsional Loads on Members
Temperature Variations on Members
Pressure on Plates
Temperature Variation on Plates
Concentrated Loads on Floors
Distributed Loads on Floors

LOAD COMBINATIONS SPREADSHEET:

Load Combinations Definitions

ENVELOPES SPREADSHEET:

Definition of Envelopes

MOVING LOADS SPREADSHEETS:

Moving Load Cases
2D Axle Factors
Moving Load Envelopes

SEISMIC DIRECTIONS SPREADSHEETS:

Seismic Directions (Linear)
Seismic Directions (Non-linear)

GENERAL DYNAMIC LOADINGS SPREADSHEET:

General Dynamic Loading Diagrams

Analysis Menu

Modal Analysis
Spectral Analysis
Time History Analysis
Non-linear Time History Analysis
Moving Loads Analysis
Static Analysis
Analysis and Design

Results Menu

LOAD COMBINATIONS RESULTS:

Summary
Node Displacements
Reactions at Supports
Internal Forces in Members
Internal Forces in Members (min./max.)
Internal Stresses in Members
Internal Stresses in Members (min./max.)
Internal Forces in Triangular Plates
Internal Forces in Rectangular Plates
Tower Load Combination Results
Stresses in Composite Beams

ENVELOPES RESULTS:

Node Displacements
Reactions at supports (min/max)
Reactions at supports (min/max) and Critical Load Combinations
Diagram of Internal Forces and Deflections
Diagram of Internal Forces and Deflections (min./max)
Min./Max Internal Forces and Deflections for Design Groups
Stress Variations in Member
Internal Forces in Triangular Plates
Internal Forces in Rectangular Plates
Tower Envelopes Results

MODAL/SPECTRAL ANALYSIS RESULTS:

Frequencies
Information on Levels according to Seismic Directions
Node Displacements (Vibration mode)

TIME HISTORY ANALYSIS RESULTS:

Displacement at Nodes vs. Time
Reactions vs. Time
Forces vs. Time
Forces vs. Displacement

DESIGN RESULTS:

Steel Design Results
Bolted Connection Design Results
Concrete Design Results
Timber Design Results

FOUNDATION RESULTS:

Footing Results
Reinforcement in Footings
Pile Results
Pile Reactions

BILL OF MATERIALS:

Bill of Materials (Members)
Bill of Materials (Plates)
Bar List (2-way slabs)
Bar List (Continuous Systems)

Window Menu

VisualDesign™ allows you to visualize parts of your project from different angles simultaneously by using different zoom options. Open the additional windows from within the main window.

You will also learn how to rearrange windows shrunk into icons and how to switch from one window to another when they are covering the whole page or they overlaid each other.

See also
New Window
Cascade
Tile Horizontally
Tile Vertically
Rearrange Icons
Refresh
List of Open Windows

Help menu

Using VisualDesign On-Line Help
Contents
How to Use VisualDesign
Toolbars and Icons Definition
Using Spreadsheets' Contextual Menu
Keys to Control the View on your Screen
Print Functions
Structural Modeling
Analysis Procedures
Technical Support
A Few Tricks
FAQ
Disclaimer
About VisualDesign

Shortcut Keys - Main Window

Menu	Command	Shortcut Key
File	New	**Ctrl+N**
	Open	**Ctrl+O**
	Save	**Ctrl+S**
	Graphic Printing	**Ctrl+P**
Edit	Undo	**Ctrl+Z**
	Redo	**Ctrl+A**
	Properties	**Ctrl+T**
	Default Spreadsheet	**Ctrl+H**
	Copy	**Ctrl+C**
	Paste	**Ctrl+V**
	Translation	**Ctrl+B**
	Delete	**[Delete]**
	Create a Selection	**Ctrl+L**
	Choose a Selection	**Ctrl+K**
	Invert Selection	**Ctrl+I**
	Cancel Selection	**[Escape]**
	Multiple Split	**Ctrl+M**
	Pin Connection	**Ctrl+X**
	Rigid Connection	**Ctrl+R**
	Split at Exact Position	**Ctrl+E**
	Split according to Node	**Ctrl+Y**
Edit / Activate Element	Node	**Ctrl+1**
	Support	**Ctrl+2**
	Member	**Ctrl+3**
	Continuous System	**Ctrl+4**
	Triangular Plate	**Ctrl+5**
	Rectangular Plate	**Ctrl+6**
	Floor	**Ctrl+7**
View	View Options	**O**
	Camera	**C**
	Zoom window	**Z**
	Global Zoom	**G**
	Zoom +	**+**
	Zoom –	**-**
	Dynamic Pan	**P**
	Previous View	**V**
	Mask	**M**
	Unmask	**D**
	Perspective View	**S**
Structure	Grouping Members	**Ctrl+G**
Window	Refresh	**F5**
Help	Online Help	**F1**

Toolbars

Toolbars give access to the most frequently used commands. When you open VisualDesign™, all toolbars selected during the previous work session will be posted by default.

See also

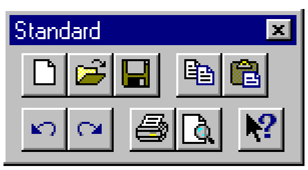
Description of Toolbars and Icons
Displaying or Hiding Toolbars and Tool tips
Moving and Sizing Toolbars

Description of Toolbars and Icons

VisualDesign™ Toolbars and Icons:

Toolbars | **Icons**

Standard

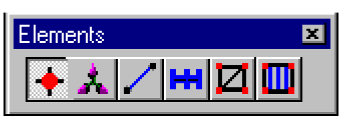


New– Open– Save– Copy– Paste– Undo– Redo– Print – Print Preview– Help

Cursor

Extended Window– Restricted Window– Add

Elements



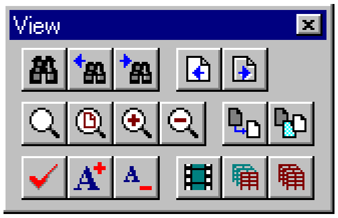
Node – Support – Member – Continuous System - Plate – Floor

Edit

Properties– Move– Translate Rotate– Find– Delete –Create a Selection

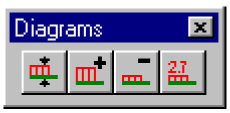
Toolbars	Icons

View



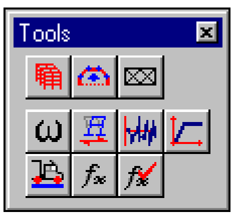
Camera– Previous Camera– Next Camera– Previous View– Next View– Zoom Window– Global Zoom- Zoom +– Zoom - – Static Pan– Dynamic Pan– View Options– Increase font size– Decrease font size– Animation– Mask– Unmask

Diagrams



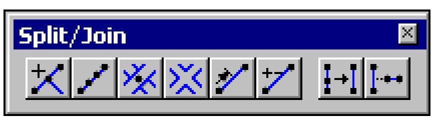
Automatic Scaling – Increase Amplitude – Reduce Amplitude – Display Numerical Values

Tools



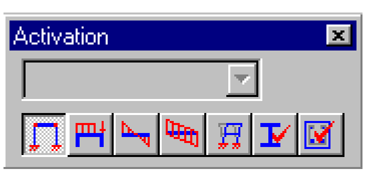
Building Generator– Arc Generator – Truss Generator– Modal Analysis – Spectral Analysis – Time History Analysis – Non-linear Time History Analysis – Moving Load Analysis – Static Analysis – Analysis and Design

Split/Join



Split – Multiple Split – Split with pin connection – Split with rigid connection – Split at exact position – Split according to node– Join - Connect

Activation



Title Selection – Structure mode – Load Case mode – Load Combination mode – Envelope mode – Frequency mode Shapes – Design Results mode – Rebar Placement mode

View Options Selection tree

This toolbar is a selection tree that includes the view options as you can find them in the **View Options** dialog box. Options are grouped per element.
To activate this toolbar, go to **View** menu / **Toolbars** and check the "View Options" box.

See also

View Options Selection tree
View Options dialog box
Displaying or Hiding Toolbars and Tool tips

Moving and Sizing Toolbars
Toolbars

Displaying or Hiding Toolbars and Tool tips

Toolbars and tool tips may be displayed or hidden during a work session.

PROCEDURE:

- Select **Toolbars** from the **View** menu.
- In the dialog box, tick off the toolbars that you wish to display.
  - If you wish to display the toolbar icons' on-line description, activate the "View" radio button under Tool Tips.
  - If you do not wish to display the description, activate the "Hide" radio button.
- Press "OK" to validate your choice or "Cancel" to annul the modifications.

Moving and Sizing Toolbars

You may move your toolbar by clicking on the title bar and dragging it to the new location.

Your toolbars can be positioned beneath the Menu bar, above the Status bar, or vertically on the left or right side of the screen.

Twinning Toolbars Functions

Activation modes, Elements toolbar and Properties function:

In the Structure activation mode, activate an element on the Elements toolbar and click on the function **Properties** . It will allow you to display and edit activated element characteristics or common characteristics of this type of element if you selected many of them.

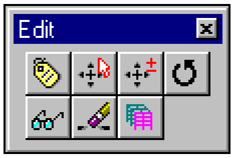
In the Load Case activation mode, the **Properties** function will display Loads dialog boxes so that you can enter loading data for the selected elements, and for the load case that you chose in the "Title Selection" drop-down list box.

In the Load Combination activation mode, the **Properties** function will supply the results for the chosen load combination (Title Selection drop-down list box of Activate toolbar) and for the selected elements (nodes, supports, members, etc.).

In the Envelope activation mode, the **Properties** function will supply the results for the chosen envelope (Title Selection drop-down list box of Activate toolbar) and for the selected elements (nodes, supports, members, etc.).

In the Design Results activation mode, the **Properties** function will supply the steel design or timber design results for the selected elements (nodes, supports, members, etc.).

Activation modes, Elements toolbar and Edit toolbar:



Activate the Structure mode and activate a type of element on the Elements toolbar to add an element of this type or to delete this element.

Also, in the Structure mode, use one of the functions of Edit toolbar (Move, Rotate, Search and Delete) after having activated a type of element on the Elements toolbar. Only nodes can be rotated or moved. (Members that are attached to these nodes will also be moved).

The **Delete** function twinned to the Load Case activation mode will delete the loads that were applied to the selected element.

See also

Displaying or Hiding Toolbars and Tool tips

Moving and Sizing Toolbars

Description of Toolbars and Icons

Toolbars

Status bar

Status bar is located in the lower part of VisualDesign™ program window and provides information about current document or current operation. In addition, when elements are selected on the screen, the status bar displays the type and number of selected objects.

See also

Displaying or Hiding the Status bar

Reading the Status bar

Reading the Status Bar

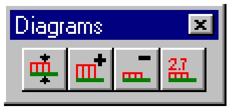
The status bar offers a brief description of the command or toolbar button chosen. In the case of a command with longer execution time, the status bar will give you a message explaining that the operation you chose is in-progress.

Displaying or Hiding the Status Bar

By default, the status bar will be displayed on the screen. To hide it, choose **Status Bar** from the **View** menu.

The **Status Bar** option from the **View** menu is activated when you tick it off.

The Diagrams Toolbar



The "Diagrams" toolbar of VisualDesign main window

Use the **Diagrams** toolbar functions to size displayed objects, forces or diagrams that are shown on the screen. These functions are available in "Structure", "Load Case", "Load Combination", "Envelope" or "Results" modes. Display diagram numerical values by selecting the function "Numerical Values".

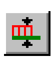
This toolbar includes the following functions:

Automatic Diagram Scaling
Increase Diagram Amplitude
Reduce Diagram Amplitude
Numerical Values

GRAPHIC RESULTS FOR FINITE ELEMENTS RESULTS:

When stress/force contours are displayed, press an icon on Diagrams toolbar to open the tool **Scaling for Intervals**.

Automatic Diagram Scaling



The "Automatic diagrams scaling" icon of Diagrams toolbar

With "Structure" in activated mode, automatically adjust the size of the displayed objects (nodes, supports, etc) using the automatic diagrams scaling button.

With "Load Case" in activated mode, automatically adjust the size of the displayed load case.

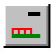
With "Results" in activated mode, automatically adjust the size of the displayed diagrams.

Increase Diagram Amplitude

The "Increase diagram amplitude" icon of Diagrams toolbar

Use this button to increase the size of the objects, applied loads or displayed diagrams, according to the selected activation mode.

Reduce Diagram Amplitude



The "Reduce diagram amplitude" icon of Diagrams toolbar

Use this button to decrease the size of the objects, applied loads or displayed diagrams, according to the selected activation mode.

Numerical Values

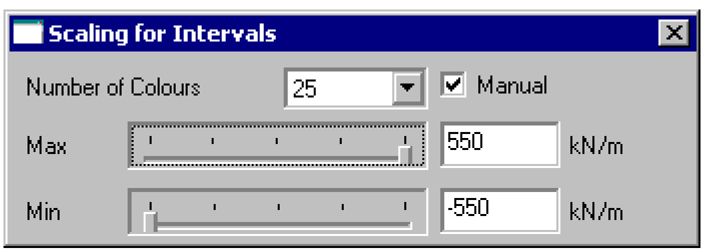
The "Numerical Values" icon of Diagrams toolbar

Use this button to display numerical values associated to applied loads or displayed diagrams, according to the selected activation mode.

Scaling for Intervals

When coloured stress/force contours are displayed for consulting finite elements results, press an icon on Diagrams toolbar to open the **Scaling for Intervals** dialog box.

This tool allows modifying intervals (values and number of colours) for the displayed legend.



Select a number of colours for the display (9, 25, 45, 105, 225, 525 or 1021).

The dialog box is composed of two scroll bar sliders that allow the scaling of intervals from maximum or minimum values when displaying stress/force contours (legend). Each scroll bar is subdivided into 100 units. Marks indicate a subdivision of 25 units. To move the indicator, click on it and slide the mouse at the right or left on the scroll bar. Displayed intervals will be increased or reduced as a result.

The Manual Mode:

When the *Manual* mode is activated, the absolute minimum and maximum values will still be displayed in the coloured legend but the specified value (min and max) will be used as upper and lower limits for the critical zone. If scroll bar sliders are used to modify the displayed values for intervals, the min and max specified values would be used for the modification instead of absolute values.

Therefore, critical zones (red or blue) are always delimited by the absolute value and specified value if the *Manual* mode is activated. If sliders are used, the modification will be done according to specified values.

The legend displays the lower and upper limits for the two critical zones and intermediate zones (colours) display the middle value of the interval.

Shortcut Keys:

Use keyboard shortcut keys to adjust intervals when your cursor is located in the **Max** or **Min** scroll bar.

Shortcut key	Action
[Home]	The [Home] key moves the indicator at the beginning of the scroll bar slider.
[End]	The [End] key moves the indicator at the end of the scroll bar slider.
→	This arrow moves the indicator one unit right.
←	This arrow moves the indicator one unit left.

See also

Grouping plates

The Results FE tab

Interpreting Plates Analysis Results

View Options Toolbar

Display the **View Options** toolbar to quickly verify the structural model and results. This toolbar, which is a selection tree, include the same options as the **View Options** dialog box and will be posted at the right of the screen.

To activate or hide this toolbar, go to **View** menu and select **Toolbars**. Check or uncheck the "View options" toolbar.

The **View Options** selection tree is divided into many roots, which correspond to element (Members, Nodes, Floors, and Plates) in such a way that options are grouped together and can be selected quickly. Display options for results are also available for each element. You will also find the *Foundations* root and *General* root.

See also

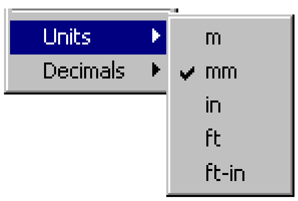
View Options Dialog Box

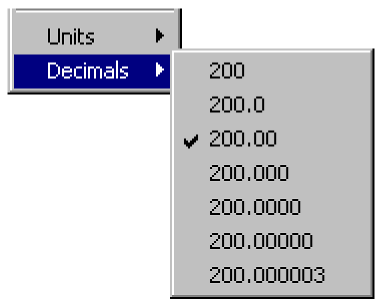
Units

Units and Decimals

Units and decimals can be modified through dialog boxes by right clicking in a field that contains units. A contextual menu appears, which displays the command **Units** and **Decimals**, as shown below. Place the cursor on the command and a list box will display available units and decimals. Click on a listed unit or decimal.

Contextual menu





Units and decimals are also editable in spreadsheet cells through the spreadsheet contextual menu. Refer to topic Change Units (Spreadsheet contextual menu).

Axis System

Global Axis System

Set the global axis system through the **Preferences** tab of **Project Configuration** dialog box (File menu).

Displaying or Hiding the Global Axes

- To hide the global axis system, go to the **Attributes** tab of **View Options** dialog box and disable the display option.
- You may modify the colour of global axis system through the **Colours** tab.

See also

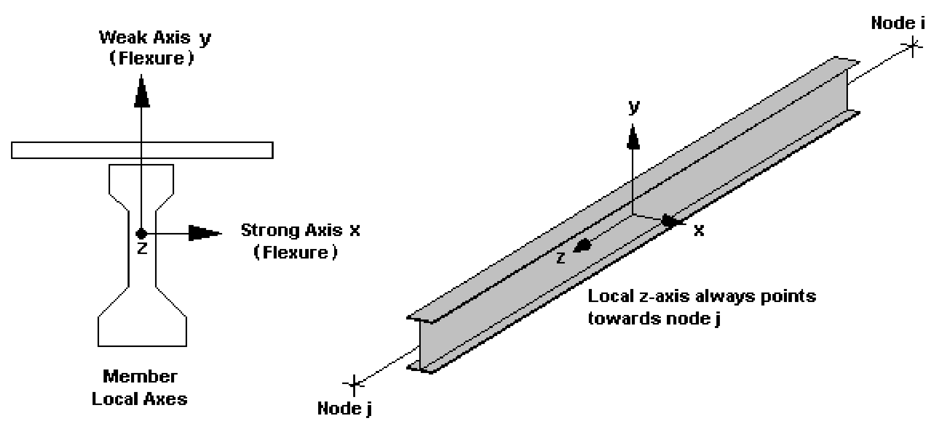
The Member Element
The Colours Tab
The Attributes Tab

Local Axis System

Displaying Member Local Axis System

- Select the **Attributes** tab of **View Options** dialog box. In the **Member** zone, activate the *Local Axis System* option.

The local *y-axis,* which is the weak axis, is the longest of x and y local axes. The local *z-axis* always points towards node j to facilitate the localization of member's incidence nodes.



- To modify the colour of local axes, select the **Colours** tab and expand the *General* title until you reach *Member*.

See also

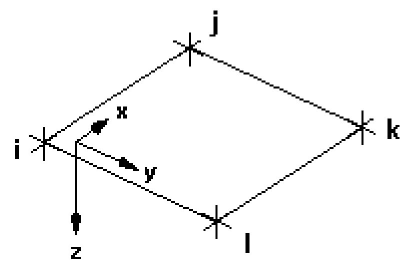
Convention- Forces in Members
The Member
The Attributes Tab
The Colours Tab
Display Member Characteristics

Displaying the Floor Local Axis System

- Go to the **Attributes** tab of **View Options** dialog box and activate the *Local Axis System* option in the "Floors" section.

By default, point (0,0,0) corresponds to the floor's node *i*. Local y-axis always points towards node j.

Floor Local Axes System



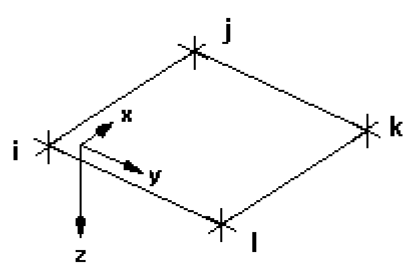
See also
Floor Characteristics
The Attributes Tab
Display Floor characteristics

Displaying the Plate Local Axis System

- Go to the **Attributes** tab of **View Options** dialog box and activate the option *Local Axis system* in the Plates section.

When the local axis system of a plate is displayed, point (0,0,0) corresponds to the plate's Node *i*.

Plate Local Axis System



Dialog Boxes

VisualDesign Dialog Boxes

VisualDesign dialog boxes are used for entering and modifying data for a single element or many of them. We recommend using dialog boxes for inputs because they are more explicit than spreadsheets.

Bi-directional Data Exchange

Generally, data entered in dialog boxes are transferred into corresponding spreadsheets, and vice-versa, for elements characteristics and loads. Spreadsheets are useful for consulting and sorting data or to modify data in a quick way, using spreadsheets' contextual menu.

Calling up a Dialog Box

A dialog box is called up when double clicking on an element with any Activation mode.

It can also be called up by clicking once on an element (or many of them) and by selecting the **Properties** icon on Edit toolbar or in **Edit** menu.

The dialog box that will appear on screen depends on the Activation mode and activated element (Node, Member, Floor, or Plate):

- The Structure activation mode calls up **Element Characteristics** dialog boxes.
- The Load Case activation mode calls up **Loads** dialog boxes.

Edit Functions

A dialog box is also used when some functions or tools are called up such as **Copy/Paste**, **Translation**, **Rotate**, and **Automatic Generation of a Building**, among others.

Dialog Box's Contextual Menu

Numerical values included in VisualDesign dialog boxes can be edited through a contextual menu (right click). This menu is composed of the following functions: Undo, Cut, Copy, Paste, Delete, Select all, Units and Decimals. These functions cannot be applied to list box.

Modify Units in a Dialog Box

Please refer to Units and Decimals

Spreadsheets

Spreadsheet Functionalities

VisualDesign™ spreadsheets let you copy and paste data to and from a commercial spreadsheet.

The spreadsheets may be printed out directly without using the **Print Spreadsheet** command on the **File** menu. You can choose to display or hide columns, this way formatting your spreadsheet print output. In the contextual menu of the spreadsheets, you will find different options allowing you to change measuring units, launch search operations, replace data, apply arithmetic operations, and quickly modify alphanumerical numbers using auto-numbering.

You may insert, delete, or sort spreadsheet lines.

See also

Shortcut keys

Left mouse button

Spreadsheet's Contextual Menu

Selected Elements and Spreadsheet Contents

Short cut Keys

Inserting a Line in a Spreadsheet

Print spreadsheets (File Menu)

Shortcut Keys (Spreadsheets)

VisualDesign™ lets you perform certain operations with the keyboard control keys.

Activate shortcut key	To
ENTER	Exit the active cell while validating the new input value. The cursor will move to the next right-hand cell.
ESC	Stop editing an active cell and replace its content by the previous value.
INS	Insert a line above the cursor.
Arrow keys	Move the cursor one cell at a time (right, left, up, down)
CTRL +↑ **CTRL + ↓**	Move the spreadsheet up or down while keeping the initial cell selected.

Activate shortcut key	To
Mouse scroll wheel	Move the spreadsheet up or down while keeping the initial cell selected.
CTRL + → **CTRL + ←**	Move the spreadsheet right or left while keeping the initial cell selected.
PgUP **PgDN**	Move a spreadsheet forward or backward by as many lines as indicated on screen.
CTRL + Scroll wheel	Move a spreadsheet forward or backward by as many lines as indicated on screen.
HOME **END**	The HOME key moves the cursor to the top of the screen; the END key moves it down to the bottom of the screen.
CTRL + HOME **CTRL + END**	Press CTRL HOME to display the beginning of a spreadsheet from Line 1. Press CTRL+ END keys to display the last line of the spreadsheet.

Left Mouse Button

The left mouse button is a selection button.

To	Do the following
Select a cell	Click the cell with the left mouse button.
Select more cells consecutively	Click the first cell with the left mouse button, keep the button pressed and drag the mouse diagonally across to the last cell.
Select an entire column	Click the column header with the left mouse button.
Select an entire line	Click the line header with the left mouse button.
Select the entire spreadsheet	Click the "Select All" button in the upper left corner of the spreadsheet.
Edit the cell content	Click the cell and enter the new data.

Selected Elements and Spreadsheet Contents

When elements of the same type are selected on screen, the spreadsheet will include selected elements only. However, if no element is selected, the spreadsheet will include all elements when the spreadsheet will be called up.

Elements that are selected in a spreadsheet will be selected on screen when the spreadsheet will be closed. This functionality applies for Structure, Load case, Load combination, Envelope, Vibration mode, and Design Results Activation modes.

Default Spreadsheet

Use the function **Default Spreadsheet** function of **Edit** menu or use the short-cut key **[Ctrl]+H** to quickly call up a default spreadsheet instead of doing it through VisualDesign™ different menus (Structure, Loads or Load Combinations). The "default " spreadsheet that will be displayed depends on the activation mode you are working in.

If elements were selected on your screen, the default spreadsheet will include these elements only. However, if none were selected, all elements will be included.

Structure activation mode:

- Select a type of element on the Element toolbar and use this function to call up the corresponding spreadsheet that includes data and coordinates.

EXAMPLE:

- Activate the "Node" icon on the Element toolbar.
- Select all nodes or few of them.
- Select the Default spreadsheet function under **Edit** menu or use the short-cut keys [Ctrl]+H.

In the "Load Case" activation mode:

- If you choose a load case in the Title bar of Activation toolbar: the default spreadsheet corresponds to loads applied on the type of element that you activated and selected on your screen.
- If you have not chosen any load case in the Title bar: the default spreadsheet will correspond to the **Loads Definitions** spreadsheet. Add new type of load cases.

In the "Load Combinations" activation mode:

- If you activated a load combination on Activation toolbar, the default spreadsheet corresponds to **Internal Forces - Members** if you selected a member. If another element is selected, the corresponding results spreadsheet will be open.
- If you have not chosen any load combination in the Title bar: the default spreadsheet will correspond to the **Load Combination Definitions** spreadsheet. Add new load combination, if desired.

In the "Design Results" activation mode:

- Selected members and the default spreadsheet will correspond to the **Design Results** spreadsheet (Steel, Concrete, or Timber).

Spreadsheet's Contextual Menu

A contextual menu is accessible from all VisualDesign spreadsheets, through a right click. It includes the following functions: Details, Change Units, Find, Auto-numbering, Modify, Replace, Sort, Sort Up, Sort Down, Selection of Contiguous Items, Insert, Delete, Copy, Copy with Titles, Paste, Paste with MSAccess, Duplicate, Mask, Display, Column Width, Page Layout, Print and Print All Tabs.

If functions are shaded, it means that they are not available. It depends on the selected objects (a line, a column, etc.)

Details

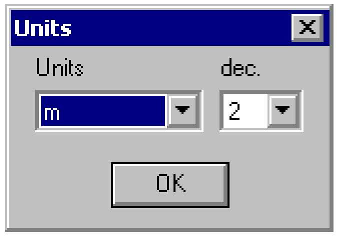
Select this command to access details related to the selected elements. If the command is shaded, it means that there is no detail available for this particular element.

Change Units

Units can be changed within dialog box or spreadsheet. A change of units in a spreadsheet column will affect the same type of data included in another spreadsheet. This exchange is defined as a bi-directional: Spreadsheet <–> Dialog box.

PROCEDURE:

- Select the column by clicking in its title and right click to open the contextual menu.
- Choose **Change Units**.
- In the **Change Units** dialog box, select one unit in the scroll list. Indicate the number of decimals to be used with real numbers.



- Press OK to validate your choice.

RESTRICTION:

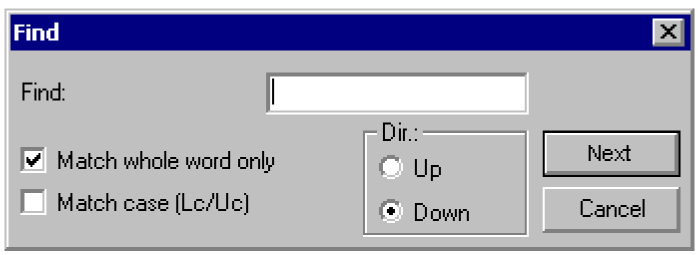
Applicable to columns having units.

Find

This function only applies to alphanumeric numbers and text items.

PROCEDURE:

- Use the left mouse button to select a cell in the column where you want to locate an element or a value. Press the right mouse button to open the contextual menu.
- Choose **Find**.



- In the **Find** dialog box, enter the word or alphanumeric number you are looking for. Activate the *Whole word only* option if you want VisualDesign™ to find the specified value as a "whole" rather than as a "part". Activate the *Match Case* option to include upper- and lower-cases as search criteria. In the "Direction" section, select the *Up* or *Down* radio button to direct the search upwards or downwards from the insertion point.
- When there is more than one value or element found with the search criteria, VisualDesign™ will activate the "Next" button allowing you to look at other data.

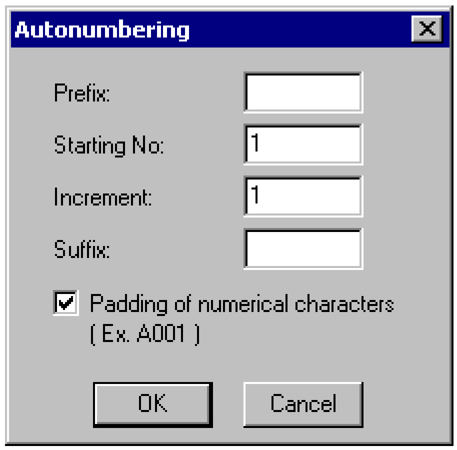
RESTRICTION:

Not applicable to columns with integral or real numbers.

Autonumbering

PROCEDURE:

- Select the column title to select the whole column or select a group of cells that you want to be numbered.
- Right click to open the contextual menu.
- Choose Auto numbering.



- To create a list of alphanumeric numbers, enter the starting number and increment. Prefix and suffix are optional.
- To create a sequential list of real and integral numbers, enter the starting number and the increment and press OK.

Option "Padding of numerical characters"

This option avoids producing identical numbers for copied/pasted elements, by replacing the identical part by its ID number, which is specific to each new element.

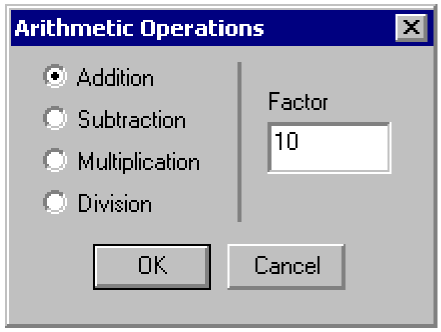
RESTRICTION:

Not applicable to a range of cells belonging to different columns.

Modify

PROCEDURE:

- Select a cell, a group of cells, or a column in which you want to apply an arithmetic operation.
- Right click to open the contextual menu.
- Choose **Modify**.



- In the dialog box, activate the radio button corresponding to the arithmetic operation you want to apply. Type in the number that will affect the selected cells.
- Click OK.

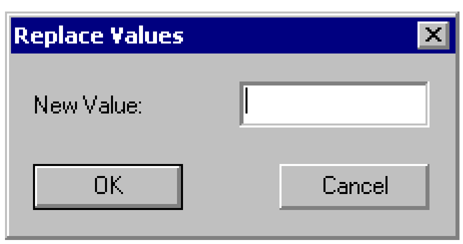
RESTRICTIONS:

Applicable to real numbers only. Not applicable to a range of cells belonging to two different columns.

Replace

PROCEDURE:

- Select a cell, a group of cells, or a column in which you want to replace values by another one.
- Right click to open the contextual menu.
- Choose **Replace**.



- Enter a new value in the **Replace Values** dialog box.
- Click OK.

RESTRICTIONS:

Not applicable with ID numbers or alphanumerical numbers.

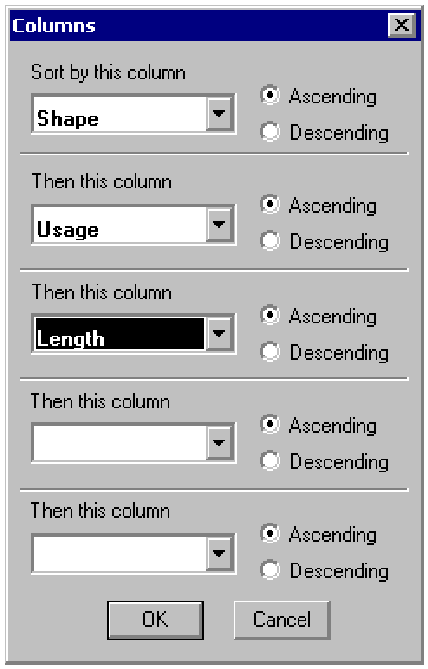
Not applicable to a range of cells belonging to two different columns.

Sort

PROCEDURE:

You may sort the whole spreadsheet or selected lines.

- To sort a whole spreadsheet, use the left mouse button to select a cell.
- To sort a restricted number of lines, choose a range of cells or line headers targeted by the sort operation.
- Right click to open the contextual menu.
- Choose **Sort.**



- In the **Column** dialog box, select columns (up to 5 columns can be added in a sorting) according to the priority you want to get, and specify an ascending or descending order.
- Click OK.

Sort Up & Sort Down

You may sort values in a column, up or down, numerically or alphabetically.

PROCEDURE:

- Click a cell located in the column that you wish to sort or select the whole column by clicking on its title.
- Right click to open the contextual menu.
- Choose **Sort Up** or **Sort Down**.

Selection of Contiguous Items

To select identical items, they must be contiguous in the spreadsheet. When there is no selection in the spreadsheet, this function allows finding and highlighting identical items. It can be useful for consulting modeling data or results.

PROCEDURE:

- Click in a cell and right click to open contextual menu.
- Choose **Selection of Contiguous Items**. The search begins at the pointed cell and ends at the last identical item.

Select Identical Items in a Spreadsheet

Use the contextual menu function **Selection of Contiguous Identical Items** to group elements in the spreadsheet. Select these lines and click OK to exit the spreadsheet.

Example:

Open the Steel Design Results spreadsheet and sort member shape W310x39. Then, use function **Selection of contiguous identical items** to select all W310x39. Click OK. These members will be highlighted on the screen. Go to **Edit** menu and select **Select/Create a Selection** or press icon . Give a name to this selection. You can call back this selection any time.

See also

Create a selection (Personalized Selections of Elements)

Insert

PROCEDURE:

- Select the same number of lines you want to insert.
- Right click to open the contextual menu.
- Choose **Insert**.
- New lines will be inserted above the group of selected lines.

NOTE: INSERTING A LINE IN AN EMPTY SPREADSHEET

- Use the left mouse button to select the first line header of the empty spreadsheet.
- Press the [Insert] key or right click to open the contextual menu and choose "Insert".
- To insert more lines, position the cursor in the end cell of first line, then press [Enter].

Delete

PROCEDURE:

- Select the line(s) to be deleted.
- Right click to open the contextual menu.
- Choose **Delete**.

Copy

PROCEDURE:

- Select the line(s) that you want to copy.
- Right click to open the contextual menu.
- Choose **Copy.**

Copy with Titles

VisualDesign allows to copy cells along with corresponding units to another spreadsheet that does not belong to VisualDesign™.

PROCEDURE:

- Select the line(s) that you want to copy.
- Right click to open the contextual menu.
- Choose **Copy with titles**.
- Open the other spreadsheet and paste data.

Paste

PROCEDURE:

- Move the cursor up to the cell where data will be pasted.
- Right click to open the contextual menu.
- Choose **Paste**.

Paste with MSAccess

The software allows pasting data from an MS Access spreadsheet to any VisualDesign spreadsheet.

PROCEDURE:

- Select the line(s) in the MS Access spreadsheet.
- Back in VisualDesign spreadsheet, move the cursor to the appropriate place. Right click to open the contextual menu.
- Choose **Paste from MS Access**.

Duplicate

This function is available in spreadsheets that are composed of tabs. These tabs include related parameters that belong to the master spreadsheet. This function allows you to copy a line along with the data included in all the other tabs.

Example: This function will copy a load combination along with its load factors.

PROCEDURE:

- Open a spreadsheet that is composed of many tabs.
- In the master spreadsheet, usually the first one, select the line that you want to copy along with data included in other tabs.
- Right click to open the contextual menu.
- Choose **Duplicate**.
- The copied line will be added at the bottom of the spreadsheet. Edit the name or number to avoid identical data.

Mask

You are allowed to hide one or more columns to get an uncluttered spreadsheet or to select columns for printing.

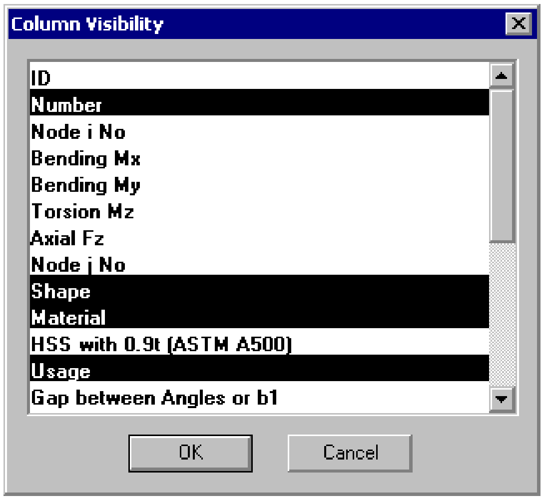
PROCEDURE:

- Click the column's header with the left mouse button to select the whole column.
- Right click to open the contextual menu.
- Choose **Mask**.
- To display the column(s) again, choose function **Display** in contextual menu.

Display

PROCEDURE:

- To display the hidden columns, click the right mouse button to access the contextual menu and choose **Display**.
- Click on column titles that you want to display in the spreadsheet, while keeping down the [Ctrl] key. If you want the whole column to be displayed, push down the left mouse button while you move down the cursor.

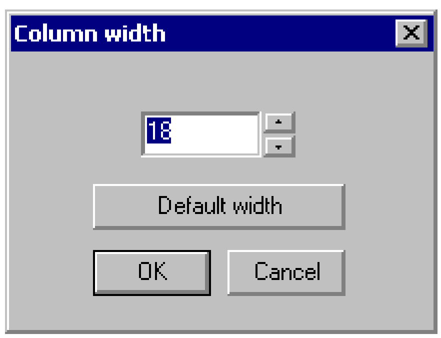


Column Width

You can define column widths to format your spreadsheet.

PROCEDURE:

- Click the column header with the left mouse button to select the whole column.
- Right click to open the contextual menu.
- Choose **Column Width**.



- In the dialog box, enter the desired width (between 1 and 255 characters) and press OK or choose a default value by clicking the "Default width" button.
- Use the same procedure for other columns.

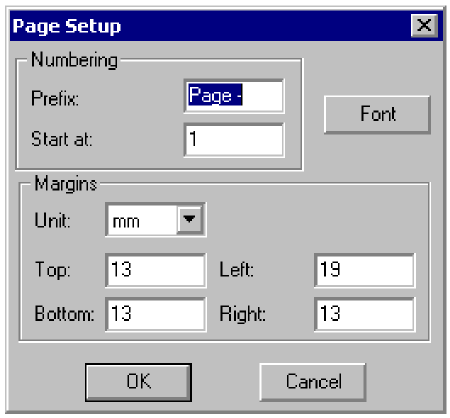
Note that the modifications done to column widths will be saved for the next sessions until you change them.

Page Setup

Before printing a spreadsheet, select this function to fix printing attributes.

PROCEDURE:

- Click a cell and right click to open the contextual menu.
- Choose **Page Setup**.



- In the **Page Setup** dialog box, enter the layout options such as numbering, margins, and font in the dialog box.

Note that the modifications will only be visible at the printing.

Print

You may print the whole spreadsheet or only a part of it. If you want to print only a part, you must first select the cells or the lines to be printed.

PROCEDURE:

- Select the whole spreadsheet or a part of it.
- Right click to open the contextual menu.
- Choose **Print**.

Print All Tabs

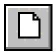
This function allows you to print all the tabs that are composing a spreadsheet, as the command **Print / Spreadsheets** in the **File** menu would do.

PROCEDURE:

- Click in any cell in the first spreadsheet.
- Right click to open the contextual menu.
- Choose **Print All Tabs**.

Managing your Project

New Project



The "New Project" icon of Standard toolbar

When activated, the **New Project** function opens a blank document, temporarily named Projet1. It is then recommended to select the **Project Configuration** command from the **File** menu to register identification data for this new project (project name and number, selected units, type of analysis, selection of the gravitational axis, etc.).

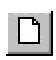
You can create a new project at any stage of your work session. Select the **New** function from the **File** menu or click on the "New Project" icon from the Standard toolbar.

See also

Project Configuration

Creating a New Project

Create a new project by doing one of the following:

- Select the **New** function from **File** menu.
- Click the icon  on Standard toolbar.

Open a Project

The "Open Project" icon of Standard toolbar

To open an existing document on which you have worked recently, select it from the latest opened files that appear at the bottom of the **File** menu.

You can also access an existing document by selecting the **Open** function of **File** menu or by clicking on the "Open Project" icon of the Standard toolbar.

When the **Open** dialog box is displayed on the screen, select the name of a document from the "Name" list, another disk drive, another directory, or another "type" of file.

If some shapes or materials are missing at the opening of a project, the user has the choice to cancel the reading or to continue and lose results. However, this will not happen if a user saved all common and personalized objects used in his project with option *Archiving common objects* of **Preferences** tab (**Project Configuration**). This option saved all objects within the .vd1 or .vdz file.

See also

Preferences Tab

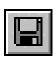
Opening an Existing File

- Do one of the following:
  - Click the icon  of Standard toolbar
  - Select **File/Open**
- In the "File Name" zone, type or select the name of the document to open.

If the document you want to open is not on the list, select a different disk drive, directory or file, or select another type of file in the "Types of Files" zone.

- Select "OK".

Save



The "Save" icon of Standard toolbar

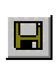
From time to time, during your project at hand, the program will ask you to save your project.

It will happen when:

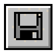
- You have modified some elements of the structure, load cases, load combinations or envelopes and you launched an analysis.
- You changed a parameter in the **Preferences** tab or **Analysis** tab (**Project Configuration** dialog box). If the number of member subdivisions have changed, VisualDesign™ will ask you to confirm the load combination.
- You changed one of the parameters for the design of footings in the **Foundations** tab (**Project Configuration** dialog box).

However, we recommend that you regularly save your project during a working session.

Saving Current File

- Use one of the following procedure:
  - Click the icon  of Standard toolbar.
  - Go to **File/Save**.

Save As



The "Save" icon of Standard toolbar also gives access to the "Save as" function

VisualDesign™ uses the features of the operating systems Windows NT™ and Windows 95+™:

Select **File/Save As** or click on the "Save" icon in the Standard toolbar to save a new document. The **File/Save As** function is often used to save the results of one structure under one name and create a copy of the same structure under another name which can then be modified and re-analyzed.

Note that the "Save" icon gives access to the **File/Save As** dialog box when it is the first time you save a new document.

File and directory names may have up to 256 characters, including extensions. File names may contain capital or small characters, except for the following symbols: * " \ / < >. Extensions must be .vd1 or .vdz. Extension .vdz is a compressed file that includes .vd1 and .vr1 files and is created at the closing of a project if the *Compression of VD files* option has been activated in the **Preferences** tab of **Project Configuration**.

For more information concerning types of files, refer to the documentation about operating systems Windows NT™ and Windows 95+™.

Saving a File As or Copying a File

- Go to **File** menu and select **Save As**.
- From the "Disk Drive" section, select the disk drive where you wish to save your document.
- Select the directory in which you want to file this new document.
- In the "File Name" section, type the new name of the document.
- Push down the "OK" button to save the document or on the "Cancel" button to stop the operation.

Compact document

By default, your .vd1 and .vr1 files are automatically compressed (zipped) when exiting a project. This function compacts unused spaces of files .vd1 and .vr1 in order to reduce the space occupied by these files on the hard disk. It could be useful if you deleted many objects.

If you do not want the files to be zipped, uncheck the appropriate box in the **Preferences** tab of **Project Configuration**.

Note. This type of compression is not a ZIP. Only empty spaces are deleted.

Compression of VD Files

By default, .vd1 and .vr1 files will be compressed (*zip*) at the closing of a project. This option can de deactivated in the **Preferences** tab of **Project Configuration**, at section "Compression of Files".

Import a DXF File from AutoCAD

You can import a DXF file from AutoCAD to VisualDesign. To do so, go to **File** / **DXF In**. Elements from the DXF file will be converted into members that can be modified afterwards in VisualDesign.

- Make a copy of the drawing file (*.DWG).
- Use the AutoCAD **Explode** command to breakdown the drawing into blocks.
- Use the AutoCAD **Purge All** command to purge unused blocks.
- Remove irrelevant parts of the drawing for VisualDesign, such as texts, cartridge, etc.
- Save the *.DXF file.
- Go to VisualDesign **File/Import** menu and select **Import a DXF.**
- A dialog box will appear on screen while importing the file. Specify the units for the imported drawing (in, ft, m or mm) and VisualDesign will convert them, if applicable.

VisualDesign reads the *Line*, *Pline* and *3D Face* AutoCAD objects. VisualDesign transforms the *Line* and *Pline* objects for members when it reads the DXF file. *3D Face* objects are transformed into rectangular and triangular plates.

Export a DXF File to AutoCAD

You can export a DXF file of your structure to AutoCAD. Elements (floors, members and plates) are saved on different layers. Members imported to AutoCAD can be seen the same way as in VisualDesign™ (lines) or with relief (3D-Face).

Rebar placement drawings coming from the Rebar placement window (Concrete Design module) can be exported too.

Procedure

- Select the function **Export a DXF** from **File / Export** menu in VisualDesign main window or Rebar Placement window.
- The **Save as** dialog box will appear on screen and the file name will have the extension .dxf. Choose the directory where the file will be stored. Press the OK button.
- Open AutoCAD and open the dxf file.

Close a Project

Closing a project without saving it will erase the memory of that project.

To close a project, select **File /Close**. If your last modifications have not been saved, the **Save** dialog box will appear on the screen.

You can also double-click on the control box of the document located in the upper left corner to close your window. All other active projects will remain open.

See also

Compression of VD files
Compact Document

Latest Opened Files

In the **File** menu, VisualDesign™ lists the four (4) latest files that you have been working with. To reopen any one of these projects, simply click on it from the list.

Note. If you erased a file on the Latest -Opened -Files list before or during a VisualDesign™ session, the name of that file will still appear on the list. However, VisualDesign™ will give an error message if you try to reopen the file.

Exit

To exit VisualDesign™, choose **Exit** from the **File** menu.

VisualDesign™ will close all the open records simultaneously. If you have forgotten to save the changes made to one or more documents, a dialog box will appear and allow you to save these modifications.

Export Bitmaps and EMF Files

Export a screen image or a screen selection as a bitmap

Functions **Export as a BMP** and **Export a selection as a BMP** are located in **File** / **Export** submenu**.**

Export as a BMP

Users can export the visible image on screen as a bitmap. It can be saved in a file or in the clipboard. This new tool is useful to extract images for reports and can help users to find a particular project among many.

Export a selection as a BMP

Select this function to make a selection of image on your screen and export this partial view as a BMP. When this function is selected, the cursor will change into a cross. Draw a window around the image that you want to export.

Export as a Bitmap Dialog box

The selection of either functions calls up the **Export as a Bitmap** dialog box in which you will define the image parameters. The dialog box is composed of the following sections:

Section	Description
Current size	Current image width and height.
Sizing	Select an option among the following: 320x240 pixels 640x480 pixels 800x600 pixels 1024x768 pixels Scale factor (Enter a factor) Resolution (Define o resolution in dpi)
Final size	According to the selected option, the image will have these width and height. You can also enter dimensions directly in these fields.
Save	Choose an export option: Export to clipboard Export to file (Choose a path in the drop-down list box).

Export an image or a screen selection as an EMF File

An image in EMF format ((*Enhanced MetaFile*) can be reduced or magnified without loosing its resolution and quality. This type of file as an .emf extension. A metafile can include a file of vectorial images, point images, and another file that contains text, and it can be included as an image in a document that supports EMF image format.

Functions **Export as an EMF File** and **Export a selection as an EMF File** are located in **File / Export** submenu**.**

Export as a EMF File

Users can export the visible image on screen as a EMF image file. It can be saved in a file or in the clipboard. This new tool is useful to extract images for reports and can help users to find a particular project among many.

Export a selection as a EMF File

Select this function to make a selection of what is displayed on screen and export this partial view as a EMF image file. When this function is selected, the cursor will change into a cross. Draw a window around the image that you want to export.

Export as an EMF File Dialog box

The selection of either functions calls up the **Export as a EMF File** dialog box in which you will define the image parameters. The dialog box is composed of the following sections:

CadWork Applications

CadWork Files

VisualDesign is now partially compatible with **CadWork** applications concerning the reading of the file. The importation of a CadWork file can be done through the function **Import a CadWork File** in VisualDesign **File/Import** menu. Bi-directional exchanges will be possible in a future compilation.

ProSteel Applications

ProSteel Files

VisualDesign is now partially compatible with **ProSteel** applications concerning the reading of the file. The importation of a ProSteel file can be done through the function **Import a ProSteel File** in VisualDesign **File/Import** menu. Bi-directional exchanges will be possible in a future compilation.

Project Configuration

The **Project Configuration** dialog box is available in **File** menu and contains specific tabs allowing you to define basic parameters that need to be considered for the specific analysis that you are planning to run with VisualDesign. The number of tabs depends upon the number of modules that you own. The complete list is: **General**, **Preferences**, **Analysis**, **Seismic**, **Foundation**, **Steel**, **Composite Beam**, **ASCE-10-97**, **Concrete Design** and **Prestressing**.

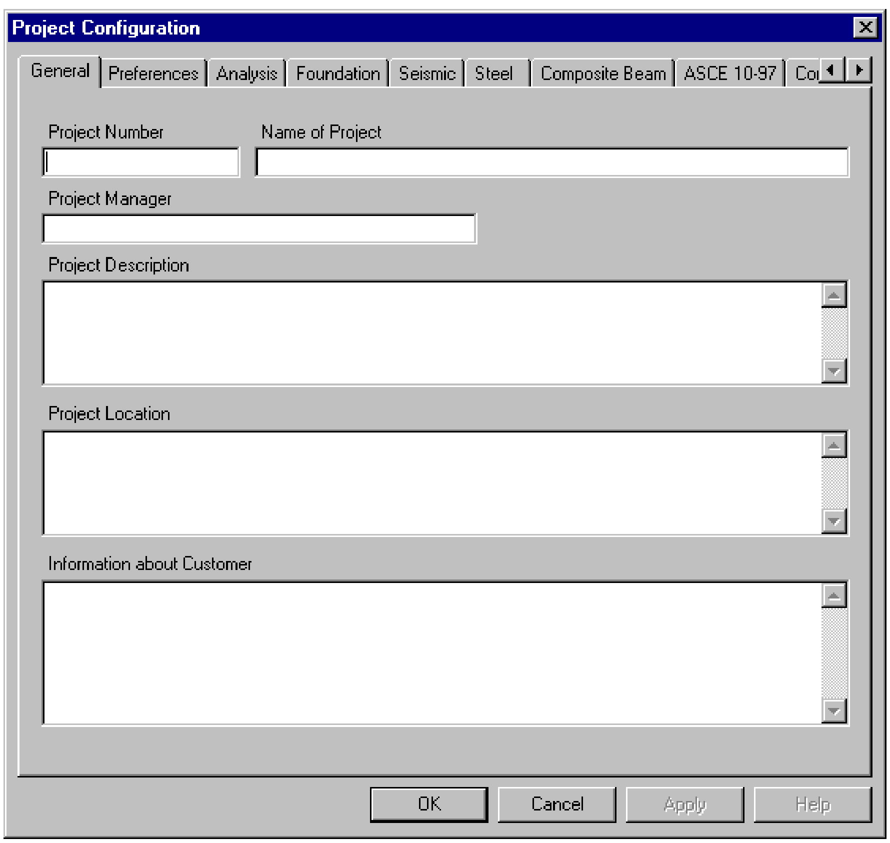
Print the project configuration data by using function **Print** / **Spreadsheets** in the **File** menu. Choose information that you want to print in the selection tree. See topic *Printing spreadsheet*.

Units

Units are specified in the **Preferences** tab along with the shape designation (metric or imperial)

General Tab

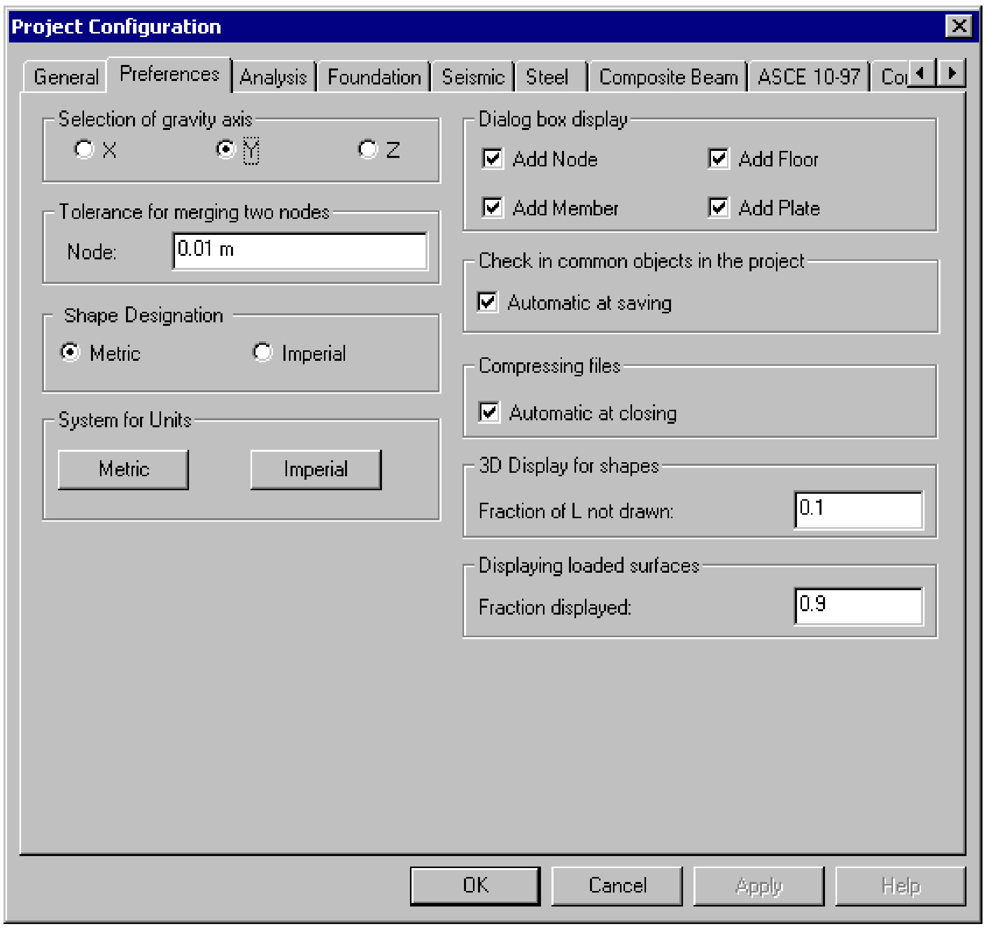
The **General** tab may include information about the project such as name and number, person in charge, a description of the project and location, and information about the client.



Preferences Tab

Specify your preferences for the gravity axis, minimum distance between two nodes, metric or imperial designation of shapes & system of units and the display of characteristics dialog boxes when adding elements.

Specify an automatic compression of your .vd1 and .vr1 files and the check in of all personalized objects within your project (.vd1 or .vdz) when exiting the project.



Heading	Description
Selection of gravity axis	By default, the selected gravity axis is Y. It is recommended to select the gravity axis before beginning a project.
Tolerance for merging two nodes	Minimum distance between two nodes. If the distance is less than this value, nodes will be merged.

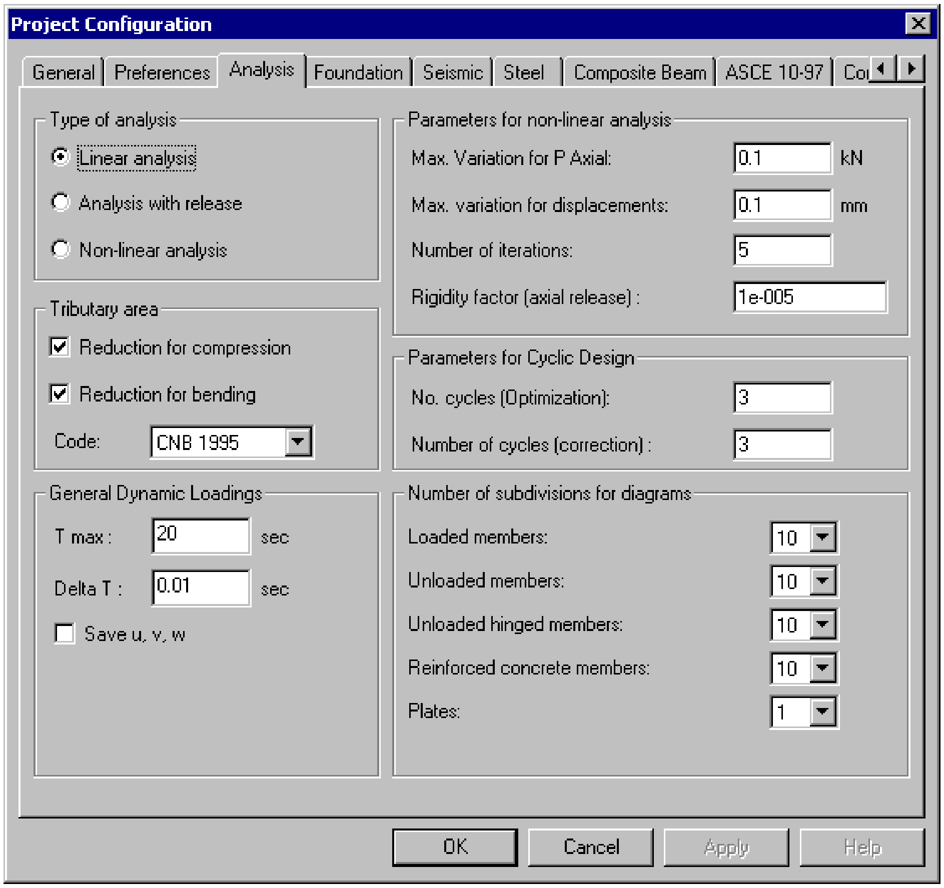
Heading	Description
Shape Designation	Select the Metric or Imperial system of units from the shapes designation and also the country to know the shape availability
System of Units	Select a Metric or Imperial system of units by clicking the appropriate button.
Dialog Boxes Display	If you want the dialog boxes for node, member, floor and plate characteristics to be displayed each time you add an element, select the respective check box. **Note.** It is usually faster to create all the elements first without the dialog boxes- then to proceed to defining their properties in groups using the **Properties** function.
Check in common objects in the project	By default, VisualDesign saves all objects (common and personalized) within your .vd1 file. To deactivate this option, uncheck box "Automatic at the saving".
Compressing files	By default, option "Automatic at closing" is activated allowing the compression of files .vd1 and .vr1.
3D Display for shapes	Fraction of L not drawn: The default value of 0.1 means that 10% of the member length will not be displayed in 3D, at nodes.
Displaying loaded surfaces	Fraction displayed: The default value of 0.9 means that loads applied on surfaces will be display on 90% of the surface of a floor or plate. This help to visualize alternated loaded floors. This value shall be greater than 0.5 and less than 1.0.

See also

Properties
Compression of VD files
Archiving common objects within your .vd1 file

Analysis Tab

Specify the type of static analysis to be run (linear, non-linear, or with release), parameters for non-linear analysis, subdivision of members for the display of internal forces, reduction factor for tributary area, and parameters for a general dynamic (transient) analysis.



This table describes the fields in this dialog box:

Heading	Description
Type of analysis	Activate a linear or non-linear static analysis or a static analysis with release if supports or members need to be released during analysis. Refer to topics: Types of Static Analysis and Tension-only Members
Tributary Area	Specify the type of reduction factor that will be applied to tributary areas: Compression or Bending. With the drop-down list box, select the code that will be apply to reduce overload.
Parameters for Non-linear Analysis	Parameters are shaded if you activated a linear analysis. If you activated an analysis "with release", only the number of iterations can be specified for said analysis.
Max. Variation for axial P	The non-linear analysis terminates when the variation over P axial falls below this value. This is a convergence criterion.

Heading	Description
Max. Variation on Displacements	This tolerance is applied to the displacements of axially released supports only.
Number of iterations	The non-linear analysis or the one considering release will end when the specified maximum number of iterations will be reached.
Rigidity factor (axial release)	Specify a rigidity factor for axially released members.
Parameters for Cyclic Design	
Number of cycles (optimization)	Number of cycles for optimization when designing members.
Number of cycles (corrections)	When the optimized number of cycles is reached, members that have not been optimized will be evaluated with the correction mode.
Number of subdivisions for the diagrams	Number of subdivisions applied to all members no matter the load condition. It can be specified for loaded beams, unloaded beams, unloaded pinned beams, concrete members and for rectangular plates.
General Dynamic Loadings	
Duration	Allows fixing a maximum time for the application of this type of dynamic loading on a structure
Time pitch	Specify the time pitch. Make sure that dti is larger than the time pitch otherwise there will be a warning. See the topic "General Dynamic Analysis" for more details.
Save Node Displacements	Save the time responses of node displacements in the database (Project_Name.vr1). See the note below.

Save Node Displacements

If you prefer, you can save only a few nodes to shorten the time of analysis. Select the desired nodes and create a personalized selection before launching the analysis. From menu **Results/ Time History/ Nodes displacements**, observe the graph of nodes displacement in time.

See also

Released Supports
Released Members
General Dynamic Analysis
Automatic calculation of Kx, Ky, Kt and Kz
Load reduction due to tributary areas

Seismic Tab (General)

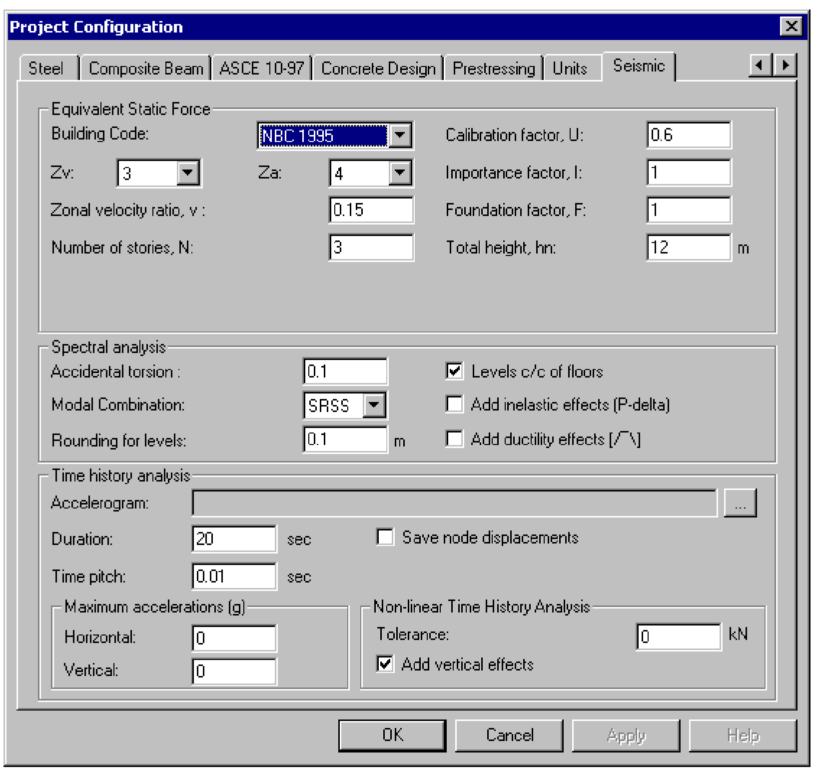
When the engineer specifies the R factor that represents the ductility of a structure, this allows a certain deflection of the structure during an earthquake. These deflections, caused by additional forces in the structure, can be evaluated in two ways: an elastoplastic analysis or an approximate method as defined in the *Canadian National Building Code* (1995 or 2005).

VisualDesign™ does elastoplastic analysis or the approximate method for each seismic direction that the user wishes to study. VisualDesign™ corrects the spectral analysis (or time history) by including inelastic effects. The engineer will find in the **Levels spreadsheet** all parameters used by VisualDesign™ to compute the stability coefficient Θx used to amplify the forces.

Furthermore, VisualDesign™ adjusts the spectral analysis (or time history analysis) by including these inelastic effects. The user may select the "Information on Levels" spreadsheet to have a look at the parameters that have been used to compute the *qx* coefficient that is needed for the calculation of the amplified forces in the structure.

The **Seismic** tab includes general parameters required to run a spectral and time history analysis. Specific parameters will appear in this tab according to the code that will be selected in the "Construction Code" box. Available codes are: CNBC-05, CNBC 95, UBC 94, UBC 97, and CAN-S6-00.

Seismic tab - NBC-95 Code



This table describes the fields included in this tab when the *Canadian National Building Code 95* has been selected in the "Building Code" list box.

Heading	Definition
Equivalent Static Force	
Building Code	Each building code uses a normalized spectrum according to the occurrence probability of each country. The selection of a building code automatically fixes the spectrum to be used.
Zv	Velocity-related seismic zone
Za	Acceleration-related seismic zone
Zonal velocity ratio, v	Horizontal velocity at ground level for this zone, expressed in m/s units.
Calibration factor, U	Factor representing level of protection based on experience, as described at paragraph 4.1.9.1 4).
Importance factor, I	Seismic importance factor of the structure, as described at paragraph 4.1.9.1 10): Essential public services: 1,5 School buildings: 1,3 Other buildings: 1,0
Foundation factor, F	Foundation factor, as specified at paragraph 4.1.9.1 11). Refer to topic: "Foundation Factor, F"
Total height, hn	Total height of the structure from the base, where the base of the structure is corresponding to the level at which horizontal earthquake motions are considered to be imparted to the structure.
Number of stories, N	Total number of stories above the mean exterior grade up to the total height hn.
Spectral Analysis	
Accidental Torsion	Proportion of "V" to apply as accidental torsional effects during spectral and time history analyses.
Modal Combination	Method of calculation used to evaluate likely internal stresses in elements. Choose the SRSS or CQC method. (SRSS: Square Root of Sum of Squares. CQC: Complete Quadratic Combination.) Refer to topic The CQC Method.
Rounding for levels	Tolerance that is used to distinguish a dynamic level from another. If the distance between two levels is within this tolerance, seismic loads will be merged.

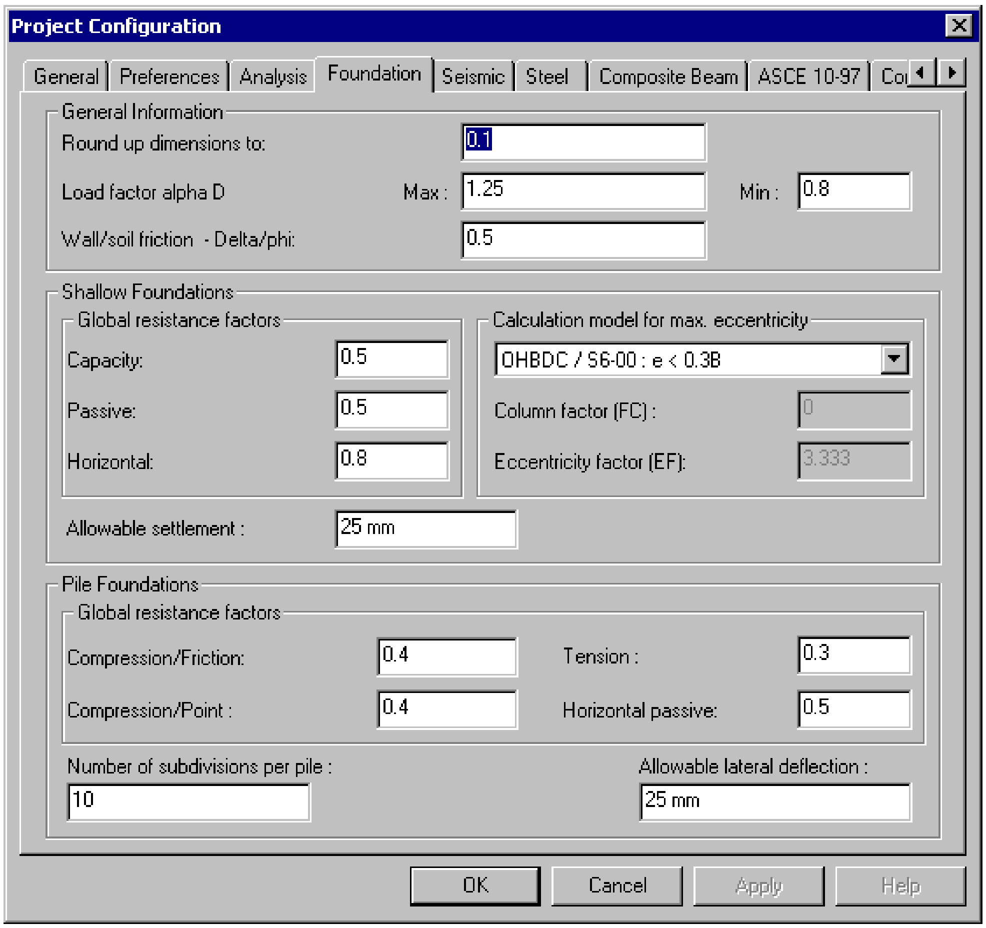
Heading	Definition
Levels c/c of floors	Seismic levels will be considered c/c of floors.
Add inelastic effects	Activate this option to add inelastic effects in analyses (P-Delta), as specified at paragraph 4.1.9.1.28).
Add ductility effects [/¯\]	Activate this box to consider ductile frames for a seismic steel design according to section 27 of S16-01 standard.
Time History Analysis	
Accelerogram	Click on the button to open a selection tree that allows you to select an accelerogram. The accelerogram is graphically shown and can be printed with the toolbar on top of the graphic.
Duration	Fix a maximum time in seconds for applying the accelerogram (This time shall be less than or equal to 200 sec.). Be careful with this parameter because the time history analysis calculation can go on for a very long time.
Time pitch	Time pitch of the selected accelerogram. If you type in another value, VisualDesign will use this time. If you select another accelerogram and want to use its time pitch, enter a value of zero.
Save node displacements	Activate this option to save the time responses for node displacements in VisualDesign results file ".vr1". Look at note 1 below.
Maximum Accelerations (g)	
Horizontal	Maximum horizontal acceleration that will be considered for linear and non-linear time history analysis. It can also be required for a spectral analysis if some vibration mode(s) act towards the gravity axis.
Vertical	Maximum vertical acceleration that will be considered for linear and non-linear time history analysis. It can also be required for a spectral analysis if some vibration mode(s) act towards the gravity axis.
Non-linear Time History Analysis	
Tolerance	Tolerance that applies to the tension and compression force in elastoplastic members and compared to the maximum values that a *Pall* system (friction dampers) can absorb.
Add vertical effects	Activate this box to statically add the contribution of vertical effects to the structure for the non-linear analysis.

Note 1: Save Node Displacements

If you prefer to save only a few nodes to shorten the time of analysis, select the desired nodes before launching the analysis. Then, from menu **Results/ Time History/ Nodes Displacements**, observe **Node displacements in time**.

Foundation Tab

Parameters used for the design of shallow foundations (soil pressure, friction angle, cohesion and dimension of foundation footings), as well as those applied to the verification of pile foundations are listed in the **Foundation** tab of **Project Configuration**.



Heading	Definition
General Information	
Round-off Dimensions to:	Allow you to round off dimensions of the foundation according to a specified value.

Heading	Definition
Load Factor Alpha D	Max: If the load factor for dead load is greater than 1.0 for ULS, VisualDesign™ will used this load factor. Min: If the load factor for dead load is smaller than 1.0 for ULS, VisualDesign™ will used this load factor.
Friction Wall/Soil - Delta/Phi	Specify the reduction (in decimals) to apply to the angle of friction, delta (δ), of a granular soil against a concrete wall. According to *Bowles*, this angle can be considered as the angle of internal friction of the soil, phi (Φ), for concrete materials. The delta angle is used to calculate the active (Pa) and passive (Pp) earth pressure. Some standards reduce this angle by a factor of 2/3 or 50%.
Shallow Foundations	
Global Resistance Factor	USD global resistance factors replace the safety factor used in the allowable stress method. For a shallow foundation, a global resistance factor of 0.50 equals to a safety factor of 3 when the ASD method is used. Default values are: ϕ capacity = 0.5 ϕ passive = 0.5 ϕ horizontal = 0.8
Calculation Method for Max Eccentricity	Choose a model for calculating the maximum allowable eccentricity. Three models are proposed: 1. Model OHDBC: $e < 0,3B$ 2. Model ACI 318: $B > 4e + w$ 3. Custom model: $B > FE(e) + FC(w)$ where "w" is the column width.
Column Factor (FC) and	If you chose a custom model, enter FC value needed to compute B.
Eccentricity Factor (EF)	If you chose a custom model, enter FE value needed to compute B.
Allowable Settlement	Allowable settlement of spread footings for the analysis of limit conditions during service. (Usually, 25mm)

Heading	Definition
Pile Foundations	
Global Resistance Factor	USD global resistance factors replace the safety factor used in the allowable stress method. For a pile foundation, a global resistance factor of 0.40 equals a safety factor of 3 when the ASD method is used. Default values are: ϕcs compression/friction = 0.4 ϕcb compression/point = 0.4 ϕts tension = 0.3 ϕ passive horizontal = 0.5
Number of subdivisions per pile	Specify pile number of subdivisions for the display of internal stress diagrams.
Allowable Lateral Deflection	Specify allowable pile lateral deflection. Generally equal to 25 mm

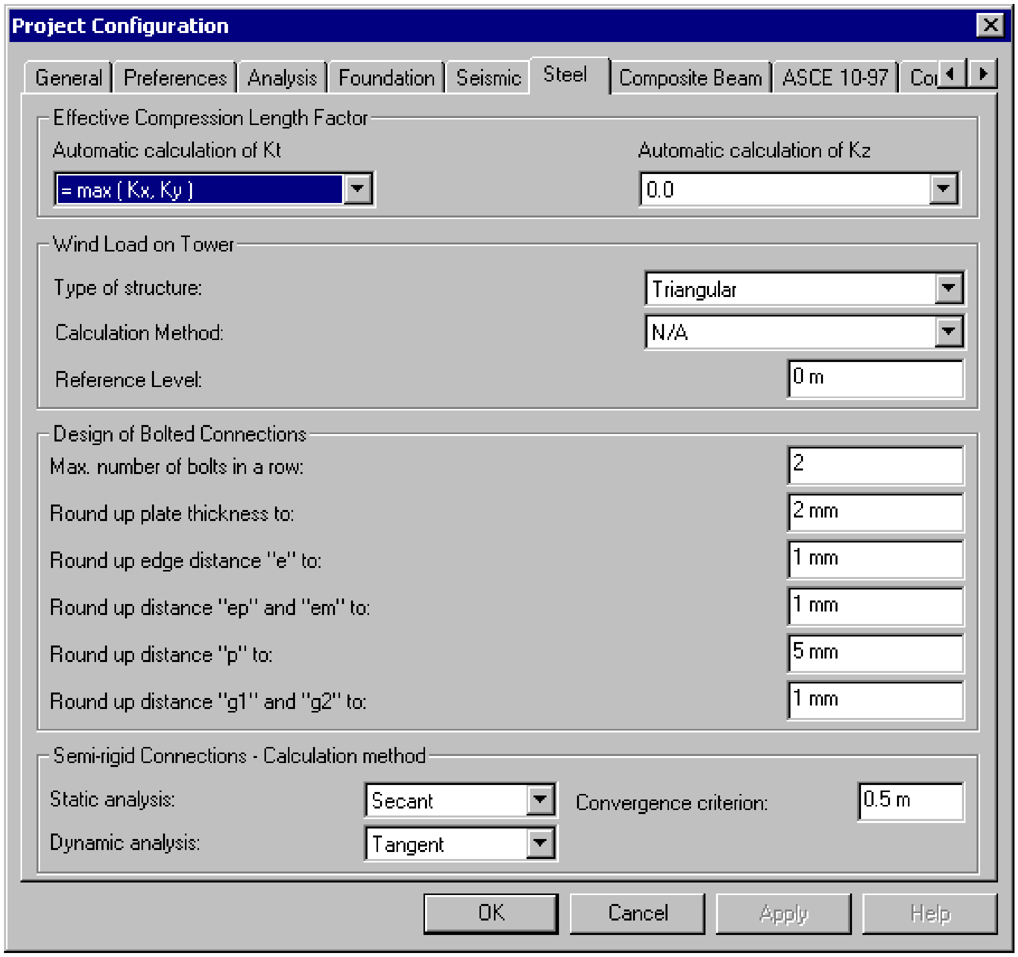
See also

Project Configuration
The Foundation Design Module
The Foundation Factor F

Steel tab

This tab will be part of the **Project Configuration** dialog box if you own the **Steel Design Module.**

This tab includes parameters that have to be specified before launching a steel design. If you own the **Tower Design Module**, you have to select a calculation method for wind loads before defining an "Auto-wind" type of loads in the **Loads Definition** dialog box. If you want to design bolted connections, consult the default values in section "Design of Bolted Connections".



Heading	Definition
Effective Compression Length Factor	
Automatic Calculation of Kt	Among the list box, choose the value that will be considered for the automatic calculation of buckling length factor due to torsion, Kt: 0.0, Kx, Ky, max. (Kx, Ky) ou min. (Kx, Ky).
Automatic Calculation of Kz	Among the list box, choose the value that will be considered for the automatic calculation of effective compression length factor on major axis, Kz: 0.0, Kx, Ky, max. (Kx, Ky) ou min. (Kx, Ky).
Wind Load on Towers	
Type of structure	Select the type of tower: Square tower or Triangular tower for the calculation of drag factor *Cd* (Clause 4.9 of CAN/CSA-S37-01).

Heading	Definition
Calculation Method	Choose CAN/CSA-S37-01 standard method for calculation of wind load or Environment Canada. A method must be selected before using the generator of wind loads. By default, it is set to Not applicable
Reference Level	Select the reference level (base of structure) from which the wind load will be applied.
Design of Bolted Connections	
Max. number of bolts in a line	When this number of bolts in line is exceeded when designing connections, VisualDesign will change the bolts layout that is *in line* to a *staggered* layout.
Round up the plate thickness to:	Specify a rounding for the design of plate thickness.
Round up edge distance "e" to:	Specify a rounding for the transverse edge distance measured from free edge of member to nearest bolt hole.
Round up "ep" and "em" distances to:	Specify a rounding for these distances for the design of bolted connections.
Round up "p" distance to:	Specify a rounding for this distance for the design of bolted connections.
Round up "g1" and "g2" distances to:	Specify a rounding for these distances for the design of bolted connections.

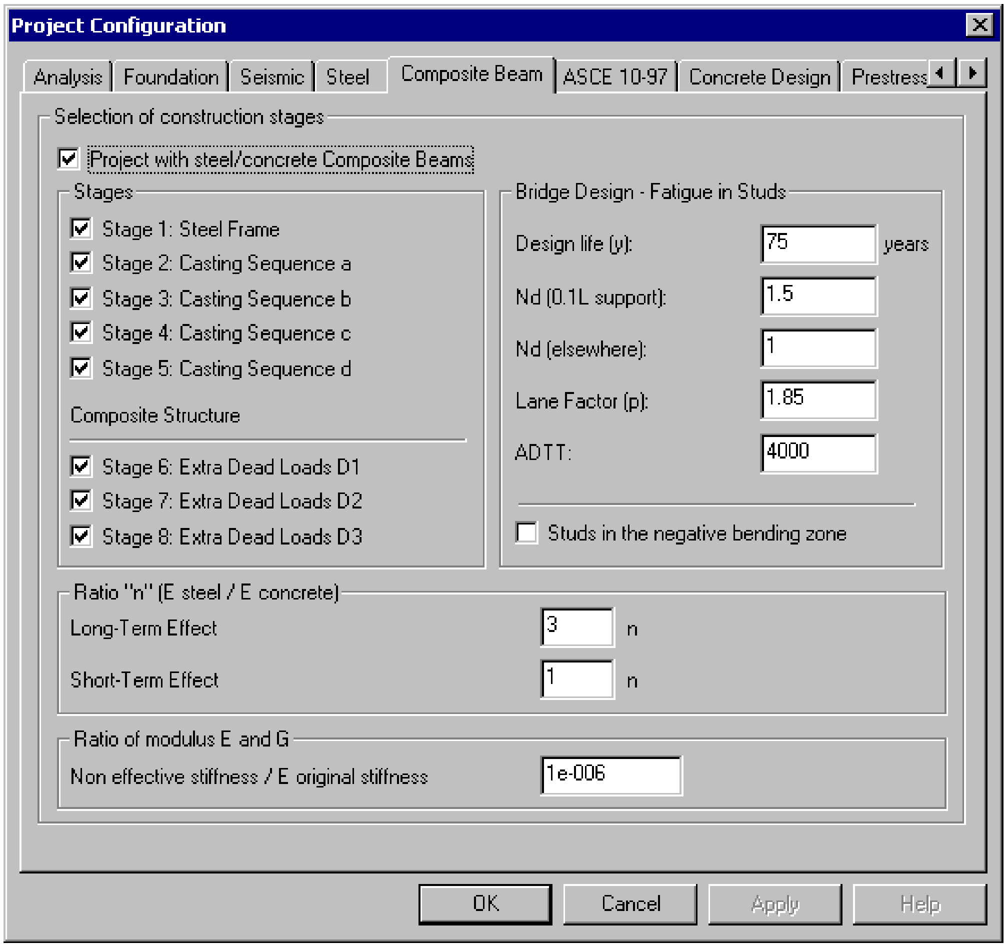
See also

Tower Design Module
Wind Loads Definition
Generating Wind Loads
Automatic Calculation of Kx, Ky, Kt and Kz
Steel Design Module
Bolted Connections Spreadsheet

Composite Beam tab

This tab is available for users owning the Steel Design module.

Activate the " Project with Steel/Concrete Composite Beams" box that is located in the upper part of the dialog box to activate construction stages.



You need to define construction stages if you need to:

- Design or verify composite beams that are not shored;
- Consider the casting sequences for concrete slabs (rectangular and triangular plates)
- Obtain an accurate calculation of deflections considering the ratio "n" for long-term deformations.

Results will depend on these construction stages.

E steel/E concrete Modulus ratio "n":

Long Term: Ratio *n* is equal to 3.0 for the calculation of long-term deflection under permanent loads, and inflexion points.

Short Term: Ratio *n* is equal to 1.0 for the calculation of forces due to live loads considered for a short-term period.

Ratio of modulus E and G

Non effective stiffness / E original stiffness: enter this ratio to consider the difference between the stiffness of liquid concrete and solid concrete for the analysis of construction stages.

Studs for Bridge Design – Fatigue

This topic is based on Code S6-00, clause 10.17.2, titled **Live Load-induced Fatigue**.

Here is a description of each field:

Field	Description
Design life (y)	Design life equal to 75 years unless otherwise specified.
Nd (0.1L support)	Refer to Table 10.17.2.3b) **Values of Nd** - Code S6-00.
Nd (elsewhere)	
Lane factor (p)	p is 1.0, 0.85 or 0.8 for the cases of 1, 2, or 3 or more lanes available to trucks, respectively, and ADTT is as given in Table 10.17.2.3 c) – Code S6-00.
ADTT	**Average Daily Truck Traffic**. Refer to table 10.17.2.3 c) Code S6-00.
Studs in the zone of M-	Activate this box if you wish to consider studs in the negative bending zone.

Table 10.17.2.3b) Values of Nd - Code S6-00

Longitudinal Members	**Span Length, L = 12 m**	**Span Length, L < 12 m**
Simple-Span Girders	1.0	2.0
Continuous Girders		
1. Near interior support (within 0.1L on either side)	1.5	2.0
2. All other locations	1.0	2.0
Cantilever Girders	5.0	5.0
Trusses	1.0	1.0
Transverse Members	**Spacing = 6 m**	**Spacing < 6 m**
All Cases	1.0	2.0

Table 10.17.2.3 c) Average Daily Truck Traffic (ADTT) – Code S6-00

Class of highway	**ADTT**
A	4000
B	1000
C	250
D	50

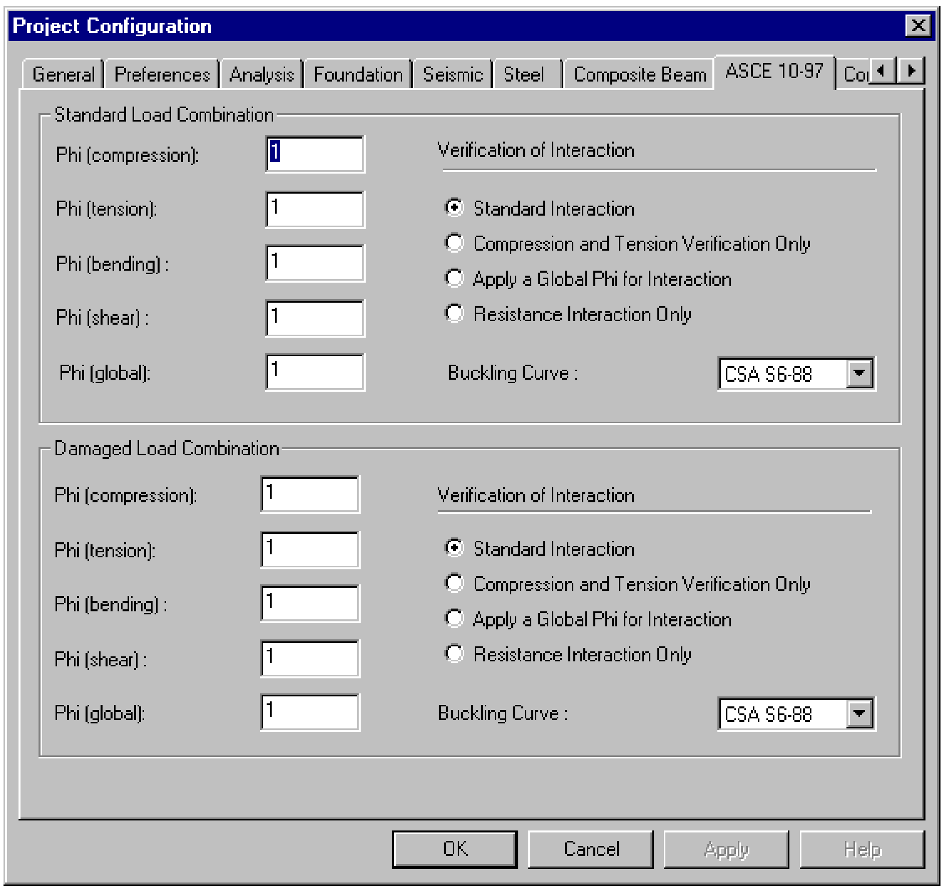
See also

Steel Design Module
Composite Beams
Thermal Gradient
Construction Stages
Calculation Method for Composite Beams
Defining a Member as Composite

ASCE-10-97 Tab

This tab allows the user to define a parametric code for the verification or design of members. The specified Phi (resistance factor) coefficients are used to calculate member resistance. The base code for the calculation is according to CAN/CSA-S16-01 code.

To launch a design or verification according to this standard, select ASCE specification in the **Steel Design** tab (Specification field)



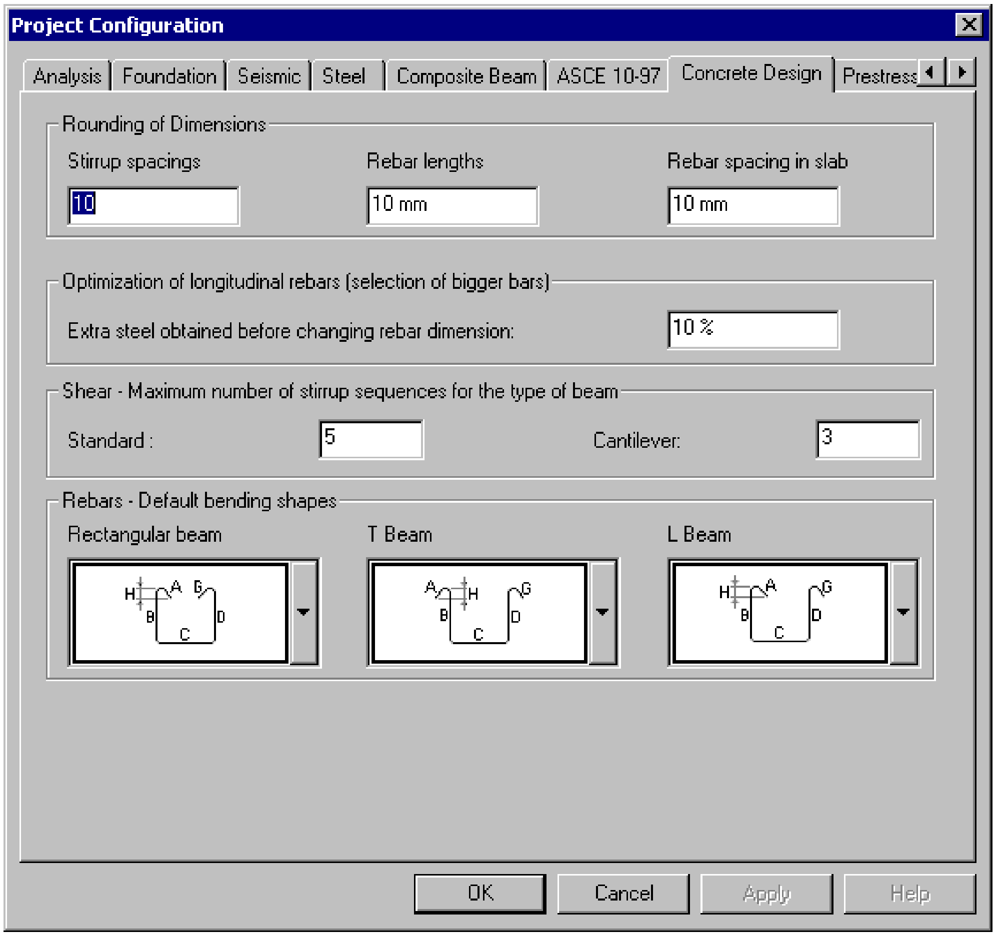
The following table gives a definition of the headings included in this tab:

Heading	Description
Standard Load Combination	**(Phi factor to apply to standard load combinations)**
Phi (compression)	Resistance factor applied to compression
Phi (tension)	Resistance factor applied to tension
Phi (bending)	Resistance factor applied to bending

Heading	Description
Phi (shear)	Resistance factor applied to shear
Phi (global)	Global resistance applied to the equation
Verification of Interaction	
Standard Interaction	Check this radio button for standard interaction verification.
Compression and Tension Verification Only	Check this radio button for compression and tension verification only.
Apply a Global Phi on the Interaction	Check this radio button to apply a global phi on the interaction.
Resistance Interaction Only	Check this radio button to verify resistance interaction only.
Buckling Curve	Select the code from which the buckling curve will be calculated.
Damaged Load Combination (Phi factor to apply to damaged load combinations)	
Phi (compression)	Resistance factor applied to compression
Phi (tension)	Resistance factor applied to tension
Phi (bending)	Resistance factor applied to bending
Phi (shear)	Resistance factor applied to shear
Phi (global)	Global resistance applied to equation
Verification of Interaction	
Standard Interaction	Check this radio button for standard interaction verification.
Compression and Tension Verification Only	Check this radio button for compression and tension verification only.
Apply a Global Phi on the Interaction	Check this radio button to apply a global phi on the interaction.
Resistance Interaction Only	Check this radio button to verify resistance interaction only
Buckling Curve	Select the code from which the buckling curve will be calculated.

Concrete Design Tab

Select this tab in the **Project Configuration** dialog box and specify default values to be used for a reinforced concrete design of your structure.



See the table below to know the definition of headings included in this tab.

Heading	Description
Rounding of Dimensions	
Stirrup spacing	Specify a rounding for the calculation of stirrup spacing.
Rebar lengths	Specify a rounding for the calculation of bar lengths.
Rebar spacing in slabs	Specify a rounding for the spacing of rebars in slabs.

Heading	Description
Optimization of Longitudinal Rebars	Specify the amount of extra steel that would be tolerated until the program changes rebars in the design of reinforcement.
Shear – Maximum Number of Stirrup Sequences	
Standard Beam	Fix the maximum number of stirrup sequences that you wish to have for a beam located between two supports.
Cantilever	Fix the maximum number of stirrup sequences that you wish to have for a cantilever beam.
Rebars – Default Bend Shape	
Rectangular Beam	In the drop-down list box, choose the default bend shape that will be used for the design of rectangular beam.
T Beam	In the drop-down list box, choose the default bend shape that will be used for the design of T beam.
L Beam	In the drop-down list box, choose the default bend shape that will be used for the design of L beam.

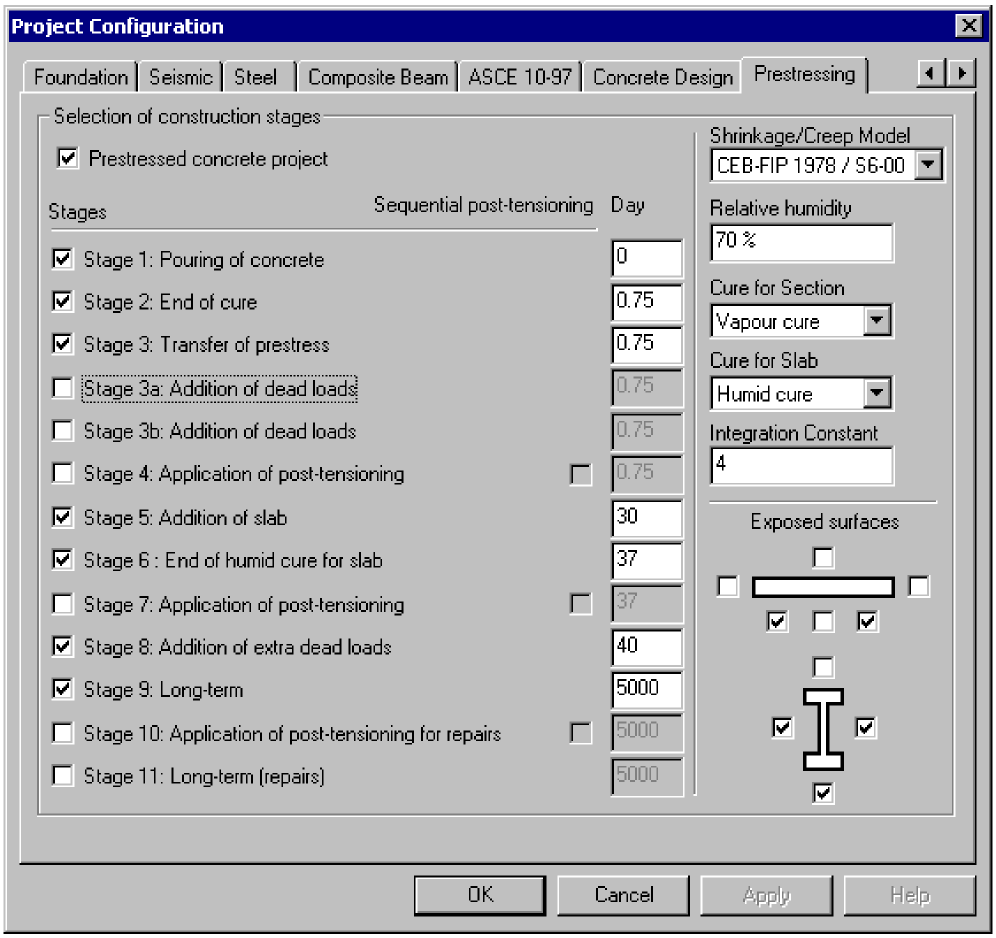
See also

Effective Stiffness
Member tab
Reinforced Concrete Design module
Bending Shapes

Prestressing tab

You must complete this dialog box if your structure has prestressed concrete elements (semi-continuous pre-tensioned and/or post-tensioned elements). Call this dialog box by selecting **Project Configuration** under **File** menu.

You must check the box "Prestressed Concrete Project" in the upper part of the dialog box in order to activate the construction stages below. Otherwise, you will not be allowed to define cable groups.



Then, define the construction stages by ticking off the appropriate boxes. Below the field "Day", enter the number of days (which is cumulative) where each stage will be applied. Superposition of results will be automatically done according to construction stages. The information will be used to calculate prestress losses in cables due to creep and shrinkage.

Stages 4, 7 and 10 correspond to post-tensioning stages. Post-tensioning can be sequential or not. If sequential, cables are not jacked at the same time. Example: a cable is made of three sheaths. The first sheath is jacked. It will not create any loss of prestress in the remaining cables. When the second sheath is jacked, the concrete will shrink and will cause losses of prestress in the already jacked cables. The same thing will happen at the jacking of the third sheath.

Note For each construction stage, you must create a corresponding load combination. Each stage load combination must have a "Construction Stage" status. If stage load combinations are not compatible with those defined in the Prestressing tab (Project Configuration), warning messages will appear on your screen..

Shrinkage and Creep Effects

- Choose a shrinkage/creep model in the drop-down list box: CEB-FIP 1978/S6-00, ACI 203, or AFNOR 1999.
- Enter percentage of relative humidity at this location.
- Choose a type of cure for the section and slab.
- Specify the integration constant (Default value is 4.0). If you want more precision in the calculation of shrinkage and creep effects, reduce this value. The value may range from 1.0 to 10.0. However, the more the value is small and the more time it will take for the calculation.
- In "Exposed Surfaces" section, tick off boxes that represent surfaces that are exposed to air. This information will be use for the calculation of shrinkage and creep effects.

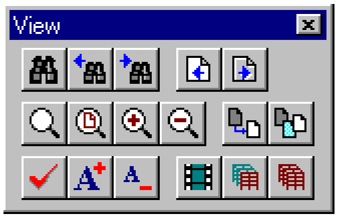
See also

Prestressed Concrete Module
Load Cases
Construction Stage Load Combinations
Other Load Combinations

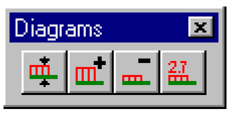
View Functionalities

View Menu and Toolbars

VisualDesign **View** menu includes the following functions and toolbars:



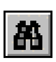
First Row: Camera, Previous Camera, Next Camera, Previous View and Next View;
Second row: Global Zoom, Zoom Window, Zoom +, Zoom –, Static Pan and Dynamic Pan;
Third row: View Options, Increase Font size, Reduce Font size, Animation, Mask and Unmask.



Diagrams Toolbar: Automatic scaling, Increase amplitude, Reduce amplitude, Numerical values.

Perspective view, Tools Toolbars and Status Bar are functions included in the **View** menu (no corresponding icons) and will be explain further in this section.

Camera

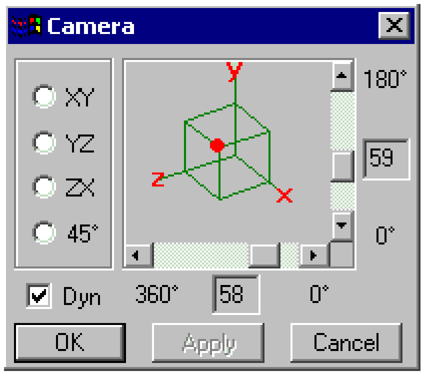


The "Camera" icon of View toolbar

VisualDesign™ allows visualizing the structure from a variety of angles. Select the **Dynamic** option to get a real-time display.

You may rotate the camera using the two scroll bars in the dialog box. The horizontal scroll bar corresponds to θ and allows for a 0° to 360° rotation, whereas the vertical scroll bar corresponds to ϕ and allows for a 0° to 180° rotation.

The camera angle moves one unit at a time when you click one of the arrows of the scroll bar, and 15 units every time you click above the elevator. You may also scroll the angle quickly by gliding the selection cursor along the scroll-bar slider.



You may also choose standard work planes such as **XY**, **YZ** or **XZ.** Activate one of the radio buttons to obtain respectively, coordinates: (0,0,1000), (1000,0,0) and (0,1000,0).

The displayed cube shows the selected camera angle. The red point indicates the closest point from you and helps you orient the structure.

When you select		You get
XY Plane	(0,0,1000)	A *z*-axis perpendicular to the screen plane and pointing in your direction.
YZ Plane	(1000,0,0)	An *x*-axis perpendicular to the screen plane and pointing in your direction.
XZ Plane	(0,1000,0)	A *y*-axis perpendicular to the screen plane and pointing in your direction.

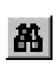
Note that whenever you modify the camera angle, you will automatically return to **Global Zoom**, no matter what zoom mode you were in to begin with.

See also

Previous Camera

Next Camera

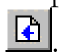
Modifying Camera Angle

- Do one of the following:
  - Click the icon  on View toolbar.
  - Choose **Camera** from the **View** menu.
- Work in XY, YZ, or XZ plane by activating the appropriate radio button.

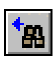
You are allowed to modify default coordinates.

- Use the horizontal and vertical scroll bar to rotate the camera in 3D spherical coordinates system. The horizontal scroll bar corresponds to θ and allows for a 0º to 360º rotation, whereas the vertical scroll bar corresponds to ϕ and allows for a 0º to 180º rotation.
- Select the "Apply" radio button to view the effects of the new camera angle on-screen.
- Press "OK" to validate your choices or "Cancel" to annul the operation.

Note: If you have already activated the **Apply** option, you will not return to the previous coordinates if you press **Cancel**.

- Return to the first viewpoint by selecting **Previous View** from **View** menu or clicking the icon .

Previous Camera



The "Previous Camera" icon of View toolbar

The **Previous Camera** function allows you going back to the previous camera angle. VisualDesign™ will then display the entire structure (Global Zoom) with the parameters applicable when the camera angle was chosen.

See also

Camera

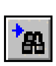
Next Camera

Going Back to Previous View

Going Back to Previous Camera Angle

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Previous Camera** from **View** menu.

Next Camera



The "Next Camera " icon of View toolbar

The **Next Camera** function is available only if the Previous Camera function has been activated before. The structure will be viewed from the same angle as it was when selecting **Previous Camera**.

Like the **Previous Camera** function, it allows viewing the structure in **Global Zoom** mode. Parameters are the same as those applicable at the time of the camera configuration.

See also

Camera

Previous Camera

Going to Next Camera

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Next Camera** from **View** menu.

Zoom Window

The "Zoom Window" icon of View toolbar

The **Zoom Window** function allows viewing in full-screen any part of the structure that you have selected by way of the Zoom Window cursor. The selected part of the structure will be magnified to scale and centred on the screen.

The cursor will appear as a cross when activating **Zoom Window**.

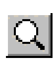
See also

Zoom +

Zoom -

Global Zoom

Viewing a Part of a Structure in Full-Screen

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Zoom Window** from **View** menu.
- Use the cursor and draw a window around the part that you want to magnify.

To draw a window, press the left mouse button and keep it down while you move the mouse diagonally. The elements that are contained within the window will be magnified and centred on the screen.

When the chosen part is magnified, VisualDesign™ will return by default to **Extended-Window** mode.

Note. To go back to previous view, click the icon .

Global Zoom

The "Global Zoom" icon of View toolbar

The **Global Zoom** function reduces the structure to scale, letting you view the whole structure on the screen.

Global Zoom Margins

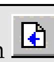
By default, VisualDesign puts margins that equal to 25% of the size of the structure, all around the screen image. It is possible to modify these margins. Use the following shortcut keys to increase or reduce margins:

Shift+ Global Zoom: Reduces margins so image will be bigger. **Ctrl+ Global Zoom**: Increases margins so image will be smaller.

Viewing the Structure in Full-screen

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Global Zoom** from the **View** menu.

When the part that you have selected has been magnified, VisualDesign™ will return by default to **Extended-Window** mode.

Note. To go back to previous view, click the icon .

Zoom +

The "Zoom+" icon of View toolbar

The **Zoom +** function allows you stretching the content of the current window by 200%. The content of the window is stretched to scale and centred on screen.

Magnifying the Content of Current Window to 200%

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Zoom +** from the **View** menu.

- Mouse scroll wheel: Place the mouse cursor on the spot you want to magnify and scroll the wheel up.

When the content of the current window has been magnified by 200%, VisualDesign™ returns by default to the **Extended-Window** selection mode.

Note. To go back to previous view, click the icon  or the icon .

Zoom -

The "Zoom –" icon of View toolbar

The **Zoom -** function allows you reducing the content of the current window by 50%. The content is reduced to scale and centred on the screen.

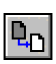
Reducing the Content of Current Window by Half

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Zoom -** from the **View** menu.
  - Mouse scroll wheel: Place the mouse cursor on the spot you want to magnify and scroll the wheel down.

When the content of the current window has been reduced by 50%, VisualDesign™ returns by default to the **Extended-Selection** mode.

Note. To go back to previous view, click the icon  or the icon .

Static Pan



The "Static Pan" icon of View toolbar

The **Static Pan** command allows you moving what you see on screen while letting you monitor the displacement vector.

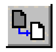
When you work in Static Pan mode, the pointer looks like four arrows.

Click anywhere on the structure. The point will become the displacement vector's starting point. Slide the cursor to the target position. When you let go the left mouse button, the starting point will move onto the target point, taking the whole structure with it.

See also

Dynamic Pan
Using the Static Pan

Using the Static Pan

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Static Pan** from **View** menu.
- Use the hand pointer to click on the structure.
- Keep the left mouse button pressed and slide the pointer to where you want to move the structure.
- Release the left mouse button. Your selection's starting point will move onto the target point, taking the whole structure with it.

When the move has been completed, VisualDesign™ returns by default to the **Extended-Window** selection mode.

Dynamic Pan

The "Dynamic Pan" icon of View toolbar

The **Dynamic Pan** function allows you moving in "real time" what you see on screen, with reference to the pointer movement.

When you activate the **Dynamic Pan** function, the pointer will look like a pointing hand.

Click anywhere on the structure. Press the left mouse button and slide the hand to move the structure to the destination point. Then, release the left mouse button.

See also
Static Pan

Using the Dynamic Pan

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Dynamic Pan** from **View** menu.
- Use the hand pointer to click a point on the structure and slide it to its new position.
- Release the left mouse button to fix the structure new location.

Once the move has been completed, VisualDesign™ returns by default to the **Extended-Selection** mode.

Previous View

The "Previous View" icon of View toolbar

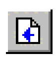
The **Previous View** command allows you going back to the last displayed screen: the camera angle, the zoom and the pan will be the same.

Please note that if you use the static or dynamic pan with the same zoom and camera angle more than once during the same session, VisualDesign™ will only take account of the last pan when it takes you back to the previous view. VisualDesign™ will only memorize the last pan of a series of consecutive pans.

See also

Next view

Going Back to Previous View

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Previous View** from **View** menu.

Next View

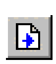
The "Next View" icon of View toolbar

The **Next View** command is only available if you have already activated the **Previous View** option. It allows you going back to the last screen. The camera angle, the zoom, and the pan will be the same as the ones used before activating the **Previous View**.

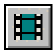
See also

Previous view

Going to the Next View

- Do one of the following:
  - Click the icon  on View toolbar.
  - Select **Next View** from **View** menu.

Animation



The "Animation" icon of View toolbar

Press this icon to animate the structure. The animation you will see on screen depends on the display options you entered (stresses, deflection, etc…) and the activation mode you are in:

- **Structure and Load Case Modes**: the animation function rotates the model according to the gravity axis.
- **Load Combination and Envelopes Modes**: the deflections and stresses are animated by multiplying them by factors varying from –1.0 to 1.0.
- **Vibration Mode**: the vibration mode is animated.

Mask

The "Mask" icon of View toolbar

This function, available in **View** menu, retains only the selected elements on the screen. It is not necessary to select contiguous elements.

To display the entire structure, select the **Unmask** function from **Edit** menu.

Note. This function is very useful when personalized selections of elements have been defined. These selections can be called back anytime and the **Mask** function will help you to check the model or consult diagrams results.

See also

Unmask
Personalized Selections of Elements

Unmask

The "Unmask" icon of View toolbar

This function, available in **View** menu, displays the whole structure after part of it has been masked.

Perspective View

Use function **Perspective** available in **View** menu to get a perspective view of the structure or use the shortcut key "S".

Mouse scroll wheel:

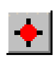
Press simultaneously the control keys [Shift] + [Ctrl] and use the scroll wheel to adjust the perspective view for very tall structures.

Increase Font Size

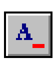
The "Increase font size" icon of View toolbar

Press this icon to magnify the font size for characters linked to elements such as numbers, length, materials, sections, areas, etc. The modified characters are those associated with the activated element (node, member, plate or floor).

For example, if you wish to increase the displayed node numbers:

- Make sure that the node numbers are displayed.
- Make sure the node  button is down.
- Press the Increase Font Size  button.

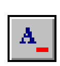
Reduce Font Size



The "Reduce font size" icon of View toolbar

Press this icon to reduce the font size for characters linked to elements such as numbers, length, materials, sections, areas, etc. The modified characters are those associated with the activated element (node, member, plate or floor).

For example, if you wish to reduce the displayed number for members:

- Make sure that the member numbers are displayed.
- Make sure the Member icon is activated.
- Press the **Reduce Font Size**  button.

Control Over the Image on Your Screen

In this table, you will find short cut keys to help you managing the screen image.

Short-cut key	Mouse scroll wheel	Action
PgDn	N/A	To obtain a XY plan view of the structure.
End	N/A	To obtain a XZ plan view of the structure.
Home	N/A	To obtain an YZ plan view of the structure.
PgUp	N/A	To obtain a 45 degree isometric view of the structure.
Vertical arrow	N/A	1-degree rotation of the XZ plane of the structure (if Y is gravity axis).
Horizontal arrow	N/A	1-degree rotation about gravity axis.
[Ctrl] + Vertical arrow	Press down the [Shift] key and use the scroll wheel.	15 degrees rotation of the XZ plane of the structure (if Y is gravity axis).
[Ctrl] + Horizontal arrow	Press down the [Ctrl] key and use the scroll wheel.	15 degrees rotation about gravity axis.
[Shift] + Horizontal arrow [Shift] + Vertical arrow	N/A	Dynamic Pan in the direction of arrow.
[Shift] + [Ctrl] + Horizontal arrow [Shift] + [Ctrl] + Vertical arrow	N/A	Bigger Dynamic Pan in the direction of arrow.
+ or -	Scroll the wheel in both directions to zoom in or out	Zoom + or Zoom -
[Shift] + Global Zoom	N/A	Reduce margins around displayed image on the screen. Therefore, the image will be bigger.

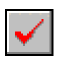
Short-cut key	Mouse scroll wheel	Action
[Ctrl] + Global Zoom	N/A	Increase margins around displayed image on the screen. So, the image will be smaller
N/A	Press down the [Shift] + [Ctrl] keys and roll the mouse wheel	Adjust the perspective view for very tall structures

See also

Camera

Perspective View

View Options Dialog Box



The "View Options" icon of View toolbar

The **View Options** command enables you to select for the types of objects you wish to see displayed on the screen. It also gives you full control on their graphic attributes presentation (colour, pen thickness, filling) and text (numbers, results values).

The **View Options** dialog box is composed of many tabs, namely the **View** tab, **Attributes**, **Limits**, **Loads**, **Results**, **FE Results** and **Colour** tabs.

See also

Displaying Legends on Screen

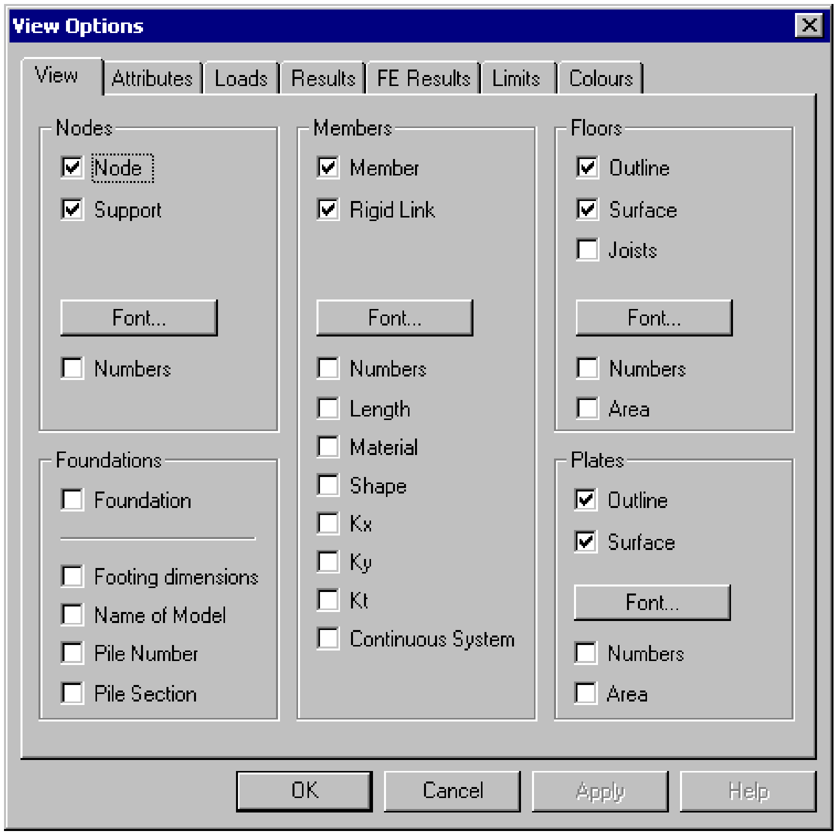
The View Options Toolbar

The View Tab

The **View** tab enables you to select for the display elements and their characteristics.

In activating or deactivating the check-off boxes corresponding to the types of elements, you will mask or display nodes, supports, members, floors, plates, and footing dimensions, model name, pile numbers and sections, and continuous system numbers. You may also choose to temporarily mask some floors components (outline, surface, joists) or plates (outline, surface) to facilitate the viewing of other characteristics.

The "Font" button allows you to set the font for the display of numbers.



Please note that some fields may be shaded if you do not possess some modules.

Note. The display of elements overrides the content of all other headings of this tab. For example, if you do not activate the "Member" box, you will not be allowed to display member characteristics included in the **Attributes** tab or **Results** tab.

See also

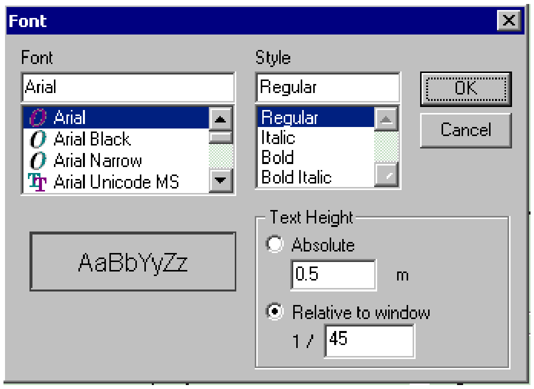
The Attributes tab

The Results tab

Font Dialog Box

Modify the font and style using drop-down list boxes. Adjust text height by activating one of the following radio buttons:

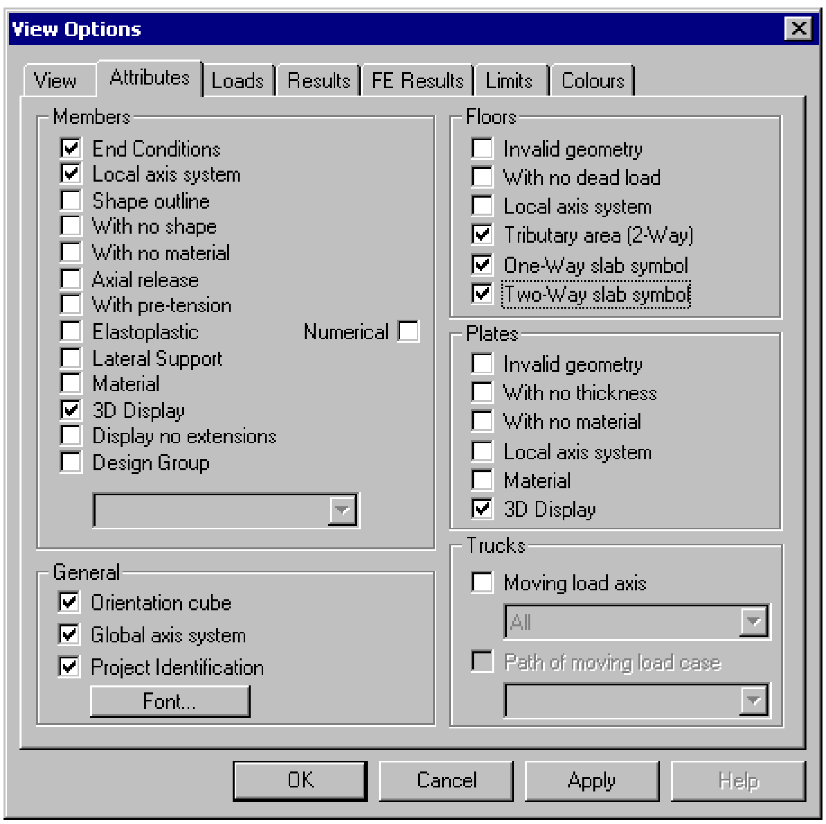
- The *Absolute* button allows fixing a height that will follow the zoom in and out of the structure.
- The *Relative to window* radio button allows adjusting the text height relatively to the size of window (text height stays the same no matter the zoom you are in).



N.B. The radio button *Relative to window* is not activated when modifying node, member, plate, floor and dimension fonts.

The Attributes Tab

The **Attributes** tab enables you to identify elements that might cause a problem at the time of analysis.



Members:

Members characteristics can be displayed, such as end conditions, local axes, shape outline or 3D member shape, members having no specified sections or material, elastoplastic members, axially released members, pre-tensioned members, members with lateral support, design group, member materials and their corresponding legend. If you checked several boxes, some have precedence over others, namely:

1. The selected object (member);
2. Members design groups;
3. Members part of the moving load axis;
4. Members with unspecified shape;
5. Material colour;
6. Members with unspecified material;
7. Members with pre-tension (in the "Structure" activation mode only);
8. Released members:
   - In the Structure activation mode: all members with axial release.
   - In the Load Combinations activation mode: members that have been released during analysis.
9. The member colour.

Floors:

For floors, you can display geometrically invalid floors, floors having no dead load, the local axes system, the tributary surfaces (two-way slabs), and symbols for two-way or one-way slabs.

Plates:

For plates, you can display geometrically invalid plates, plates having no thickness, plates having no specified material, and the plates local axes system. Use the *3D Display* option to display plates with volume.

Moving Loads:

Display all moving load axes or display one moving load axis (1, 2 or 3) at a time. When all axes are displayed, no moving load case will be available in the list box. However, when one axis is selected, moving load cases will be available but only those corresponding to this axis.

General:

In this section, choose to mask or unmask the orientation cube, the global axes system or the identification of the project

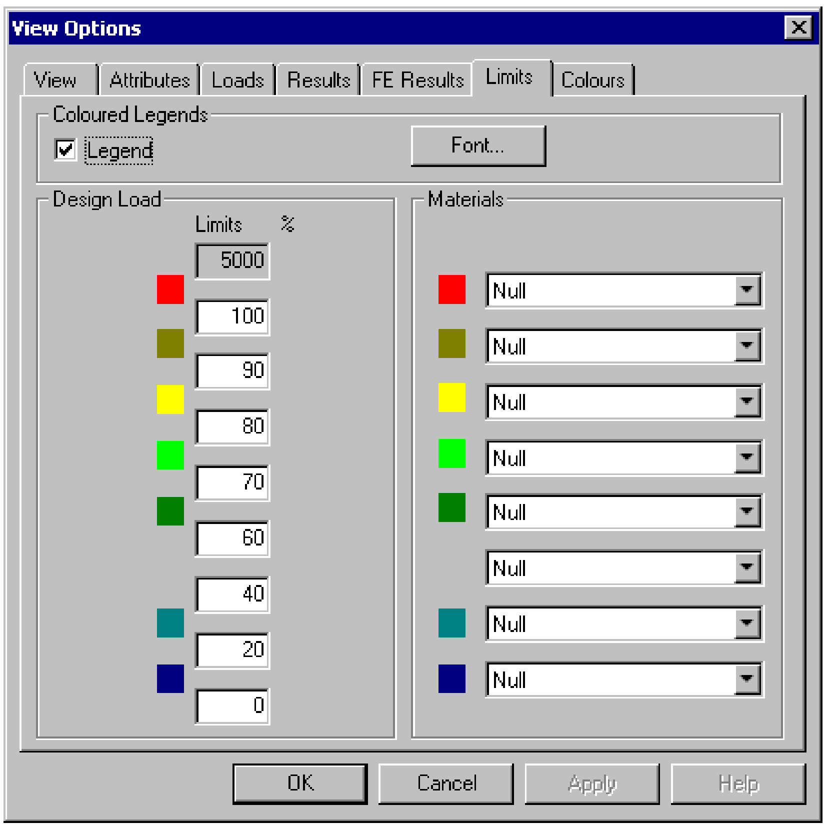
See also

Displaying Legends on Screen
Display Member Characteristics
Display Plate Characteristics
Display Floor Characteristics
Display Global Axis System

The Limits Tab

Use this tab and define eight colour ranges representing intervals for the display of members design load. You can change the associated colour of a range by clicking on the colour square and by choosing a new colour. The associated limits can be modified also. Enter ranges of your interest in each field.

You can also use eight colours to represent each material present in your project.



See also

Displaying Legends on Screen
Displaying Members' Design Load
Displaying Materials

Displaying the Legend for Materials

- Activate the Structure mode and open the **View Options**.
- Go to the **Attributes** tab and activate the *Material* option.
- Go to the **Limits** tab. Select materials and choose a colour for each material. Activate the "Legend" box. To modify the font size and style, press the "Font" button.

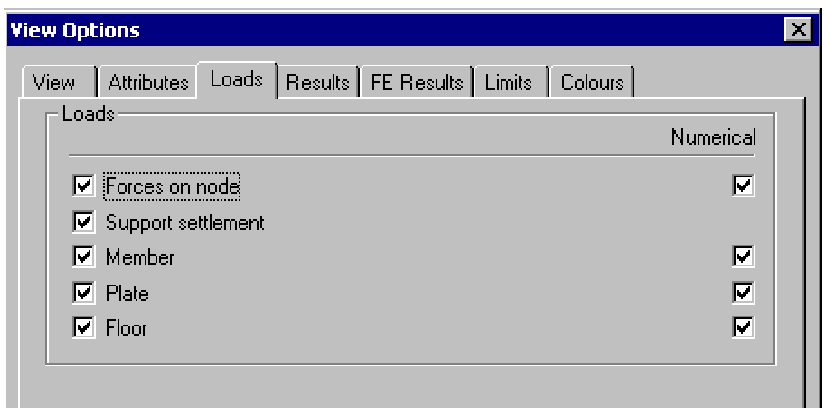
See also

The Limits Tab
Results tab
Displaying Legends on Screen

The Loads Tab

The **Loads** tab enables you to display loads on the screen.

Use this tab to display support settlements and loads applied on nodes, members, plates or floors. Tick off the *Numerical* box to display the numerical values of loads.

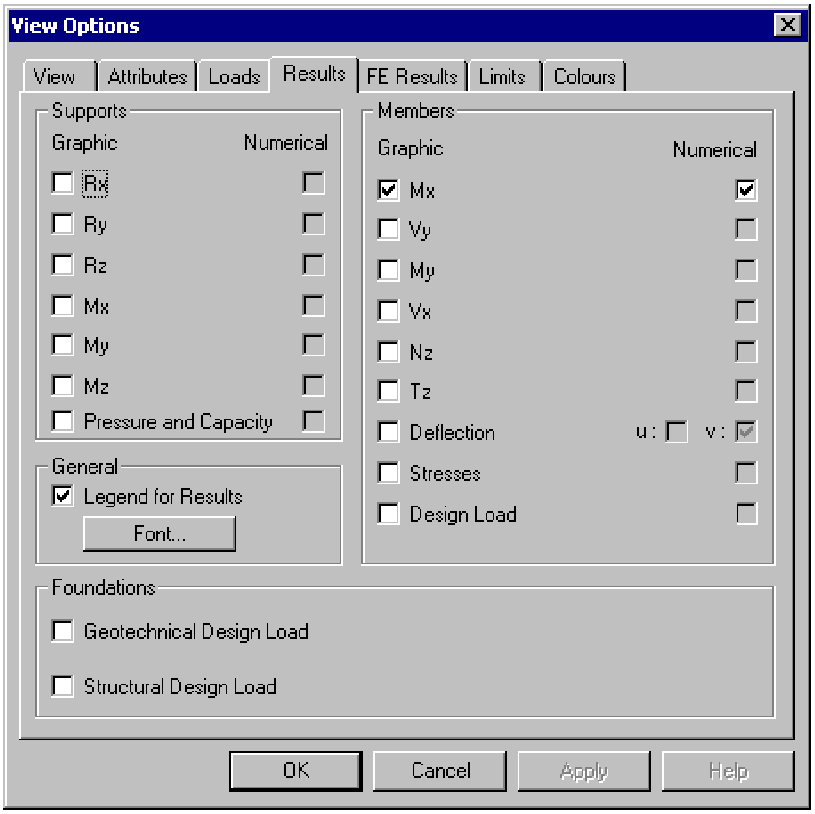


See also

Loads
Coloured Display for Loads on Supports
Coloured Display for Loads on Nodes

The Results Tab

Select the **Results** tab to graphically or numerically view static analysis results. You can display support reactions and bending moments for members. You can also display the members design load and its legend, shear stress, axial force, torsion and deflection.



Vibration Mode:

The *Deflection* box is automatically checked when the Vibration mode is activated. Consequently, if you select a mode and wish to animate it (using function Animation), deflections will be displayed.

Foundation:

Display the geotechnical and structural design loads.

Description of View Options' Results tab

Option	Description	Comment
Supports		Display numerical values by ticking off the appropriate box located below the "Numerical" title.
Rx, Ry, Rz	Display the support reaction according to global axis system.	
Mx, My, Mz	Display the bending moment at support, according to global axis system	

Option	Description	Comment
Pressure and Capacity	Display the pressure on spring support and its capacity. A coloured legend is also displayed to differentiate pressure from capacity.	Ex.: Spring supports located along a pile foundation.
Members		Display numerical values by ticking off the appropriate box located below the "Numerical" title.
Mx, My	Display the bending moment in members, for strong or weak axis, according to member local axis system.	
Vy, Vx	Display the shear forces in members, for strong or weak axis, according to member local axis system.	
Nz	Display the axial forces in members.	
Tz	Display the torsional moment in members.	
Deflection	Display the member deflections	Check the "u" and/or "v" box to display deflection numerical values for strong or weak axis.
Stresses	Display stresses in members.	
Design Load	Display the coloured design load for members and foundations.	Activate the "Legend" box in the **Limits** tab and modify the colours and values for each interval.
General		
Legend for Results	Check this box and press the "Font" button to modify the text height and font that are used in legends.	
Foundations		Display one type of result at a time.
Geotechnical Design Load	Display the coloured geotechnical design load for shallow or deep foundations.	Activate the "Legend" box in the **Limits** tab and modify the colours and values for each interval.
Structural Design Load	Display the coloured structural design load for shallow or deep foundations.	Activate the "Legend" box in the **Limits** tab and modify the colours and values for each interval.

See also

Animation
View Options Dialog Box

View Options Toolbar
Displaying Results for Members
Displaying Legends on Screen
Displaying the Pressure and Capacity of Spring Supports

Displaying Legends on Screen

MATERIALS

Use the Structure activation mode. Open the **View Option** dialog box**.**

Display the coloured legend of materials by checking the option "Material" in the **Attributes** tab and activate the "Legend" box in the **Limits** tab. To modify the font style and text height, press the Font button.

MEMBER DESIGN LOAD:

Use the Design Results activation mode. Open the **View Option** dialog box**.**

Display the coloured legend of member design load (according to the chosen colours and intervals in the **Limits** tab) by checking option "Design load" in the **Results** tab.

To modify the font style and text height, go back to the **Limits** tab and press the "Font" button.

LEGEND FOR FOUNDATIONS GEOTECHNICAL AND STRUCTURAL DESIGN LOAD:

Use the Design Results activation mode. Open the **View Option** dialog box**.**

Display the coloured legends of geotechnical or structural design load of foundations (according to the chosen colours and intervals of **Limits** tab) by checking option "Structural Design load" or "Geotechnical Design load".

To modify the font style and text height, press the "Font" button in the **Limits** tab and check the "Legend" box.

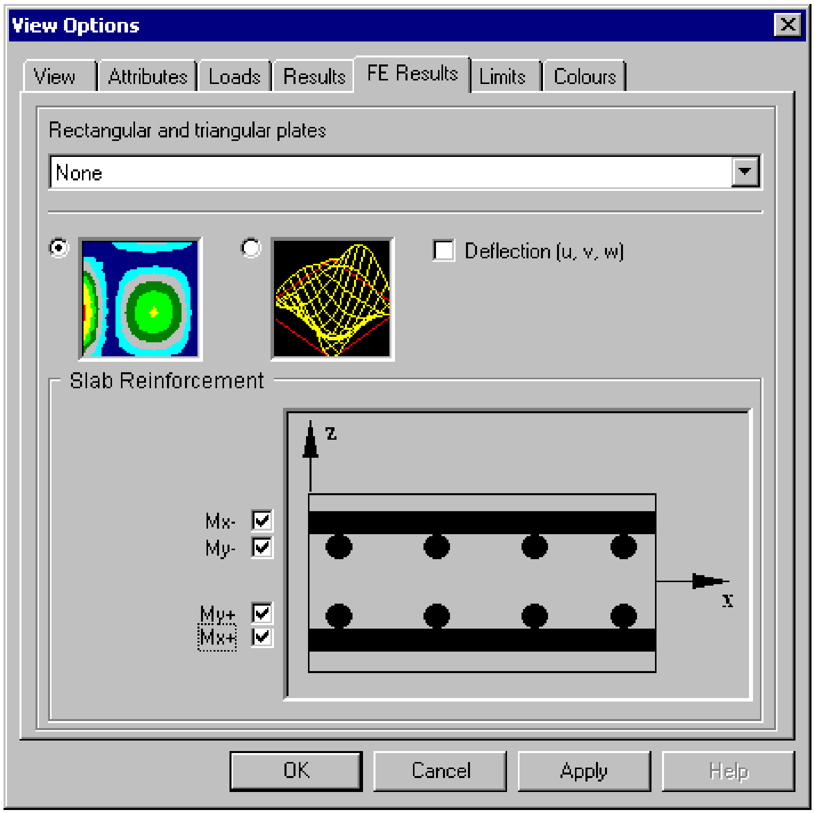
The Finite Elements Results Tab

The **Results FE** tab is useful to graphically or numerically display stresses and forces such as, bending moments, shear forces, axial forces, and deflections for rectangular and triangular plates.

Following a static analysis, select a load combination on Activation toolbar and open the **View Options** dialog box. Select the **Results FE** tab.

Select a diagram among the list box. Activate a type of display (stress/force contour or mesh, with or without deflection). Click OK.

N. B. Finite Element Analysis and results are detailed further on in this chapter.



Force/Stress Contours

Activate a load combination on Activation toolbar.

The left radio button displays graphic results in the form of stress/force contours if a graph is selected in the list box. Click any icon on Diagrams toolbar to open the **Scaling of Intervals** dialog box. This tool allows modifying the scale (upper and lower limits) for the displayed values (intervals) and colours. Refer to the topic Scaling for Intervals to learn more about this tool.

Mesh and Deflection

Activate a load combination on Activation toolbar.

The right radio button displays the mesh and allows consulting numerical results. Double click on a plate to open the **Internal forces and stresses** spreadsheet or select many plates and press the **Properties** icon to open the spreadsheet.

The check box "Deflection u, v w" displays the deflection of plates for the selected load combination. The deflection can be displayed along with the mesh or coloured stress/force contours.

Rebar Placement for 2-Way Slabs

The Reinforced Concrete Design is required.

This section applies to the display of calculated rebars for 2-way slabs. Four layers of bars can be displayed. The colour of rebars can be modified through respective reinforcement spreadsheets.

See also

Grouping Plates

Scaling for Intervals

View Options

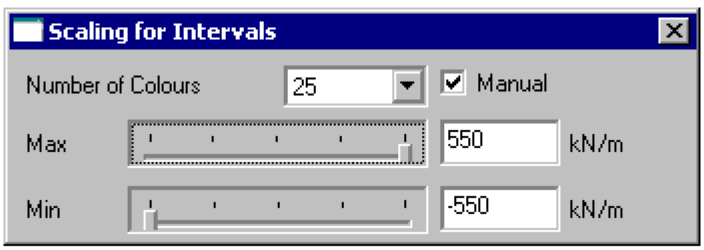
Interpreting Plates Analysis Results

Concrete Design of 2-Way Slabs

Scaling for Intervals

When coloured stress/force contours are displayed for consulting finite elements results, press an icon on Diagrams toolbar to open the **Scaling for Intervals** dialog box.

This tool allows modifying the intervals (values and number of colours) for the displayed legend.



Select a number of colours for the display (9, 25, 45, 105, 225, 525 or 1021).

The dialog box is composed of two scroll bar sliders that allow the scaling of intervals from maximum or minimum values when displaying stress/force contours (legend). Each scroll bar is subdivided into 100 units. Marks indicate a subdivision of 25 units. To move the indicator, click on it and slide the mouse at the right or left on the scroll bar. Displayed intervals will be increased or reduced as a result.

The Manual Mode:

When the *Manual* mode is activated, the absolute minimum and maximum values will still be displayed in the coloured legend but the specified value (min and max) will be used as upper and lower limits for the critical zone. If scroll bar sliders are used to modify the displayed values for intervals, the min and max specified values would be used for the modification instead of absolute values.

Therefore, critical zones (red or blue) are always delimited by the absolute value and specified value if the *Manual* mode is activated. If sliders are used, the modification will be done according to specified values.

The legend displays the lower and upper limits for the two critical zones and intermediate zones (colours) display the middle value of the interval.

Shortcut Keys:

Use keyboard shortcut keys to adjust intervals when your cursor is located in the **Max** or **Min** scroll bar.

Shortcut key	Action
[Home]	The [Home] key moves the indicator at the beginning of the scroll bar slider.
[End]	The [End] key moves the indicator at the end of the scroll bar slider.
→	This arrow moves the indicator one unit right.
←	This arrow moves the indicator one unit left.

See also

Grouping plates

The Results FE tab

Interpreting Plates Analysis Results

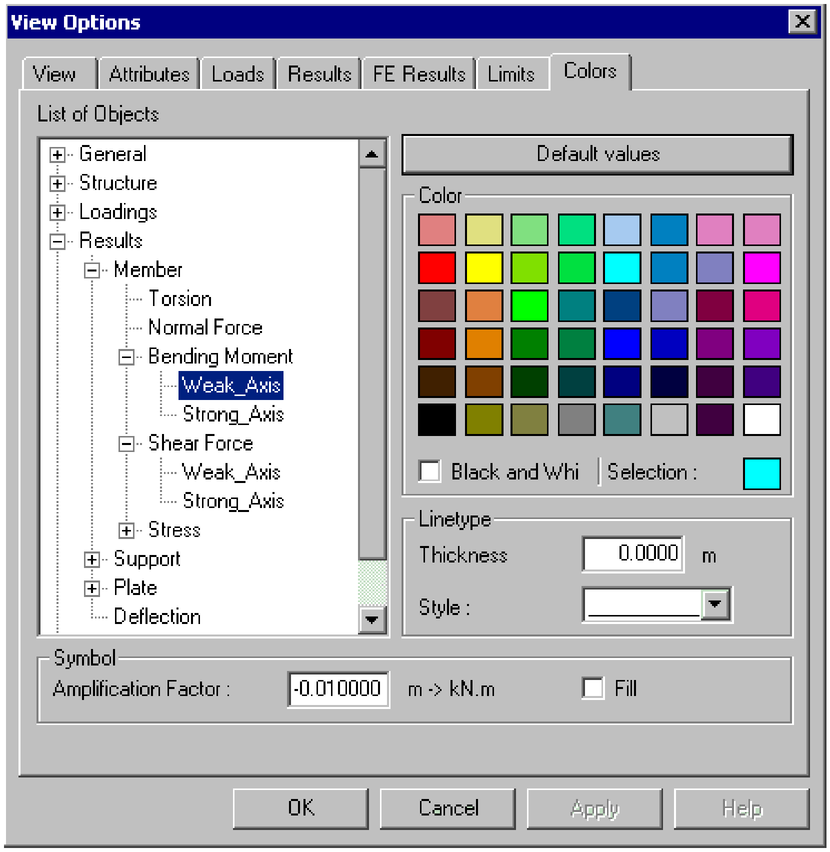
The Colours Tab

The **Colours** tab allows you to change the colour and the graphic attributes of structural elements, loads, results, foundations and moving loads.

The tab is divided into six selection trees:

- General: Colours for axes, text, print margins, background colour, selected elements, mesh, and axes systems;
- Structure: Colours for nodes, supports, members, plates, and floors;
- Loads: Colours for arrow (load symbol) that represents loads applied to nodes, members, plates, and floors;
- Results: Colours of results for members, supports, and plates;
- Foundation: Colours for the display of Design mode or Verification mode;
- Moving Loads: Colours for moving loads axis and position.

For each element in these sections, you can change the colour, the line thickness, and the symbol. Activate the *Black-and-White* option before printing using a black-and-white printer otherwise the background to the structure will be black.



See also

Graphical and Numerical Results

Coloured Results

Defining Graphic Attributes

View Options

Defining Graphic Attributes

- Select **View Options** from the **View** menu.
- Select the **Colours** tab.
- Click the expansion button for one of the trees to access the element for which you wish to modify the graphic attributes.
- To assign a colour, click one of the colours available in the "Colours" section.
- To modify the line style, choose from the options in the "Style" scroll list.
- To modify the line thickness, select a value in the *Width* box. The measuring unit for line thickness is the same as the one you selected for the coordinates. (See **Project Configuration** in the **File** menu. By default, "0" is equivalent to one pixel.)

- To modify the scale factor of an element's symbol, enter one of the values in the Scale Factor box.
- To fill in certain symbols with colour, activate the *Fill In* box.
- Press [OK] to validate your choice, click [Apply] to view the modifications without exiting the dialog box, or [Help] for a description of the different options in this dialog box.

Note. The **Preview** option lets you see how the line colour matches the background colour. You may also view the line style to see whether the form is filled with colour or not. However, the graphic representation does not take into account the thickness of the line nor the scale factor applied to the selected element.

View Options Selection Toolbar

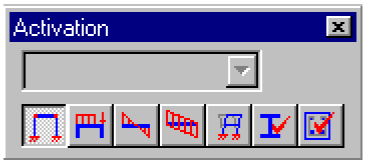
Display the **View Options** toolbar to quickly verify the structural model and results. This toolbar, which is a selection tree, include the same options as the **View Options** dialog box and will be posted at the right of the screen.

To activate or hide this toolbar, go to **View** menu and select **Toolbars**. Check or uncheck the "View options" toolbar.

The **View Options** selection tree is divided into many roots, which correspond to element (Members, Nodes, Floors, and Plates) in such a way that options are grouped together and can be selected quickly. Display options for results are also available for each element. You will also find the *Foundations* root and *General* root.

Activation Modes

The Activation Toolbar



The Activation toolbar of VisualDesign main window

With VisualDesign™ you can work under seven activation modes, namely, "Structure", "Load Case", "Load Combination", "Envelope", "Vibration Mode", "Design Results" and "Rebar Placement" mode. Before asking information about any specific element through the **Properties** function of the **Edit** menu, make sure that you have activated the right mode, since the displayed dialog box is specific to a selected mode.

For example, if you click twice on a node while you are in the "Structure" activation mode, you will get the **Node Characteristics** dialog box. If you click twice on the same node while you are working with the "Load Case" activation mode, you will get the **Loads on Node** dialog box. If you proceed the same way in the "Load Combination" activation mode, you will get the **Nodes Displacements** dialog box. Finally, if you double click on a member in the "Envelope" activation mode, you will get the envelope diagram for this member.

To visualize the structure motion under a particular vibration mode, activate the "Vibration mode", choose a mode in the "Title Selection" list box and press the "Animation" icon.

Once you have executed a steel design, the "Design Results" mode will be automatically activated. Double-clicking on a member gives you access to the **Design Results** Spreadsheet.

For those owning the **Reinforced Concrete Design** module and **Prestressed Concrete** module, activate the "Rebar Placement" activation mode to access the *Rebar Placement* window by double-clicking on any continuous system even if you had not run any analysis or design. The purpose of this is to allow you to look and modify longitudinal and transverse rebars calculated by the program or to create your own design and checking it by looking at the displayed diagrams.

See also

Description of Icons and Toolbars

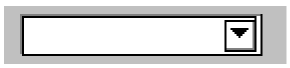
Structure Activation Mode

Load Case Activation Mode

Load Combination Activation Mode

Envelope Activation Mode
Vibration mode Activation Mode
Design Results Activation Mode
Rebar Placement Activation Mode
Display Default Spreadsheet

Title Selection

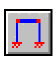


The "Title selection" box of Activation toolbar

Press this button and select a load case title, a load combination or envelope title, or vibration mode while being in the "Load Case", "Load Combination", and "Envelope" or "Vibration" activation mode.

When one of these modes is activated, the drop-down list box automatically opens to let you choose a title.

Structure Activation Mode



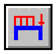
The "Structure" icon of Activation toolbar

By activating the "Structure" mode, observe or modify element characteristics such as a normal node, a support node, a continuous system, a member, a plate or a floor.

The "Structure" mode must be activated to modify element characteristics through a specific dialog box using the **Properties** function of **Edit** menu.

See also
Activation
Node Characteristics
Support Characteristics
Member Characteristics
Rectangular Plate Characteristics
Triangular Plate Characteristics
Floor Characteristics
Properties

Load Case Activation Mode



The "Load Case" icon of Activation toolbar

The "Load Case" activation mode is useful when you wish to enter or modify loads that are applied on different elements of your structure. For example, it allows setting forces and moments on nodes, support settlements (displacement / rotation), loads on member (distributed / concentrated / temperature / torsion / shrinkage), plate (pressure / temperature) and floor loads (concentrated / distributed).

Before applying loads on elements, you must first create a load case title. To do so, select **Loads/Definition**.

Press down the "Load Case" activation mode, and then select the load case title in the "Selection Title" box of the Activation toolbar.

You are now able to set or modify the loads that will be applied on each element of your structure for the selected load case.

Select an item or double click it, then select the **Properties** icon of the Edit toolbar. You will then have access to the loads dialog box that is specific to this element.

You may also set or modify loads through the spreadsheets under the **Loads** menu.

See also

Loads

Applying or modifying loads

Loads on Nodes

Loads on Supports

Loads on Members

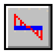
Loads on Floors

Pressure on Plates

Activation

Properties

Load Combination Activation Mode



The "Load Combination" icon of Activation toolbar

The **Edit/Activate Mode/Load Combination** function is activated by default as soon as an analysis has been carried out. However, no load combination title has been selected and you must select one from the pull-down menu of the Activation toolbar in order to view the results.

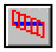
When the "Load Combination" active mode is selected, you may observe the results using one of the following procedures:

Through	Proceed as follows:
Dialog boxes	Click twice on an element, or first select an element and press down the "Properties" icon of the Edit toolbar. Load combination results for this element will be displayed.
Coloured Display	Select the **Results** tab of the **View Option** dialog box (**View** menu) and tick off the boxes corresponding to the results that you wish to have displayed. To modify graphic attributes (colour, line thickness) of different types of results, select the **Colours** tab.
Spreadsheets	Select **Load Combinations** under **Results** menu. Select the heading corresponding to the desired type of results (nodes displacements, members internal forces, plate internal forces, support reactions, etc.). Only the results for the elements selected on screen will be displayed in the spreadsheet. If no selection is made, all the element results will be displayed.

See also

Consulting Load Combination Results
Displacements at Nodes
Reactions at Supports
Internal Forces in Members
Internal Forces in Members (min/max)
Internal Stresses in Members
Internal Stresses in Members (min/max)
Internal Forces in Rectangular Plates
Internal Forces in Triangular Plates
Activate Mode
Properties

Envelope Activation Mode



The "Envelope" icon of Activation toolbar

Press the Envelope icon of Activation toolbar or select **Edit/Activate Mode**/**Envelope** function. Then, select an envelope title from the pull-down list of Activation toolbar.

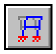
When the Envelope mode is selected, you may observe results using one of the following procedures:

Through	Proceed as follows:
Dialog boxes	Click twice on a member, or first select a member and press down the "Properties" icon of the Edit menu. The dialog box of the envelope results for this element will then be displayed.
Coloured Display	Select the **Results** tab of the **View Option** dialog box (**View** menu) and check-off the boxes corresponding to the results you wish to have displayed. To modify graphic attributes (colour, line thickness) of different types of results, select the **Colours** tab.
Spreadsheets	Select the **Results/Envelopes** menu, then one of the required results spreadsheet. Only the results for the elements selected on screen will be displayed in the spreadsheet. If no selection is made, results for all elements will be displayed.

See also

Consulting Envelope Results
Reactions at Supports (min/max)
Diagram of Internal Stresses and Deflections
Diagram of Internal Stresses and Deflections (min/max)
Variation of Stresses in Members
Internal Forces in Rectangular Plates
Internal Forces in Triangular Plates
Activate Mode
Properties

Vibration mode Activation Mode



The "Vibration mode" icon of Activation toolbar

The "Vibration mode" activation mode is useful to look at the structure deflection under a vibration mode selected in the Title Selection box. The structure deflection will automatically be displayed on your screen. The displayed deflections represent the shape of a vibration mode, not the real displacements of the model under loads.

The displacements of nodes under a selected vibration mode are available in **Results / Modal/Spectral**.

Have a look at the whole structure behaviour under a selected vibration mode by clicking on the "Animation" icon.

See also

Modal analysis

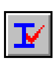
Nodes Displacements Spreadsheet

Animation

View Options

Title Selection

Design Results Activation Mode



The "Design Results" icon of Activation toolbar

The "Design Results" activation mode allows accessing steel and timber design results, design loads for concrete continuous systems and foundations design loads.

Steel, timber, and concrete design:

Go to **Results / Structure Design / Steel**, **Timber**, or **Concrete**.

Foundation Design Loads:

Go to the **Results** tab of **View Options** and activate the structural or geotechnical design loads.

See also

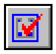
Steel Design Results Spreadsheet

Timber Design Results Spreadsheet

Concrete Design Results Spreadsheet

Display Foundation Design Loads and Legend

Rebar Placement Activation Mode



The "Rebar Placement" icon of Activation toolbar

To open the *Rebar Placement* window:

- Activate the Rebar placement mode and do one of the following:
    - Double-click on any continuous system
    - Click once and press the **Properties** icon.

See also

Rebar Placement Window
Reinforced Concrete Design
Prestressed Concrete Analysis
Design of 2-way-Slabs

Print Functions

Print

The "Print" icon of Standard toolbar

You can print a project under its graphic form or choose to print spreadsheets.

VisualDesign™ runs with printers and plotters supported by Windows NT and Windows 95+.

See also

Print Graphic (File Menu)
Print Spreadsheets (File Menu)
Print the Project Configuration
Export a screen image as a bitmap

Printer Configuration

Choose printer, format, and paper orientation by selecting **Printer Configuration** from the **File** menu. Note that this dialog box will automatically be called to the screen when you exit the Print dialog box.

Configuring your Printer

- Choose **Printer Configuration** from **File** menu.
- When the dialog box appears, select the paper feed source. Note that you may use a tracer to print the graphic presentation of a project, and a printer to print out the spreadsheets.
- Press "Configuration" and indicate how many copies you want and define the print options in accordance with the printer used.
- Select paper size from available options.
- Specify the paper source.
- Choose paper orientation. Pages can be oriented either vertically (portrait) or horizontally (landscape).
- To connect to another network user, specify the drive letter, and then press *Network*.
- To connect to another printer in a network, press the "Network" button.

- For more information about the Printer Configuration dialog box, choose Help.
- Press "OK" to validate your choices, or "Cancel" to annul the operation.

Print Preview

The "Print Preview" icon of Standard toolbar

The "Print Preview" icon of the Standard toolbar enables you to view the graphic display of your project such as it will appear when printed. Note that this function is not used to view a spreadsheet.

See also

Toolbar of "Print Preview" Window

Print

Previewing a Graphic before Printing

Previewing the Graphic before Printing

- Do one of the following:
  - Click the icon  on Standard toolbar.
  - Choose **Print Preview** from the **File** menu.
- In the **Trace Graphic** dialog box, indicate the tracing scale: either full-page or according to a scale factor.
- Select a WYSIWYG print (What You See Is What You Get) or print the whole structure. Indicate how much space the graphic shall cover on the page by entering the percentage in the Space-on-Page box.
- Select the width of the margins (mm or inches) by activating the appropriate option button.
- Enter the top, bottom, left, and right margins in the appropriate command boxes.
- Press OK to validate your choices and to preview the structure with the required specifications before printing.

Print Graphic from File Menu

The **Print/Graphics** command from **File** menu allows selecting a graphic printing of either the whole structure or what is displayed on screen (WYSIWYG).

Printing your Document in its Graphic Form

- Click the icon  of Standard toolbar or go to **File / Print / Graphics.**

The first dialog box displayed is **Plotting the Graphic** box.

- Adjust the plotting scale to the whole page or enter a scale factor to plot the structure.
- Set the printing zone. Activate the "Entire Structure" radio button to print the whole structure. Enter the percentage of the structure that should be displayed on the page. To only print what is displayed on screen, activate the "Screen (WYSIWYG)" radio button.
- Select margin dimensions (top, bottom, left and right) in the appropriate boxes. Specify units.
- Press the "OK" button to validate your input data and have access to the **Print** dialog box.

Toolbar from the "Print Preview" Window

Action	**Do the following**
View the next page	Press the Next button
View the previous page	Press the Previous button
View 2 pages at a time	Press the Two-Page button
Zoom in 150%	Press Zoom +
Zoom in 200%	Press Zoom + again
Return to the general work screen	Press Zoom -
Print	Press the Print button
Exit the Print Preview mode and return to the previous view mode	Press Close

Print Spreadsheets (File Menu)

All VisualDesign spreadsheets can be printed using the **Print Spreadsheets** function from the **File/Print** menu except for users working with a Trial Version of VisualDesign™.

The advantage of using this command, instead of the Print function included in spreadsheets' contextual menu, is that you can simultaneously print many spreadsheets for a continuous and optimized printing.

Create a PDF file with Adobe Acrobat. All spreadsheets will automatically be printed in a single file.

The list of available spreadsheets is shown in the form of an expanding tree. The roots are Project Configuration, Common, Structure, Loads and Results. Expand the roots by clicking the plus sign (+) box until reaching the category button.

The category button is green when all the spreadsheets are selected. It is grey when a partial selection has been made. It remains white when no spreadsheet has been selected.

To select a spreadsheet, click the category button next to it. To quickly select all spreadsheets included in a category, simply push down the root category button.

It is possible to print a spreadsheet with no grid, title zone, nor number zone.

Optimization of the Number of Printed Pages

To decrease the number of printed pages, two options are available: The option "Optimize with Portrait or Landscape" evaluates the number of pages according to a portrait or landscape printing, for each spreadsheet, and the second option is "Do not print ID columns".

N.B. If you have checked-off the "Print Empty Spreadsheets" option and asked to print spreadsheets of results before having analyzed, you will get empty spreadsheets. However, if you have not checked-off this option, you will get no output from the printer.

See also

Specification Sheet (Spreadsheet)
Printing Spreadsheets in Series
Managing Spreadsheets

Printing Spreadsheets in Series

- Select the "Spreadsheets" heading of **File/Print**. (The "Print" button of the Standard toolbar has not been designed to print spreadsheet).
- In the **Printing Spreadsheet** dialog box, select the specific radio buttons for the spreadsheets to be printed. To view the list of each category spreadsheet (**Project Configuration**, **Structure**, **Loads** and **Results**, click on their respective expansion button [+]. To select all spreadsheets of the same category, activate the radio button of this family.

Note: Selection of items listed in an expanding tree: When a family radio button is green, all spreadsheets are selected. When a family radio button is white, no spreadsheet has been selected. If the radio button is grey, it means that a partial selection has been made in this family.

- To reduce the number of printed pages, use option "Optimize the printing with portrait or landscape". Check also the box "Don't print ID columns".
- To print empty spreadsheets, tick the "Printing Empty Spreadsheets" box.

Warning! If you have not clicked this box and are asking for the printing of a results spreadsheet before analyzing, no spreadsheet will be printed.

- Press down the "Page Set-up" button to reach the dialog box that includes the paper settings and the margins.
- Press down the "Printer" button to set the size and orientation of paper sheet and select the printer.
- Press the "OK" button to print your document or the "Cancel" button to stop the operation.

Print the Project Configuration

- Go to **File / Print / Spreadsheets** and expand the *Project Configuration* root in the selection tree that is displayed in the **Printing Spreadsheet** dialog box.

All branches correspond to the **Project Configuration** dialog box (**File** menu), except *Statistics*, *Moving Loads* and *Bridge Evaluation*.

The *Statistics* branch includes the number of elements in your model, the number of load cases, load combinations, and envelopes.

The *Moving Loads* branch includes information you may find in the **Moving Load Analysis** dialog box.

The *Bridge Evaluation* branch includes user's input for bridge evaluation.

Printing the Project Configuration

- Select **File / Print /Spreadsheets**. (The "Print" button of Standard toolbar has not been designed to print spreadsheets).
- In the **Printing Spreadsheet** dialog box, expand the "Project Configuration" root and check appropriate boxes.
- Press the "Printer" button to choose a printer.
- Press the "OK" button to start the printing.
- Press the "Cancel" button to exit the dialog box or to cancel the operation.

Windows Management

New Window

Choose **New Window** from the **Window** menu to open one or more versions of your main project window.

The secondary window lets you display different parts of the structure. When you modify the structure in one window, the changes you make will be carried over to all the other windows. (Only the zoom functions will not be carried over.)

A secondary window displays the name of the original window, followed by a colon and the new window's number. For instance, if the original window is called BridgeA.vd1:1, the next window will be called BridgeA.vd1:2.

Creating a Secondary Window from Active File

- Choose **New Window** from **Window** menu.

Cascade

The **Cascade** option lets you to view the opened windows in a cascade. Only the title bars will be visible.

Tile Horizontally

Choose **Tile Horizontally** from the **Window** menu to view horizontally opened windows simultaneously. The windows will all be visible on the screen, not overlaid. Open windows are organized from the top towards the bottom of the screen.

Tile Vertically

Choose **Tile vertically** to view, up to four vertically opened windows simultaneously. The windows will all be visible on the screen, not overlaid. Open windows are organized from left towards right of the screen.

Rearrange

You may rearrange windows converted into icons.

To shrink a window and convert it to an icon, press the Minimize arrow on the title bar.

To rearrange windows converted to icons, choose **Rearranging Icons** from the **Window** menu. The icons will be rearranged horizontally, from the left to the right, at the bottom of the screen.

Refresh

When you move elements on the screen, some pixels may remain residual. In order to remove these parasite signals from the screen, activate the **Refresh** function or press the **F5** key.

List of Open Windows

VisualDesign™ displays a list of opened windows pertaining to the same document. To switch from one window to another, select another window from the list in the **Window** menu.

Chapter

2

MATERIALS, SECTIONS & ELEMENTS

TABLE OF CONTENTS

Chapter 2 Materials, Sections & Elements

General

About Common Objects

Chapter 2 describes some spreadsheets that are composing the **Common** menu such as Materials, Shapes, Reinforcement, Cables, Studs, Steel Decks, and Bolts. These are called *common objects* because they are part of VisualDesign database *VDBase.mdb* and they can be shared among users (network version).

Editing spreadsheets located in the Common menu

These basic data cannot be edited, except the rebar bending shape's number and aliases. However, all these spreadsheets can contain personalized data when a line is added at the end of the spreadsheet.

To create a new object (material, shape, reinforcement, cable, stud, steel deck, bolt, soil or mobile), add a line at the end of the spreadsheet. You will notice that the ID number is 5000 or more, meaning that it is a personalized object. Change the name and enter new parameters. (Example: copy & paste a type of stud, change the name and its length).

The archiving function is activated by default (**Preferences** tab of **Project Configuration**). Therefore, all personalized objects will be saved within your project file (.vd1 or .vdz).

See also

Common menu

Archiving common objects within your file

Preferences tab (Project Configuration)

VDBase.mdb file

Importing Common Objects to your Database

When you open a .vd1 or .vdz file that comes from another computer (from outside your network), it may contain personalized common objects that are not part of your VDBase.mdb file. VisualDesign will read this file and detect these elements.

To inform users and to facilitate the understanding if what is going on, a spreadsheet will appear on your screen at the end of the reading of the file. VisualDesign will let you choose the action that you want to do for each detected object: either import the new object to your database or give a new name to the identical object that it found at the reading, or cancel the importation.

If you accept the importation of an object that has an identical name in your database, you can enter a new name for the imported element and VisualDesign will modify the input and use this name if there is no conflict.

This table describes the columns that are part of the spreadsheet:

Column	Description	Editing
Type	Type of personalized common object that has been detected at the reading of the file and which is not part of the user database.	No
Number	Number or name of the detected common object.	No
Equivalent Object	If VisualDesign detected an object in your database that possesses identical parameters, the name of this equivalent object will be written in this column.	No
Action	VisualDesign can choose two actions when the reading is completed: *Import* or *Use equivalent*. If no equivalent object has been found, the *Import* option will be automatically selected. If an equivalent object has been found, the option *Use equivalent* will be selected.	Yes/No
New Number	Enter a new number/name to describe the imported object if an identical name has been detected in your database. If you do not enter a name, VisualDesign will rename this object by adding the term "%a" at the end of existing name and will use it for this project file.	Single click

See also

The *Commun.mdb* database

Archiving common objects within your .vd1 or .vdz file

The *VDBase.mdb* database

Saving styles and colours

Materials

Material Properties

Steel Materials

Properties of steel materials have been extracted from the following standards:

CAN/CSA-G40.21-M92: Handbook of Steel Construction (1995), Canadian Institute of Steel Construction, Sixth Edition, Table 6-3.

ASTM: Manual of Steel Construction (1995), American Institute of Steel Construction, Second edition, Table 1-1.

S136 Standard for Cold-Formed Sections.

ASTM A500 Grade C for American HSS.

Concrete Material

Properties of concrete materials have been extracted from the following standard:

CAN/CSA-A23.1-94: Concrete Materials and Methods of Concrete Construction;

Timber

Wood Design Manual (2001), Canadian Wood Council.

See also

Steel Materials spreadsheet
Concrete Materials spreadsheet
Timber Properties spreadsheet
Aluminium Materials spreadsheet

Steel Materials Spreadsheet

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumerical characters describing the type of steel.	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Category	Select a category among the drop down list box: G40, ASTM, Strand, or Others.	Double click
E	Young's elastic modulus	Single click
G	Shear modulus	Single click
μ	Poisson's ratio	Single click
Density	Density of steel	Single click
Thermal Coeff.	Coefficient of thermal expansion	Single click
fy	Yield strength of steel	Single click
Fu	Tensile strength of steel	Single click
Ry	Factor applied to fy for estimating the probable elastic limit (refer to section 27 of CAN/CSA-S16-01 Standard.)	Single click
Rt (Not yet activated)	Ratio of probable tensile strength and minimum tensile strength, Fu, relatively to the probable elastic limit, Ry. (Refer to AISC-I-6- Seismic Provisions for Structural Buildings, March 2005)	Single click

See also

Steel Specifications

Concrete Materials Spreadsheet

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
E	Modulus of elasticity of concrete	Single click
G	Shear modulus	Single click
μ	Poisson's ratio	Single click
Density	Material density	Single click
Density (E, f'c)	Density of material that is considered for calculating f'c.	Single click
Thermal Coeff.	Coefficient of thermal expansion	Single click
f'c	Specified compressive strength of concrete	Single click
fct	Tensile strength of concrete	Single click
λ	Modification factor taking into account the effects of concrete density on its tensile strength λ=1,00 for normal density λ=0,85 for structural semi-low-density concrete in which all of the fine aggregate is natural sand. λ=0,75 for structural low-density concrete in which none of the fine aggregate is natural sand.	Single click
a max	Maximum diameter of aggregates present in the mixture	Single click
εf	Stress of concrete due to shrinkage	Single click
Ect	Effective modulus of concrete in tension	Single click
Elt	Concrete modulus of elasticity (long term)	Single click

Column	Description	Editing
α	Factor depending on type of cement and curing conditions and is used in the calculation of f'ci for prestressed concrete beams. See topic *Calculation of factor alpha…*	Single click
Type of cement	Specify the hardening for this type of concrete: *Slow*, *Normal*, *Quick* or *Quick and High Resistance* hardening.	Double click

Note. For the λ modification factor: Linear interpolation may be applied based on the fraction of natural sand in the mix.

See also

Concrete Specifications

Factor alpha for Prestressed Concrete

Timber Materials Spreadsheet

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Number	Name (12 alphanumeric characters) describing the wood species and structural quality. Refer to Timber Nomenclature.	Double-click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Classification	Choose the classification of this material among the drop-down list box: Visual, MSR (Machine Stress-Rated), MEL (Machine Evaluated Lumber) or GLT (Glue Laminated Timber). Refer to Classification	Double-click
Grade	Select the grade among the list box: Select structural, No.1, No.2, No.3, Construction or Standard.	Double-click
Species	Select a species among the list box: S-P-F, D Fir-L, Hem-Fir or North Species.	Double-click

Column	Description	Editing
Available Cuts	This column shows the available cuts according to the classification, grade and species of this element. **N.B. While designing timber elements, VisualDesign cannot change the classification of this material.**	Double-click
E	Specified Elastic modulus.	Single click
E05	Modulus of elastic for design of compression members.	Single click
G	Shear modulus	Single click
μ	Poisson's ratio	Single click
Density	Density of this material.	Single click
Thermal coeff.	Coefficient of thermal expansion	Single click
fb|M+	Specified positive bending strength at extreme fibre.	Single click
fb|M-	Specified negative bending strength at extreme fibre.	Single click
fv	Specified shear strength.	Single click
fc	Specified compressive strength parallel to grain.	Single click
fcp|c	Specified compressive strength perpendicular to grain, compression face bearing.	Single click
fcp|t	Specified compressive strength perpendicular to grain, tension face bearing.	Single click
ftn	Specified tensile strength parallel to grain at net section.	Single click
ftg	Specified tensile strength parallel to grain at gross section of glued-laminated timber.	Single click
ftp	Specified tensile strength perpendicular to grain.	Single click

See also

Timber Specification
Timber Species and Properties

Aluminium Materials Spreadsheet

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the type of aluminium.	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
E	Young's elastic modulus	Single click
G	Shear modulus	Single click
μ	Poisson’s ratio	Single click
Density	Density of aluminium	Single click
Thermal Coeff.	Coefficient of thermal expansion	Single click
Fy	Yield strength of aluminium	Single click
Fu	Tensile strength of aluminium	Single click

See also

Aluminium Specifications

Shapes & Sections

About Shapes Spreadsheets

Metric and Imperial Designation

The Metric or Imperial designation for shapes is activated in the **Preferences** tab of **Project Configuration** dialog box (**File** menu).

Personalized Standard Shapes:

Insert a line in any standard shape spreadsheet to create your own standard shape. Enter general parameters such as b, d, t, and w. VisualDesign automatically calculates the new shape properties.

Use the shape nomenclature (name) that you may find in the shape selection tree to find the new standard shape in a particular branch. Ex.: W460x67, HS305x305x11.

Shape Availability

Symbol [cue] appears in the tree sections before the name of the shape. c = Canada, u = USA, e = Europe. When a shape is not available in a country, the symbol "_" is displayed. Suppose a shape is only available in the US, the following symbol is then displayed [_u_].

Distribution

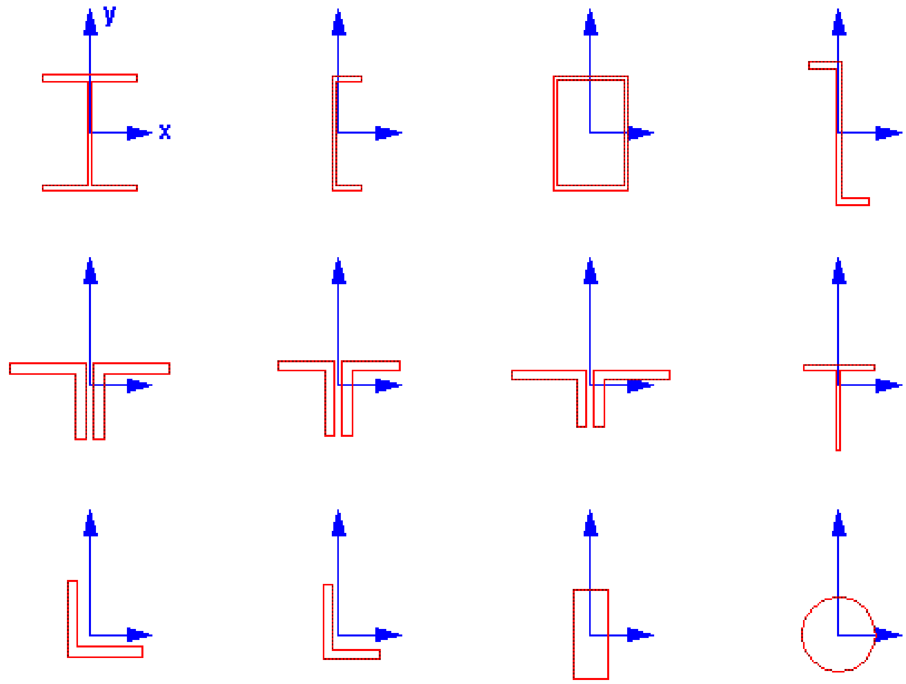
Assign a "Private" distribution to your personalized shape if you don't want it to be imported to another user's database at the opening of your file. N. B. A private distribution can only be assigned to personalized shapes. All VisualDesign shapes have a "Public" distribution.

The Shape Properties Dialog Box

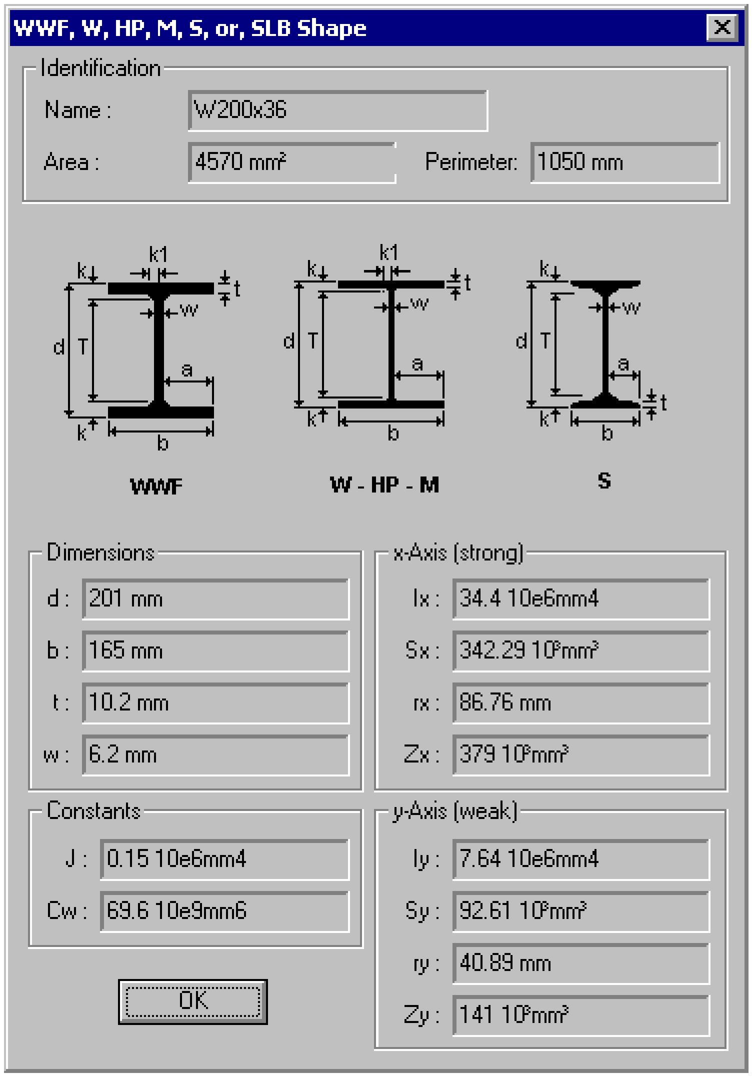
This dialog box is accessible from shapes spreadsheets: Click in any cell, right click to open the contextual menu and select the function **Detail**.

Section Strong and Weak Axis

In VisualDesign, the member strong and weak axes are as follows:



I Shapes Spreadsheet



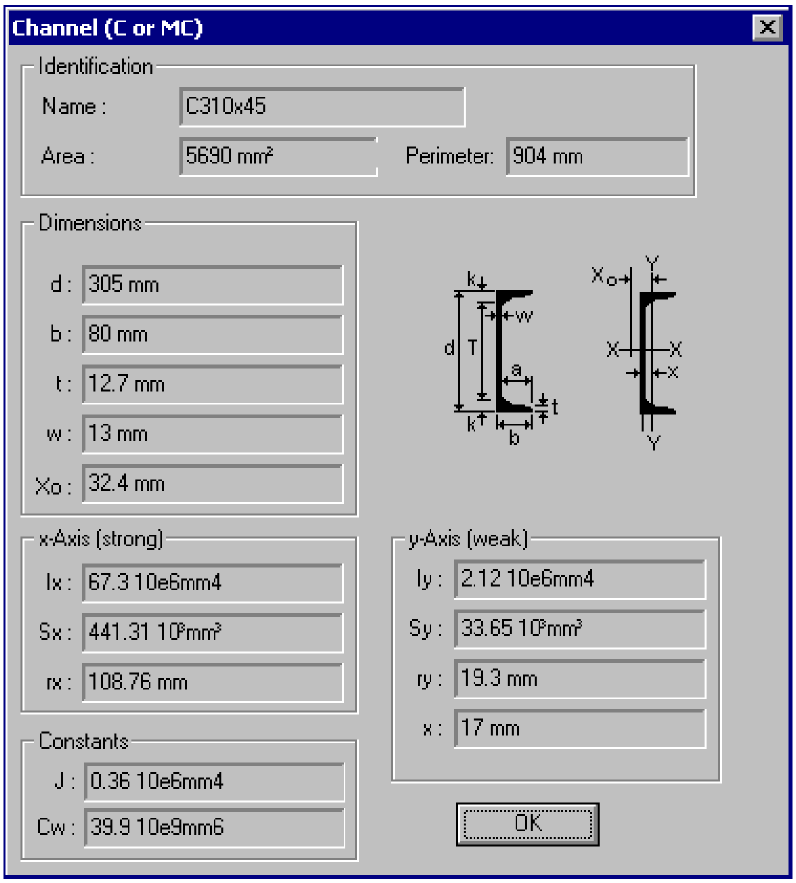
Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click

Column	Description	Editing
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Shape: HP, M, S, W, WWF, or SLB	Double-click
d	Depth	Single click
d nominal	Nominal depth	Single click
b	Flange width	Single click
w	Web thickness	Single click
t	Flange thickness	Single click
k	Distance from outer face of flange to web toe of fillet.	Single click
k1	Distance from centreline of section to flange toe of fillet.	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus on strong axis	No
rx	Radius of gyration on strong axis	No
Zx	Plastic section moment on strong axis	Single click
Iy	Moment of inertia on weak axis	Single click
Sy	Elastic section modulus on weak axis	No
ry	Radius of gyration on weak axis	No
Zy	Plastic section moment on weak axis	Single click
J	Torsional constant	Single click
Cw	Warping constant	Single click

Column	Description	Editing
Perimeter	Perimeter of the section.	Single click

C Shapes Spreadsheet

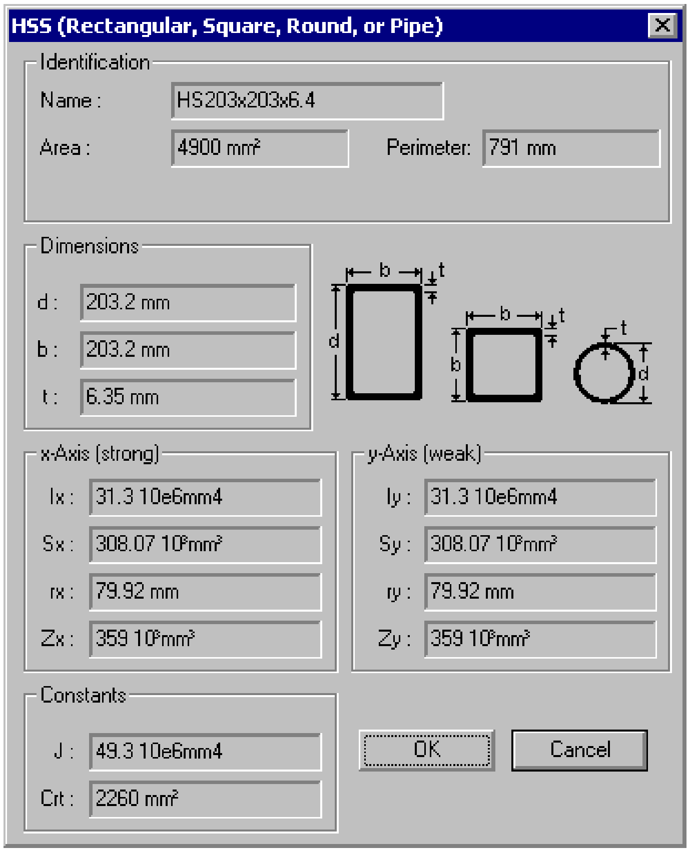


Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click

Column	Description	Editing
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Shape: C or MC	Double-click
d	Depth of the section	Single click
d nominal	Nominal depth of the section	Single click
b	Flange width	Single click
t	Flange mean thickness	Single click
w	Web thickness	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
Iy	Moment of inertia – weak axis	Single click
Sy	Elastic section modulus around weak axis	No
ry	Gyration radius – weak axis	No
x	Position of gravity axis	Single click
Xo	Shear centre	Single click
J	Torsional constant	Single click
Cw	Warping constant	Single click
Perimeter	Perimeter of the section.	Single click

HSS Shapes Spreadsheet



Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar

Column	Description	Editing
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Shape: Square, Rectangular, Circular or Pipe.	Double-click
d	Depth	Single click
d nominal	Nominal depth	Single click
b	Section width	Single click
t	Wall thickness	Single click
Ext. radius	Bend radius measured from outer face.	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
Zx	Plastic section moment around strong axis	Single click
Iy	Moment of inertia – weak axis	Single click
Sy	Elastic section modulus around weak axis	No
ry	Radius of gyration – weak axis	No
Zy	Plastic section moment around weak axis	Single click
J	Torsional constant	Single click
Crt	Shear constant	Single click
Perimeter	Perimeter of the section.	Single click

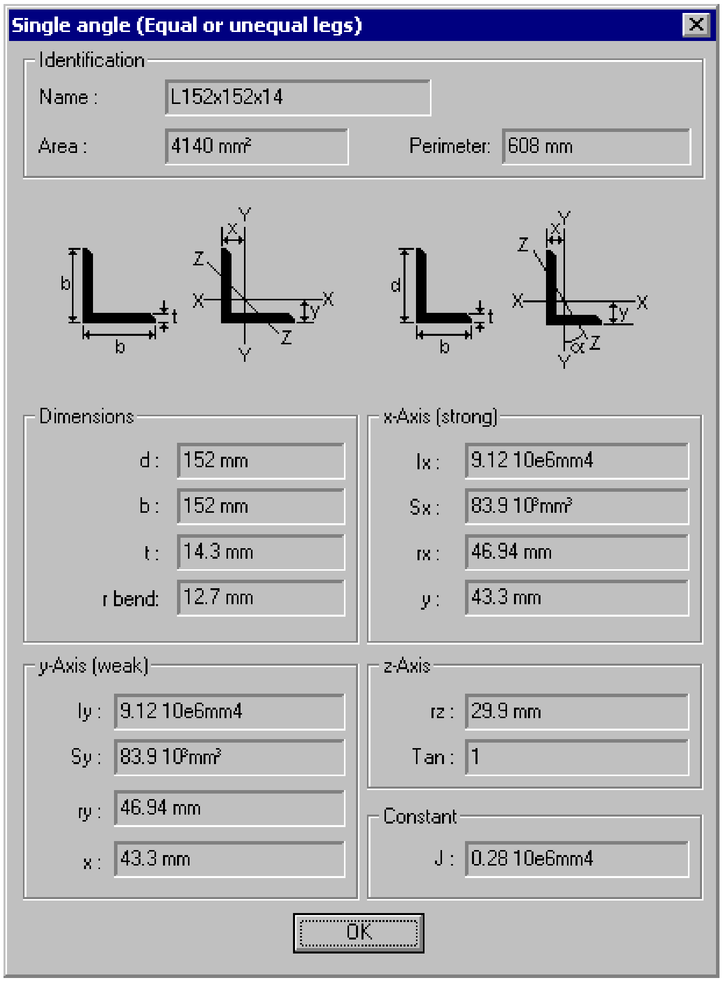
HSS Design wall thickness

The design wall thickness of a selected HSS must be specified in the **Member** tab (**Member Characteristics** dialog box). HSS properties are adjusted with respect to this wall thickness.

The display of such shapes on screen and its nomenclature in the design brief is as follows: HS305x254x9.5 (0.9t).

Note Dead load is calculated without considering the thickness reduction. Consequently, dead load applied on the structure and bill of materials are adjusted to usual HSS (full wall thickness).

L Shapes Spreadsheet



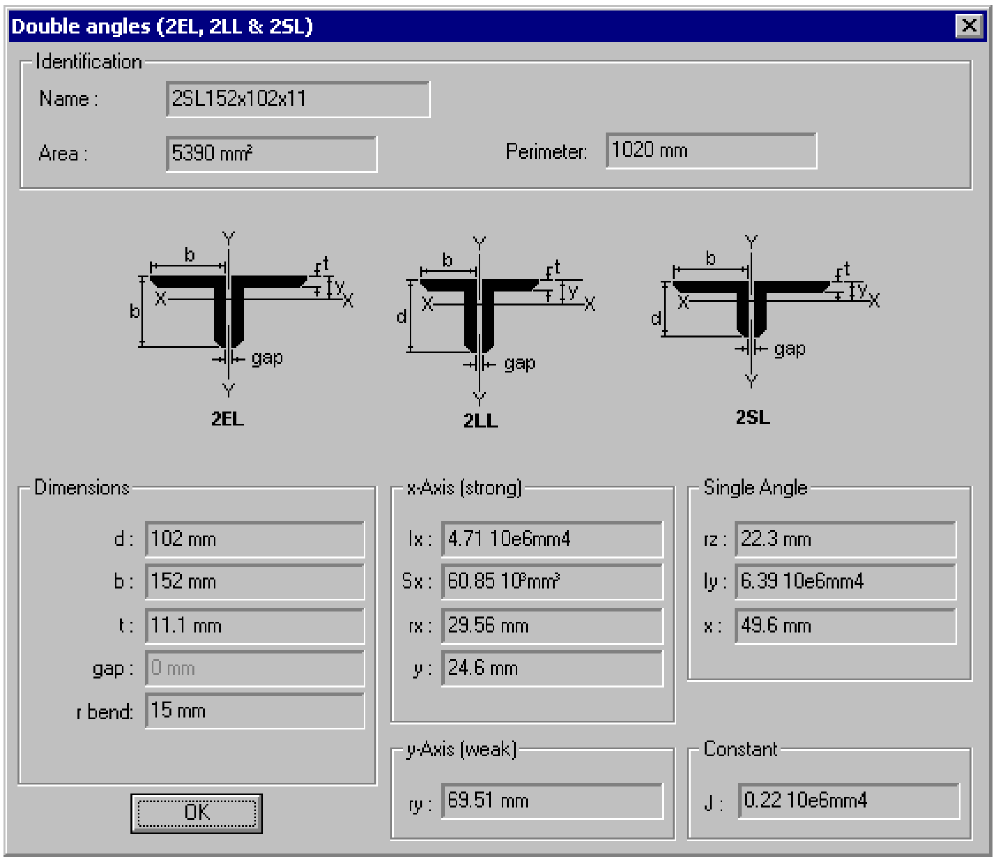
Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click

Column	Description	Editing
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Type of section: L (d=b) or L (d>b)	Double-click
d	Depth	Single click
b	Section width	Single click
t	Flange thickness	Single click
Bend radius	Bend radius used in the calculation of width/thickness ratio of compression elements (standard S37-01)	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
y	Position of gravity axis	Single click
Iy	Moment of inertia – weak axis	Single click
Sy	Elastic section modulus around weak axis	No
ry	Radius of gyration – weak axis	No
x	Position of gravity axis	Single click
rz	Lowest radius of gyration	Single click
Tan α	Angle between orthogonal and major/minor axis system and used to calculate rz.	Single click
J	Torsional constant	No

Column	Description	Editing
Perimeter	Perimeter of the section used to calculate the surface to paint and/or ice coating.	Single click

2L Shapes Spreadsheet

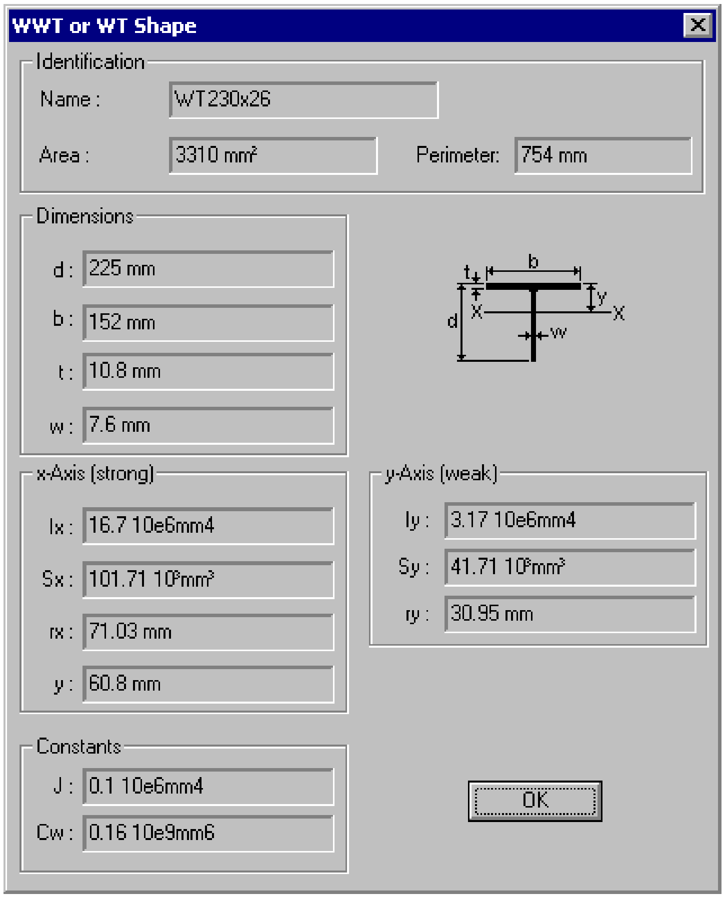


Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click

Column	Description	Editing
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Layout of double steel angles: 2EL, 2SL, 2LL	Double-click
d	Depth	Single click
b	Section width	Single click
t	Thickness	Single click
Bend radius	Bend radius used in the calculation of width/thickness ratio of compression elements (standard S37-01)	Single click
Area	Section area of 2 angles	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration - strong axis	No
y	Position of gravity axis	Single click
Iy angle	Moment of inertia – weak axis	Single click
rz angle	Radius of gyration – major/minor axis system for a single angle	Single click
x angle	Position of gravity axis	Single click
ry	Radius of gyration – weak axis	No
J	Torsional constant	No
Perimeter	Perimeter of the section used to calculate the surface to paint and/or ice coating.	Single click

T Shapes Spreadsheet



Group: Shared Data: VDBase.mdb

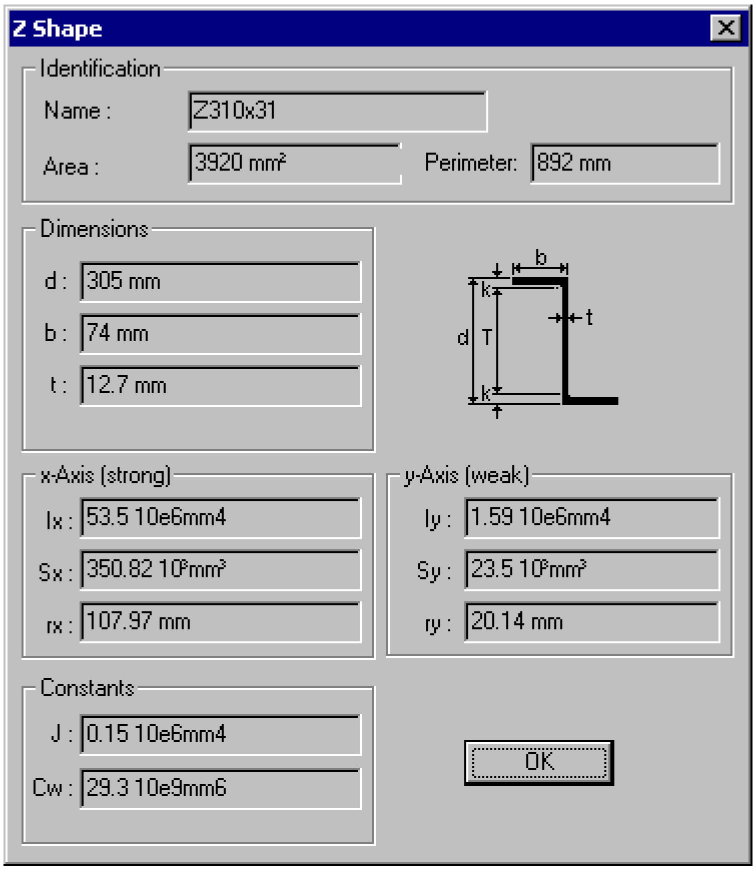
Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar

Column	Description	Editing
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Type	Type of steel shape: WT, WWT Concrete section: T	Double-click
d	Depth of section	Single click
d nominal	Nominal depth of section	Single click
b	Flange width	Single click
t	Flange thickness	Single click
w	Web thickness	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
y	Position of centre of gravity, parallel to local y-axis.	Single click
Iy	Moment of inertia – weak axis	Single click
Sy	Elastic section modulus around weak axis	No
ry	Radius of gyration – weak axis	No
J	Torsional constant	No
Cw	Warping torsional constant	No
Perimeter	Perimeter of the section.	Single click

Z Shapes Spreadsheet

VisualDesign does not design such shapes. Only compression strength is verified. You must calculate the bending capacity Mr yourself.

If you choose a major/minor axis system in the *Member* tab (**Member Characteristics** dialog box) VisualDesign, will not consider it. (A value of 1.0 kN will be displayed in the Results spreadsheet.)

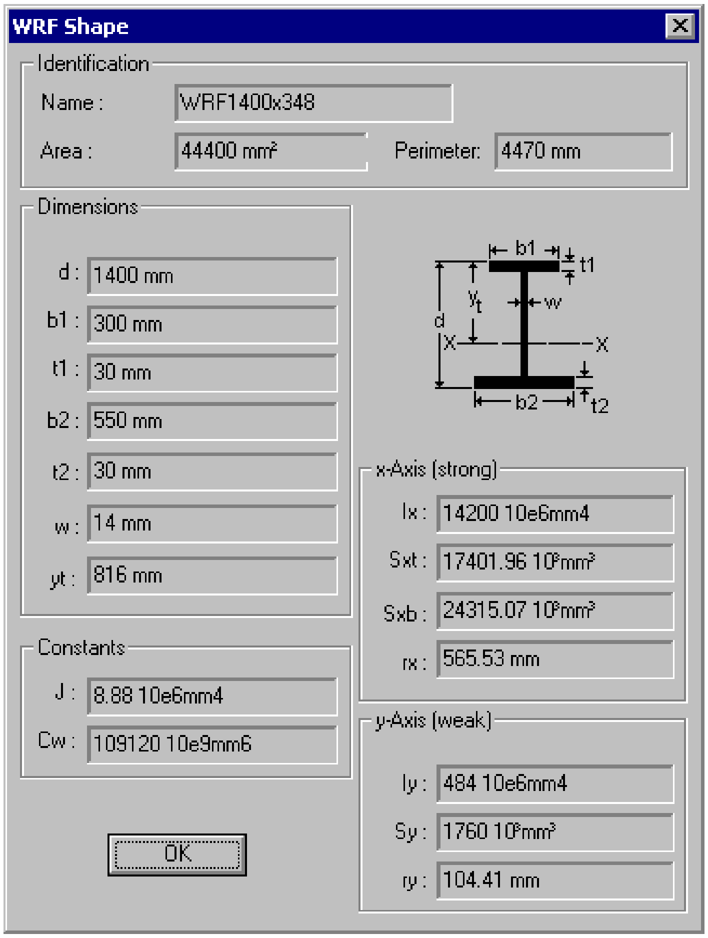


Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click

Column	Description	Editing
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Depth of section	Single click
d nominal	Nominal depth of section	Single click
b	Flange width	Single click
t	Web thickness	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
Iy	Moment of inertia – weak axis	Single click
Sy	Elastic section modulus around weak axis	No
ry	Gyration radius – weak axis	No
J	Torsional constant	Single click
Cw	Warping constant	Single click
Perimeter	Perimeter of the section.	Single click

WRF Shapes Spreadsheet

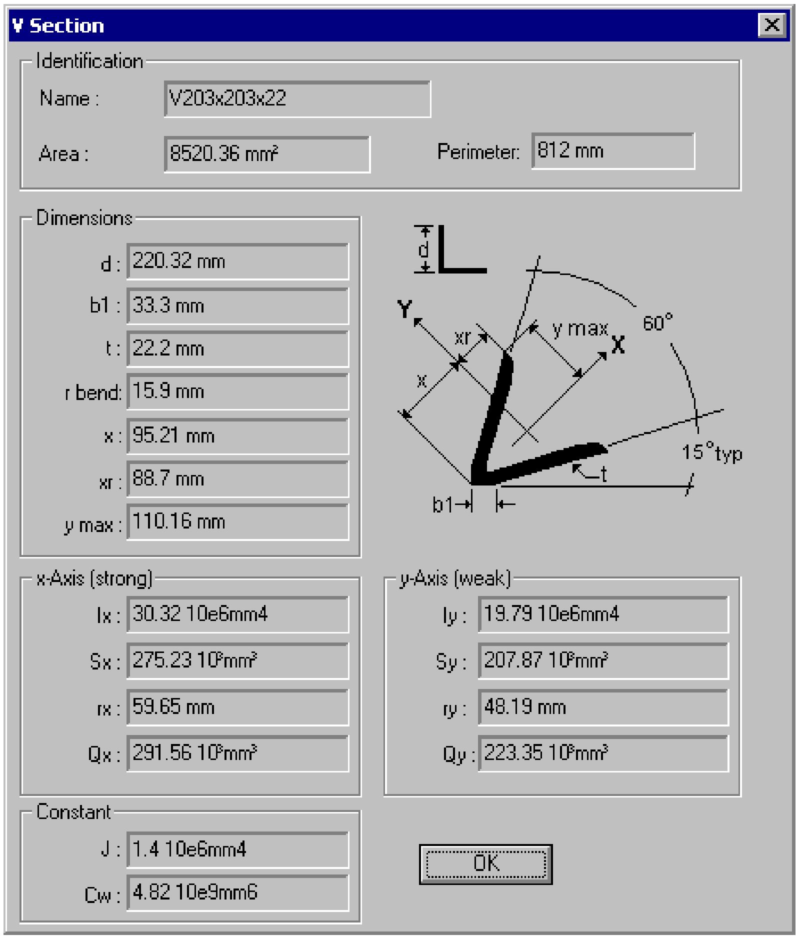


Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar

Column	Description	Editing
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Depth of section	Single click
d nominal	Nominal depth of section	Single click
b1	Width of top flange	Single click
t1	Thickness of top flange	Single click
b2	Width of bottom flange	Single click
t2	Thickness of bottom flange	Single click
w	Web thickness	Single click
yt	Distance from top to centre of gravity.	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
rx	Radius of gyration – strong axis	No
Sxt	Elastic modulus of section at the top, on strong axis.	No
Sxb	Elastic modulus of section at the bottom, on strong axis.	No
Iy	Moment of inertia – weak axis	Single click
ry	Radius of gyration – weak axis	No
Sy	Elastic modulus of section around weak axis	No
J	Torsional constant	Single click
Cw	Warping constant	Single click
Perimeter	Perimeter of the section.	Single click

V Shapes Spreadsheet



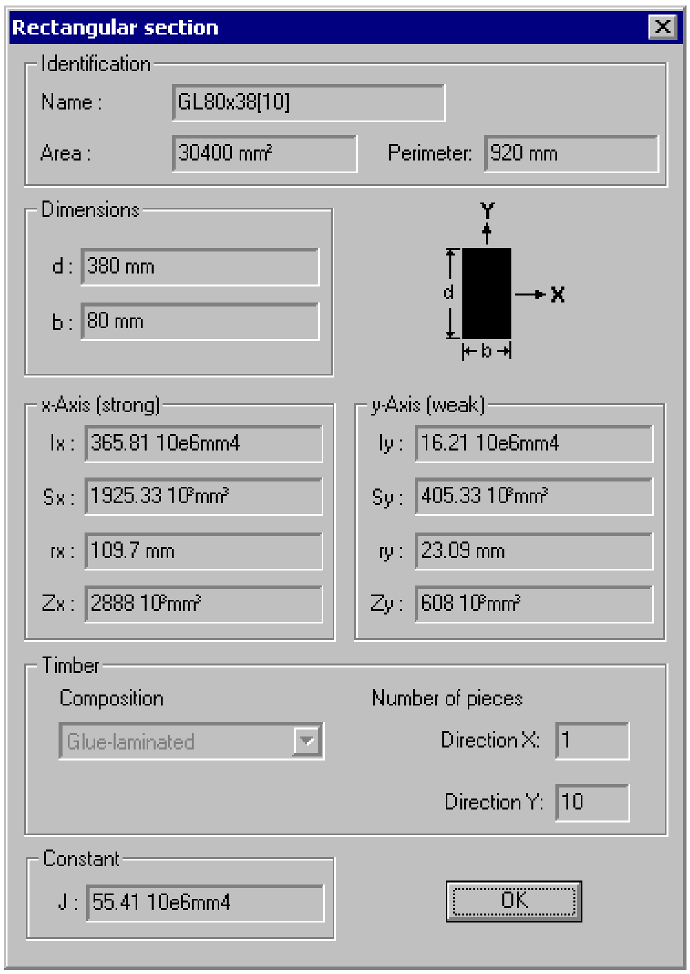
Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar

Column	Description	Editing
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Depth of the section	Single click
d nominal	Nominal depth of the section	Single click
b1	Width of the section	Single click
t	Thickness	Single click
x	Look at the image above.	Single click
xr	Look at the image above.	Single click
y max	Look at the image above.	Single click
Bend radius	Bend radius used in the calculation of width/thickness ratio of compression elements (standard S37-01)	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	Single click
rx	Radius of gyration – strong axis	No
Sx	Elastic modulus of the section around the strong axis	No
Qx	The shear stress by units of width, in the direction of the z axis, at the x face of the local system	No
Iy angle	Moment of inertia of original angle– weak axis	Single click
ry	Radius of gyration – weak axis	No
Sy	Elastic modulus of the section around the weak axis	No
Qy	The shear stress by units of width, in the direction of the z-axis, at the y face of the local system.	No
J	Torsional constant	Single click
Cw	Warping constant	No

Column	Description	Editing
Perimeter	Perimeter of section.	Single click

Rectangular Shapes Spreadsheet

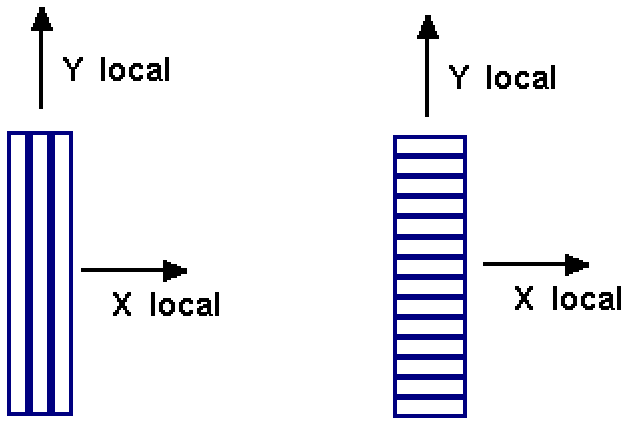


Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click

Column	Description	Editing
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Depth of the section	Single click
b	Width of the section	Single click
Area	Section area	No
Ix	Moment of inertia – strong axis	No
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
Zx	Plastic section moment around strong axis	No
Iy	Moment of inertia – weak axis	No
Sy	Elastic section modulus around weak axis	No
ry	Radius of gyration – weak axis	No
Zy	Plastic section moment around weak axis	No
J	Torsional constant	No
Composition	Timber design: Select the composition of the member: Sawn Timber, Glulam, or Composite	Double-click
Number of pieces in local x-direction	Timber design: If composition is glulam, indicate the number of laminations in the local x-direction.	Single click
Number of pieces in local y-direction	Timber design: If composition is glulam, indicate the number of laminations in the local y-direction.	Single click
Perimeter	Perimeter of the section.	No

Laminations in Glulam Sections



Round Shapes Spreadsheet

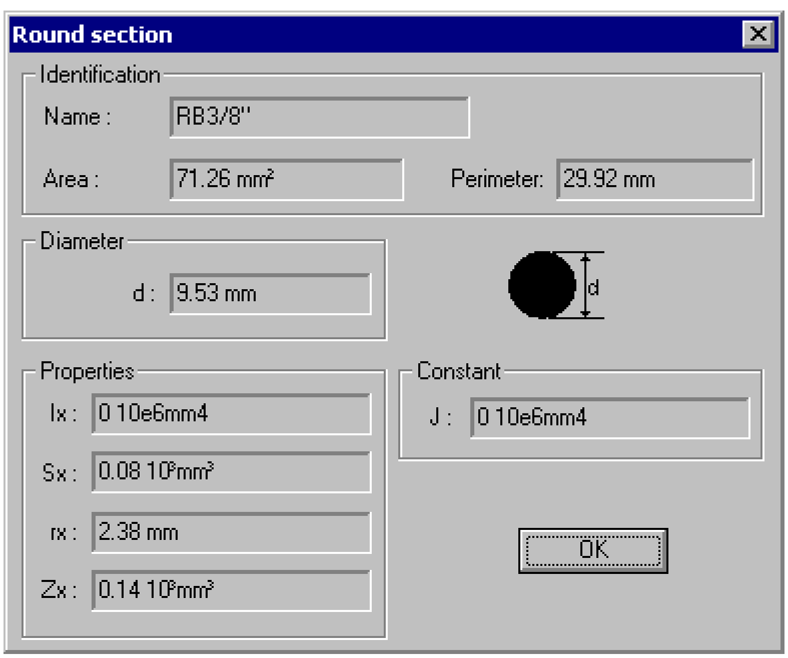
VisualDesign calculates the round section area in an automatic way using the diameter defined in this spreadsheet. To modify the area, double-click in the "Area" cell.

Shape Designation:

BS: Bridge Strand

GS: Guy Strand (for guyed tower)

RB: Round Bar



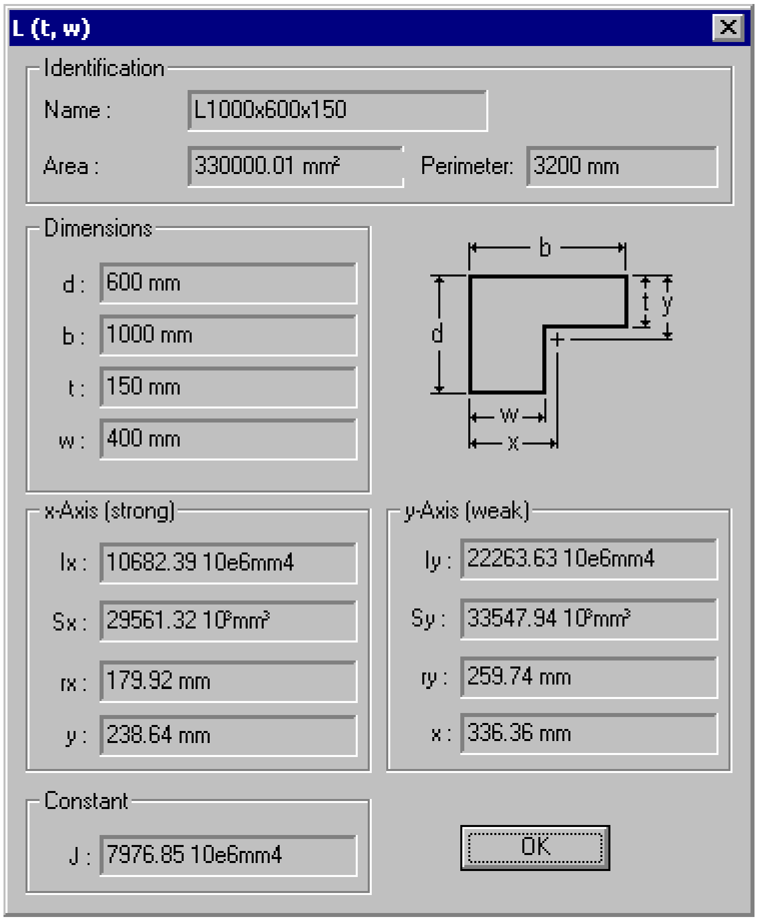
Stress Relaxation:

If you own the *Steel Design* module, you can consider stress relaxation by activating the appropriate box in the **Steel Design** tab of **Member Characteristics** dialog box.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Depth	Single click
Area	Section area	Single click
Ix	Moment of inertia – strong axis	No
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
Zx	Plastic section moment around strong axis	No
J	Torsional constant	No
Perimeter	The perimeter is used to calculate the surface to paint and/or ice coating.	No

L (t, w) Sections Spreadsheet



Group: Shared Data: VDBase.mdb

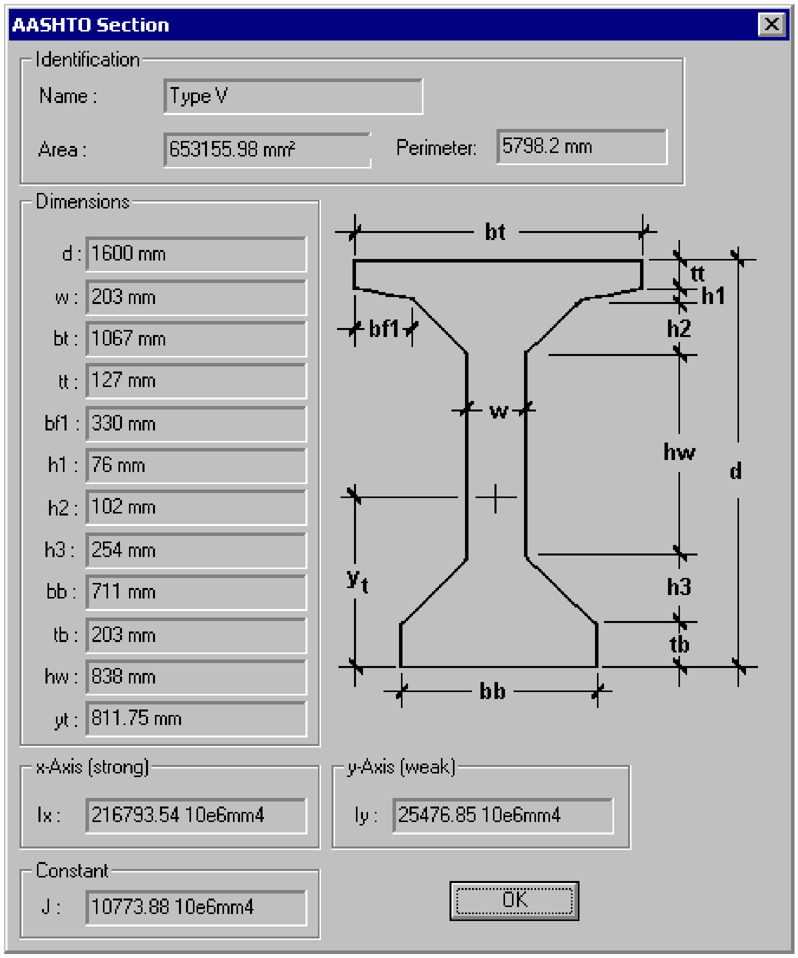
Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar

Column	Description	Editing
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Height of section	Single click
d nominal	Nominal height of section	Single click
b	Width of leg	Single click
t	Leg thickness	Single click
w	Web thickness	Single click
Area	Section area	No
Ix	Moment of inertia – strong axis	No
Sx	Elastic section modulus around strong axis	No
rx	Radius of gyration – strong axis	No
y	Distance from top fibre to section centre of gravity.	No
Iy	Moment of inertia – weak axis	No
Sy	Elastic section modulus around weak axis	No
ry	Gyration radius – weak axis	No
x	Distance from left fibre to section centre of gravity	No
J	Torsional constant	No
Perimeter	Perimeter of the section.	No

AASHTO Sections spreadsheet

This spreadsheet is located in the **Common/Shapes** menu. It is mostly used for prestressed concrete structure (composite or not) with pre-tensioning cables.

T-sections can be modeled from AASHTO sections, using appropriate dimensions. To learn more, refer to the topic: Prestressed Concrete T-Section



Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Total height of the section	Single click

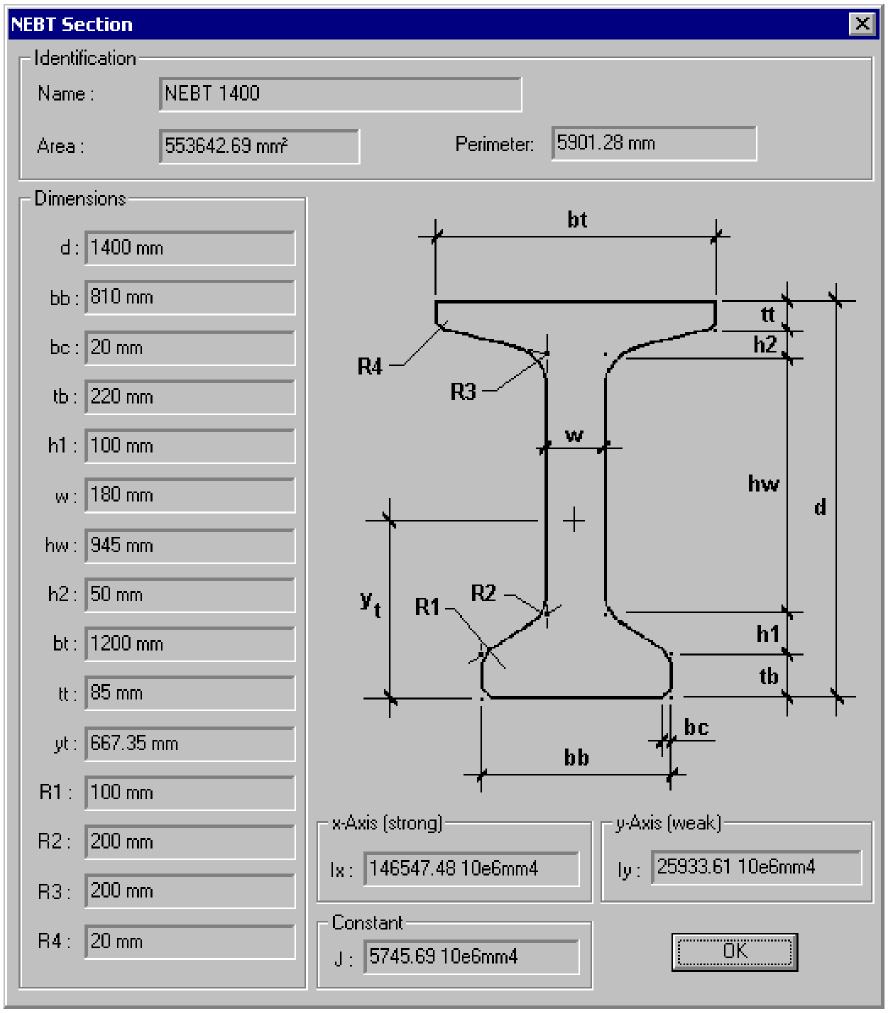
Column	Description	Editing
w	Thickness of the web	Double-click
bt	Total width of top flange	Single click
tt	Thickness of top flange	Single click
bf1	See the figure above.	Single click
h1	See the figure above.	Single click
h2	See the figure above.	Single click
h3	See the figure above.	Single click
bb	Total width of bottom flange	Single click
tb	Thickness of bottom flange	Single click
hw	Height of the web having a thickness w.	No
Area	Area of the section	No
yt	Distance from bottom to centre of gravity	No
Ix	Section inertia – strong axis	No
Iy	Section inertia – weak axis	No
J	Torsional constant	No
Perimeter	Perimeter of the section.	No

See also

Prestressed Concrete Composite Beams

NEBT Sections spreadsheet

This spreadsheet is located in the **Common/Shapes** menu. It is mostly used for prestressed concrete structure (composite or not) with pre-tensioning cables.



Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click

Column	Description	Editing
Material	Choose the shape material among the list box.	Double-click
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Total height of section	Single click
bb	Total width of bottom flange	Single click
bc	Width of truncated part (45 deg.) of bottom flange.	Single click
tb	See the figure above.	Single click
R1	Radius of curvature of bottom flange. (See the figure above)	Single click
R2	Radius of curvature of bottom flange. (See the figure above)	Single click
h1	See the figure above.	Single click
w	Web thickness of the section	Single click
hw	Height of web having a thickness w.	Single click
R3	Radius of curvature of top flange. (See the figure above)	Single click
R4	Radius of curvature of top flange. (See the figure above)	Single click
h2	See the figure above.	Single click
bt	Total width of top flange.	Single click
tt	See the figure above.	Single click
Area	Area of the section	No
yt	Distance from bottom fibre to centre of gravity	No
Ix	Section inertia – strong axis	No
Iy	Section inertia – weak axis	No
J	Torsional constant	No

Column	Description	Editing
Perimeter	Perimeter of the section.	No

See also

About Shapes

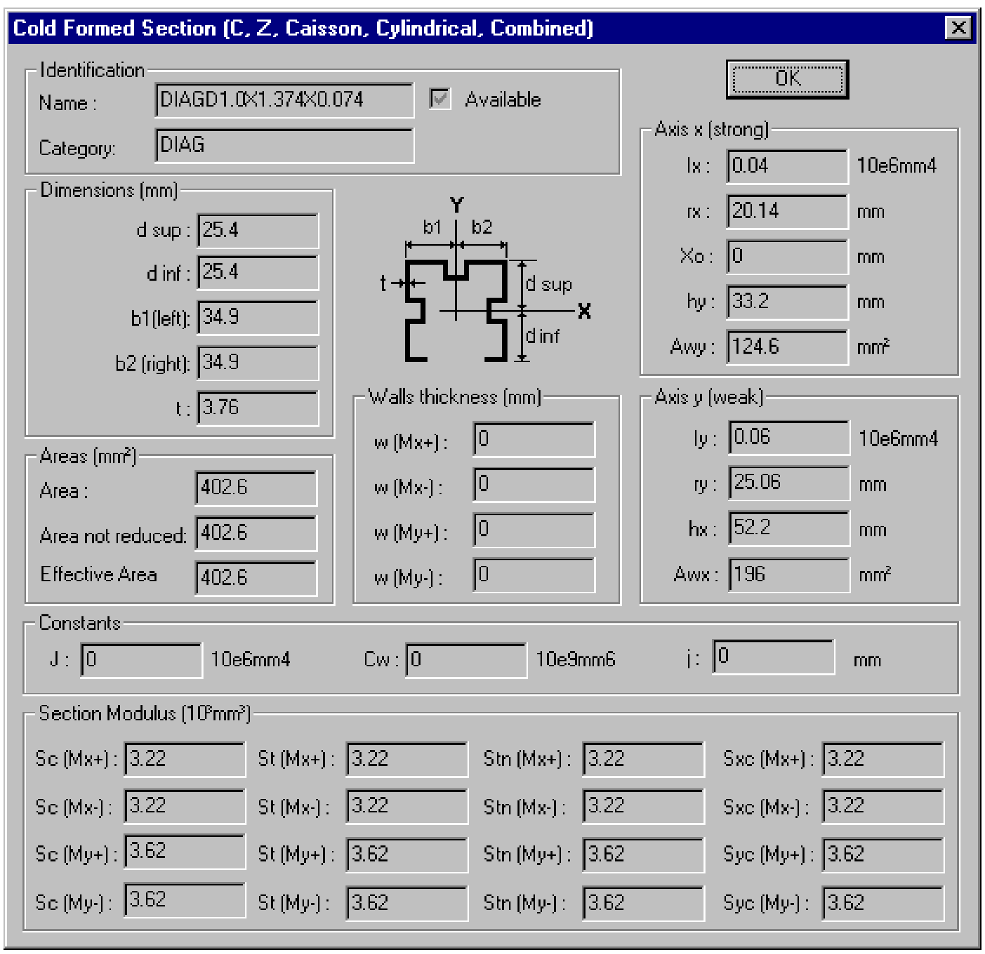
Prestressed Concrete Module

Prestressed Concrete Composite Beams

Cold-Formed Sections Spreadsheet

Cold formed sections are available for the following shapes: en C, Z, I, cylindrical, caisson and combined.

Refer to the S136 Standard for more details.



Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Category	Enter up to 20 alphanumerical characters to define the category: Useful to subdivide the type of thin wall section into several subgroups.	Single click
Type of section	Choose a type of cold-formed section: C, I, Z, Caisson, Cylindrical or Combined.	Double-click
Symmetry	Specify the type of symmetry of the section: Double symmetry, Axis X symmetry or Axis Y symmetry.	Double-click
d sup	Distance from centre to top fibre	Single click
d inf	Distance from centre to bottom fibre	Single click
b2	Distance from centre to right fibre	Single click
b1	Distance from centre to left fibre	Single click
d(τ|Tz)	Distance used for the calculation of shear stress d(Tau|Tz) * Tz / J. See the note below.	Single click
Area	Gross area of the section, used for the calculation of mass and EA/L.	Single click
Area not reduced	Gross area minus bolt holes. Used for the calculation of torsion buckling max stress (Cl. 6.6.3.1)	Single click

Column	Description	Editing
Effective Area	Section effective area for the calculation of Cr (Cl. 6.6.1.3)	Single click
Ix	Moment of inertia according to strong axis. Normally used with the unreduced section area.	Single click
Iy	Moment of inertia according to weak axis. Normally used with the unreduced section area.	Single click
IycMxp	Moment of inertia of the compressed zone of the unreduced area of the section relative to the neutral axis of the whole section parallel to the web(s), for Mrx positive.	Single click
IycMxn	Moment of inertia of the compressed zone of the unreduced area of the section relative to the neutral axis of the whole section parallel to the web(s), for Mrx negative.	Single click
rx	Radius of gyration according to strong axis. Normally used with the unreduced section area and for the calculation of compression and lateral buckling (Cl. 6.4.3 and Cl. 6.6.3)	Single click
ry	Radius of gyration according to weak axis. Normally used with the unreduced section area and for the calculation of compression and lateral buckling (Cl. 6.4.3 and Cl. 6.6.3)	Single click
Xo	Distance from shear centre to centre of gravity, in the x-axis direction.	Single click
t	Thickness of the thin wall	Single click
j	Lateral buckling constant. See the equation below.	Single click
J	Torsional constant. See the examples below.	Single click
Cw	Warping constant	Single click
Sc (Mx+)	Elastic section modulus in compression taken from the moment of inertia of effective area divided by the distance between neutral axis and the end fibre in compression (y+).	Single click
Sc (Mx-)	Elastic section modulus in compression taken from the moment of inertia of effective area divided by the distance between neutral axis and the end fibre in compression (y-).	Single click

Column	Description	Editing
Sc (My+)	Elastic section modulus in compression taken from the moment of inertia of effective area divided by the distance between neutral axis and the end fibre in compression (x+).	Single click
Sc (My-)	Elastic section modulus in compression taken from the moment of inertia of effective area, divided by the distance between neutral axis and the end fibre in compression (x-).	Single click
St (Mx+)	Elastic section modulus in tension taken from the moment of inertia of the section effective gross area divided by the distance between neutral axis and the end fibre in tension (y-).	Single click
St (Mx-)	Elastic section modulus in tension taken from the moment of inertia of the section effective gross area divided by the distance between neutral axis and the end fibre in tension (y+).	Single click
St (My+)	Elastic section modulus in tension taken from the moment of inertia of the section effective gross area divided by the distance between neutral axis and the end fibre in tension (x-).	Single click
St (My-)	Elastic section modulus in tension taken from the moment of inertia of the section effective gross area divided by the distance between neutral axis and the end fibre in tension (x+).	Single click
Stn Mx+)	Elastic section modulus in tension taken from the moment of inertia of the section effective net area divided by the distance between neutral axis and the end fibre in tension (y-).	Single click
Stn (Mx-)	Elastic section modulus in tension taken from the moment of inertia of the section effective net area divided by the distance between neutral axis and the end fibre in tension (y+).	Single click
Stn (My+)	Elastic section modulus in tension taken from the moment of inertia of the section effective net area divided by the distance between neutral axis and the end fibre in tension (x-).	Single click
Stn (My-)	Elastic section modulus in tension taken from the moment of inertia of the section effective net area divided by the distance between neutral axis and the end fibre in tension (x+).	Single click
Sxc (Mx+)	Elastic section modulus in compression of unreduced section area relative to X-X neutral axis perpendicular to web. (Ix is divided by the distance between neutral axis and end fibre in compression.)	Single click
Sxc (Mx-)	Elastic section modulus in compression of unreduced section area relative to X-X neutral axis perpendicular to web. (Ix is divided by the distance between neutral axis and end fibre in compression.)	Single click

Column	Description	Editing
Syc (My+)	Elastic section modulus in compression of unreduced section area relative to Y-Y neutral axis parallel to web. (Iy is divided by the distance from neutral axis to the end fibre in compression.)	Single click
Syc (My-)	Elastic section modulus in compression of unreduced section area relative to Y-Y neutral axis parallel to web. (Iy is divided by the distance from neutral axis to the end fibre in compression.)	Single click
Awx	Web net area – weak axis	Single click
Awy	Web net area – strong axis	Single click
hx	Web height in the direction of weak axis (longest component)	Single click
hy	Web height in the direction of strong axis (longest component)	Single click
wxp	Length used for the calculation of W, clause 6.4.4, for Mrx positive.	Single click
wxn	Length used for the calculation of W, clause 6.4.4, for Mrx negative.	Single click
wyp	Length used for the calculation of W, clause 6.4.4, for Mry positive.	Single click
wyn	Length used for the calculation of W, clause 6.4.4, for Mry negative.	Single click
a	Distance between transverse stiffeners; distance c/c between webs of caisson sections.	Single click
r1	Radius of gyration of non-reduced area of a particular section included in a combined section.	Single click
qsx		Single click
qsy		Single click
Q	For a class 4 section: This ratio is equal to f'y/fy, where f'y is corresponding to the equivalent yield stress of a class 3 section.	Single click
Perimeter	Perimeter of the section.	Single click

Calculation of Saint-Venant torsional constant

Open Sections

$$J = \frac{1}{3} * \sum_{i=1}^{n} I_i * (ti)^3$$

Where

I=median length of element sections

t= thickness of the element steel sections

Caisson Beams

$$\frac{2(ab)^2}{[(a/t_1)+(b/t_2)]}$$

Where

a = distance between web axes

b = distance between flanges axes

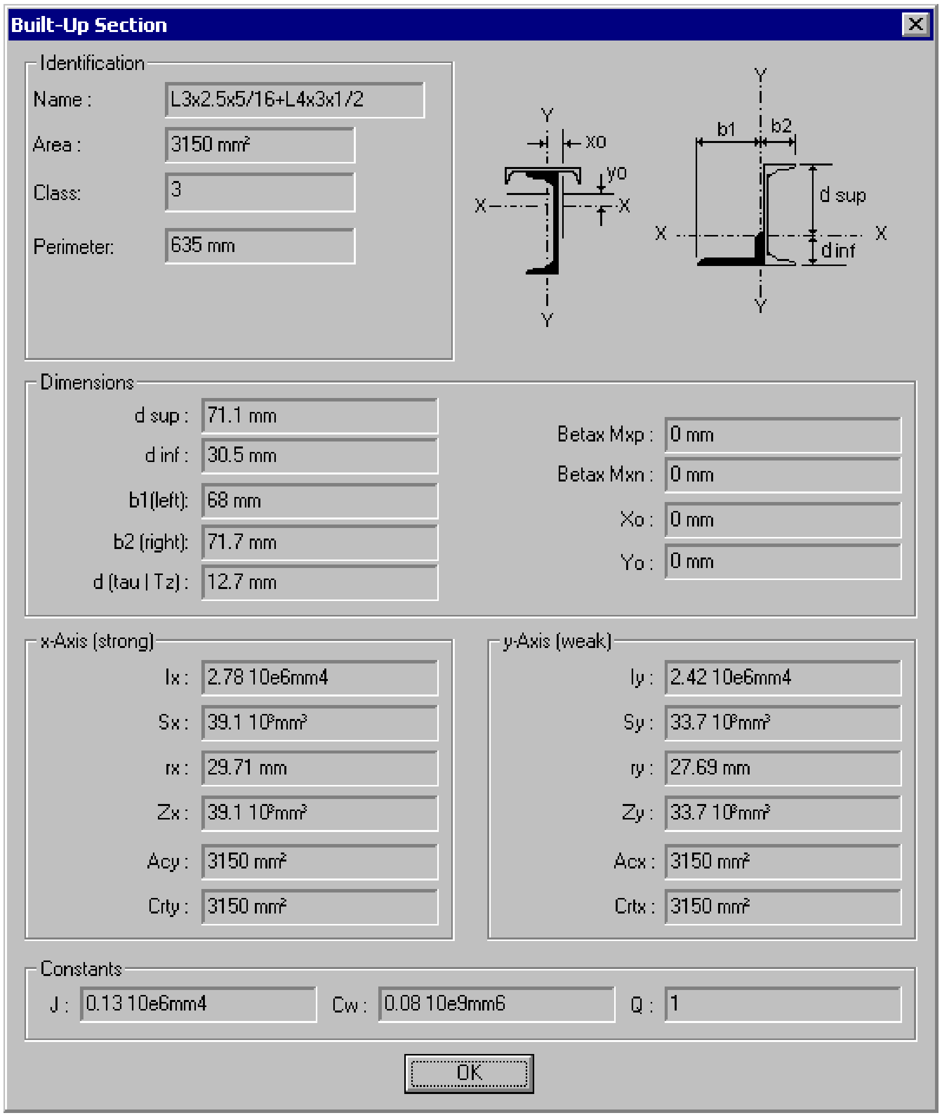
t1 = flanges thickness

t2 = webs thickness

Calculation of Lateral Buckling Constant:

$$j = \frac{1}{2Iy}\left[\int_A x^3 dA + \int_A xy^2 dA\right] + |Xo|$$

Built-up Shapes Spreadsheet



Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Metric Designation	The metric designation for this section (12 alphanumeric characters)	Single click
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material in the Material selection tree.	Double-click

Column	Description	Editing
Canada USA Europe	Shape availability in Canada, USA or Europe. These fields cannot be edited except for personalized shapes. If available in one of these countries, activate the cell [x] by double-clicking.	Double-click or Space bar
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
Class	Class of the Built-Up section.	Double-click
d sup	Distance between the centre and top fibre	Single click
d inf	Distance between the centre and bottom fibre	Single click
b2 (right)	Distance between centre and right end of fibre	Single click
b1 (right)	Distance between centre and left end of fibre	Single click
d(τ|Tz)	Distance to be used for calculating the shear stress in torsion: d(Tau|Tz) * Tz / J. See the examples below.	Single click
Area	Section area	Single click
Shear area dir.x	Shear area of the section on local x-axis, used to calculate the shear resistance.	Single click
Shear area dir.y	Shear area of the section on local y-axis, used to calculate the shear resistance.	Single click
Ix	Moment of inertia – strong axis	Single click
Iy	Moment of inertia – weak axis	Single click
Zx	Plastic section modulus – strong axis.	Single click
Zy	Plastic section modulus – weak axis.	Single click
J	Torsional constant	Single click
Cw	Warping torsional constant	Single click
Crt dir. x	Shear constant in the local x-direction.	Single click
Crt dir. y	Shear constant in the local y-direction.	Single click

Column	Description	Editing
βx Mxp	Coefficient of mono-symmetry used for calculating the positive bending strength on strong axis. Refer to T. Galambos, 5e éd., page 201.	Single click
βx Mxn	Coefficient of mono-symmetry used for calculating the negative bending strength on strong axis. Refer to T. Galambos, 5e éd., page 201.	Single click
xo	Distance between centre of gravity and shear centre, in the local x-direction, for mono-symmetric sections.	Single click
yo	Distance between centre of gravity and shear centre, in the local y-direction, for mono-symmetric sections.	Single click
Q	For a class 4 section: This ratio is equal to f'y/fy, where f'y is corresponding to the equivalent yield stress of a class 3 section.	Single click
Perimeter	Perimeter of the section.	Single click

Tau_max Computation Examples

For a cylindrical shape:

Tau_max = (Tz*r_ext)/J

Where

Tau_max: Shear stress for max torsion;

Tz: Torsion Moment;

r_ext: Exterior radius of cylinder (For a cylindrical shape, this distance corresponds to d(Tau|Tz).

For a built-up shape, this distance is measured from centre of gravity to extreme fibre.);

J: Torsional constant of the section;

Calculation of Tau_max for a single plate:

Tf = torque applied to the whole section

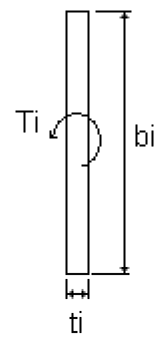
Ti = torque applied on one plate

JT = torsional constant of the section

Ji = torsional constant of one plate

$$\boxed{\tau max_i = \frac{T_i}{\left[\frac{1}{3}\cdot b_i\cdot\left(t_i\right)^2\right]}}$$

Where $J_i = \frac{1}{3}\cdot b_i\cdot\left(t_i\right)^3$



and $\tau max_i = \frac{T_f}{J_T}\cdot d$

$$\tau max_i = \frac{T_i}{J_i}\cdot d = \frac{T_i}{\left[\frac{1}{3}\cdot b_i\cdot\left(t_i\right)^2\right]}$$

Then, $d = \frac{J_i}{\left[\frac{1}{3}\cdot b_i\cdot\left(t_i\right)^2\right]} = \frac{\frac{1}{3}\cdot b_i\cdot\left(t_i\right)^3}{\left[\frac{1}{3}\cdot b_i\cdot\left(t_i\right)^2\right]}$

$$\boxed{d = t_i}$$

Therefore, the distance "d" corresponds to the plate thickness.

Calculation of Tau_max for an open section:

Tf = torque applied to the whole section

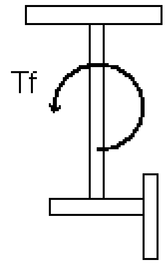
Ti = torque applied on one plate

JT = torsional constant of the section

Ji = torsional constant of one plate

Each plate will carry on a part of the applied torque according to its stiffness.

$$T_i = \frac{J_i}{J_T} \cdot T_f$$



$$J_T = \frac{1}{3} \cdot \left[\sum_n \left[b_i \cdot \left(t_i\right)^3 \right] \right]$$

$$\tau max_i = \frac{T_i}{\left[\frac{1}{3} \cdot b_i \cdot \left(t_i\right)^2\right]} = \frac{J_i}{J_T \cdot \left[\frac{1}{3} \cdot b_i \cdot \left(t_i\right)^2\right]} \cdot T_f$$

$$\tau max_i = \frac{\frac{1}{3} \cdot b_i \cdot \left(t_i\right)^3}{J_T \cdot \left[\frac{1}{3} \cdot b_i \cdot \left(t_i\right)^2\right]} \cdot T_f$$

$$\boxed{\tau max_i = \frac{T_f}{J_T} \cdot t_i}$$

Therefore, the maximum value of "d" will be equal to the biggest thickness of plates composing the section.

Composite Slabs

Composite Slabs Spreadsheet

This spreadsheet is used for composite beams only. The defined slab will be selected in the **Composite Beam** tab of **Member Characteristics**.

Group: Structural data

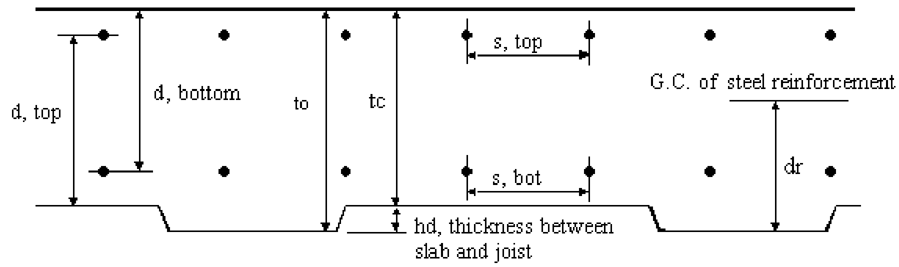
Column	Description	Editing
ID	Automatically calculated	No
Number	Number (description)	Single click
Steel deck	Double-click in the cell and choose a steel deck	Double-click
Direction	Direction of steel deck rib, relative to the beam (perpendicular, parallel)	Double-click
tc	Total slab thickness (without the steel deck)	Single click
hd	Thickness of the slab located above steel deck.	Single click
to	Total thickness (tc + hd)	No
Rod, top	Rebars located at top of slab.	Double-click
S, top	Spacing between top rebars	Single click
d, top	Distance between centre of gravity of top rebars and top of slab.	Single click
Rod, bot	Rebars located at bottom of slab.	Double-click
S, bot	Spacing between rebars at the bottom.	Single click
d, bottom	Distance between centre of gravity of bottom rebars and top of slab.	Single click
Rebar Material	Choose the rebar steel grade among the drop-down list box	Double-click
Concrete Material	Choose the slab concrete material.	Double-click

See also

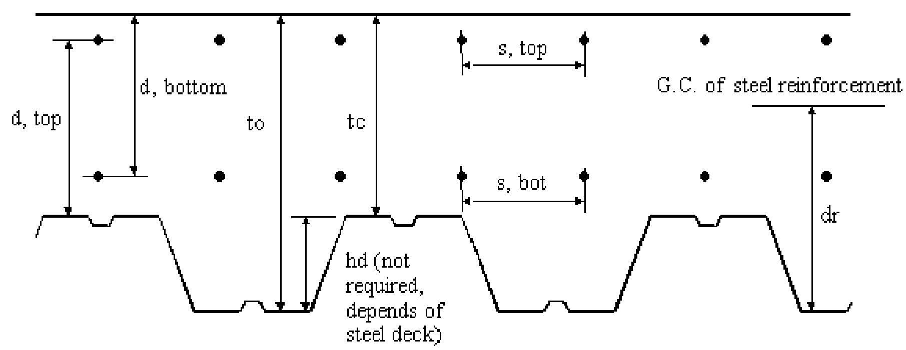
Types of Steel decks
Rebar Steel Grades
Type of Rebars
Defining a Member as Composite

Composite Slab Parameters

Slab with no steel deck



Slab with steel deck



Studs Spreadsheet

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Stud number or concise description	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Diameter	Diameter of the rod	Single click
Height	Stud total height	Single click
Fu	Tensile strength of steel.	Single click

Column	Description	Editing
Type	Type of studs (Only the Neilson type is available for the moment)	Double-click

Steel Decks Spreadsheet

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Steel deck number or concise description.	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Thickness	Thickness of steel deck.	Single click
hd	Height of steel deck.	Single click
w1	Bottom width	Single click
w2	Top width	Single click
wp	Slope width	Single click
wt	Length of the decks (information only)	Single click
nhd	Height of groove	Single click
nw2	Exterior width of groove	Single click
nw1	Interior width of groove	Single click
Composite	If the deck is textured to increase the composite action, choose option [x].	Double-click or Space bar
Type	Type of deck (Inclined grooves, Sinusoidal grooves, or Rectangular grooves, or Flat)	Double-click
Material	Material of the deck (code: ASTM A653 SS Grd 230)	Double-click

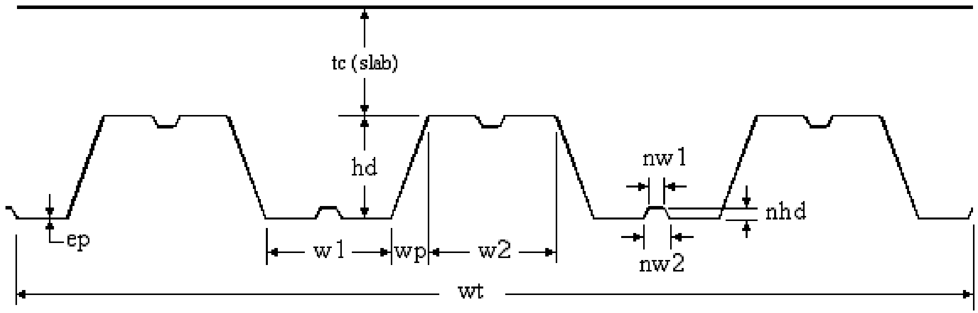
See also

Types of Steel decks
Slabs spreadsheet

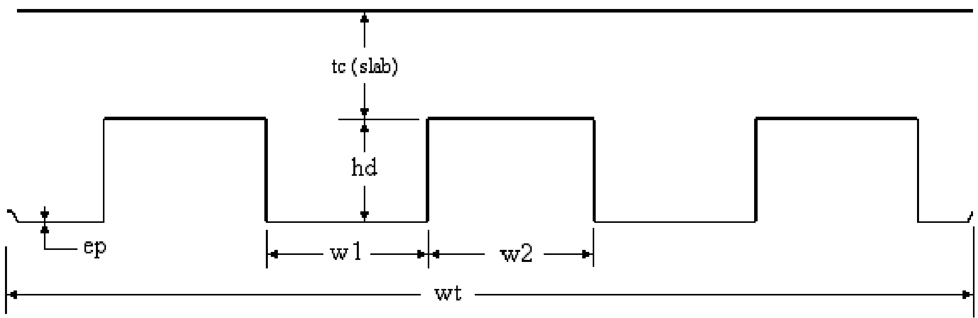
Floors spreadsheet

Types of Steel Decks

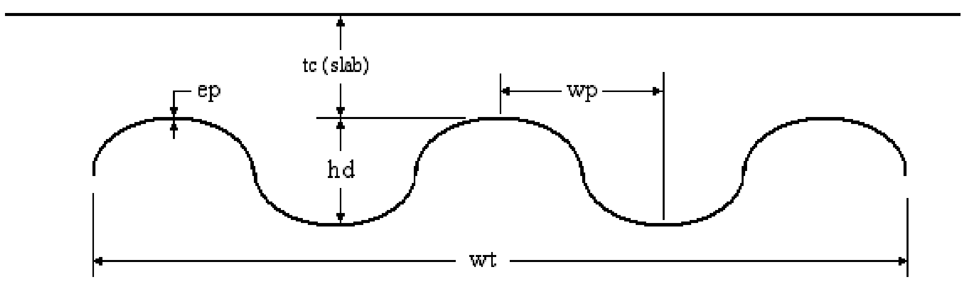
Inclined grooves (standard)



Rectangular grooves (wp, nw1, nw2 & nhd = 0)



Sinusoidal grooves (w1, w2, nw1, nw2 & nhd = 0)



See also

Steel decks spreadsheet

Slabs spreadsheet

Floors spreadsheet

Reinforcement

Rebar Steel Grades Spreadsheet

This spreadsheet, located in the **Common/Reinforcement** menu, includes a list of rebar steel grades.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Brief description of the rebar	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
E	Modulus of elasticity	Single click
G	Shear modulus	Single click
μ	Poisson's ratio	Single click
Density	Density of material	Single click
Thermal Coeff.	Coefficient of thermal expansion	Single click
Fy	Yield strength of steel.	Single click
Fu	Tensile strength of steel.	Single click
Weldable	If this steel grade is weldable, choose option [x].	Double-click or Space bar

See also

Rebar Bending Shapes

Types of Rebars

Standard Reinforcing Bars Spreadsheet

In this spreadsheet, you will find information about steel reinforcing bars: Diameters, bending dimensions according to grade (R,W), usage (S: anti-seismic), and composition (E: epoxy coating).

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters. Imperial rebars number must begin with symbol #.	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Area	Area of the steel bar	Single click
Diameter	Diameter of the steel bar	Single click
Linear Mass	Linear mass of the steel bar	Single click
Perimeter	Perimeter of the steel bar	Single click
Maximum length	Maximum manufactured length for this steel bar.	Single click
Colour	Colour assigned to this bar. To modify it, double-click in the cell and choose another one among the list box.	Double-click
k Factor Plain Bar	This factor (> 1.0) is used to calculate the development length for plain bars. (Dev. Length of plain bar = k * deformed rebar development length calculated by VisualDesign)	Single click
Surface	Surface of this steel bar: Deformed or Plain.	Double-click
DR	Standard mandrel diameter for a rebar of grade R	Single click
AG90R	Length A or G for a 90 deg. standard hook (grade R).	Single click
AG180R	Length A or G for a 180 deg. standard hook (grade R).	Single click
J180R	Height J of a 180 deg. hook (grade R).	Single click

Column	Description	Editing
DW	Standard mandrel diameter for rebar of grade W.	Single click
AG90W	Length A or G for a 90 deg. standard hook (grade W).	Single click
AG180W	Length A or G for a 180 deg. standard hook (grade W).	Single click
J180W	Height J of a 180 deg. hook (grade W).	Single click
DE	Standard mandrel diameter for an epoxy coated rebar.	Single click
AG90E	Length A or G for a 90 deg. standard hook epoxy coated.	Single click
AG180E	Length A or G for a 180 deg. standard hook epoxy coated	Single click
J180E	Height J of a 180 deg. hook with epoxy coating.	Single click
DS	Standard mandrel diameter for stirrups and cross ties.	Single click
AG90S	Length A or G of a 90 deg. standard hook for a rebar used as a stirrup or a cross tie	Single click
AG135S	Length A or G of a 135 deg. seismic hook for a rebar used as a hoop or seismic cross tie	Single click
Maximum Length	Rebar maximum manufactured length.	Single click

See also

Rebar Steel Grades
Bending Shapes

The FRP Reinforcing Bars Spreadsheet

In this spreadsheet, you will find information about fibre reinforced polymer bars. These bars can be used to reinforced concrete structures.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters. Imperial rebars number must begin with symbol #.	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Type of fibre	Select a type of fibre among the list: Glass, Carbon or Aramid.	Double-click
Area	Area of this bar.	Single click
Diameter	Diameter of this bar.	Single click
Linear Mass	Linear mass of this bar.	Single click
Perimeter	Perimeter of this bar.	Single click
Maximum Length	Maximum manufactured length for this bar.	Single click
Colour	Colour assigned to this bar. To modify it, double-click in the cell and choose another one among the list box.	Double-click
kb	This factor is used to calculate the development length for FRP bars. It must exceed 1.0. (Ex: Dev. length of FRP bar = kb * development length for deformed rebar, calculated by VisualDesign)	Single click
Ffu	Ultimate tension limit of the PRF bar.	Single click
Ef	Young modulus of the longitudinal PRF bar.	Single click
Coefficient for transv. thermal expansion	Specify the coefficient for transverse thermal expansion for this bar if temperature loads are applied to the structure.	Single click

Column	Description	Editing
Coefficient for longit. thermal expansion	Specify the coefficient for longitudinal thermal expansion for this bar if temperature loads are applied to the structure.	Single click
D	Mandrel diameter for a FRP bar	Single click
AG90	Length A or G for a 90 deg. standard hook	Single click
AG180	Length A or G for a 180 deg. standard hook	Single click
DS	Mandrel diameter for stirrups and cross ties.	Single click
AG90S	Length A or G of a 90 deg. standard hook for a FRP bar used as a stirrup or a cross tie	Single click
AG135S	Length A or G of a 135 deg. seismic hook for a FRP bar used as a hoop or seismic cross tie	Single click

The Meshes Spreadsheet

This spreadsheet, accessible through **Common / Reinforcement** menu, includes information about available meshes that can be used to reinforced concrete structures.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Name of this mesh (12 alphanumerical characters).	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Area	Area of bars composing this mesh.	Single click
Diameter	Diameter of bars composing this mesh.	Single click
Linear Mass	Linear mass of bars composing this mesh.	Single click
Perimeter	Perimeter of bars composing this mesh.	Single click
Maximum Length	Maximum manufactured length for this bar.	Single click

Column	Description	Editing
Colour	Colour assigned to this bar. To modify it, double-click in the cell and choose another one among the list box.	Double-click
Factor k Plain bar	This factor is used to calculate the development length for pain bars. It must exceed 1.0. (Ex: Dev. length of plain bar = k * development length for deformed rebar, calculated by VisualDesign)	Single click
Surface	Type of surface for bars that are composing this mesh.	Double click

Rebar Bending Shapes

This spreadsheet, located in the **Common/Reinforcement** menu, includes a list of standard bending shapes that can be used for Concrete Design.

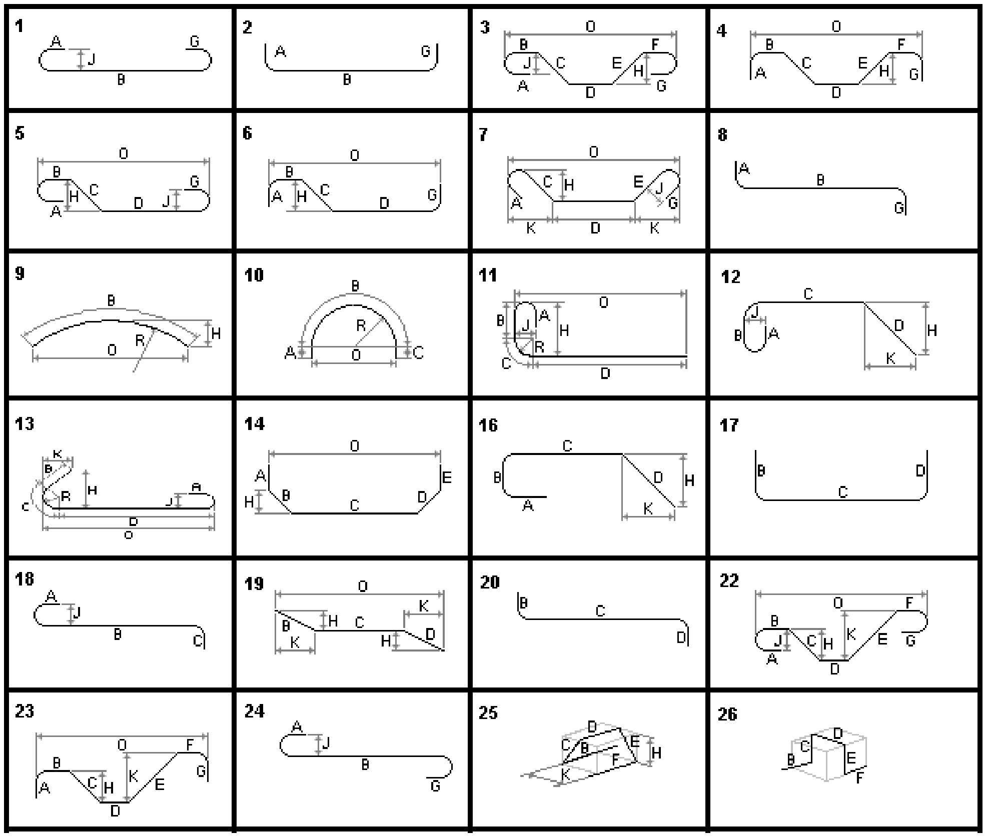
If you own the **Reinforced concrete design** module, you will choose bar bends within the **Main Reinforcement** spreadsheet and **Transverse Reinforcement** spreadsheet in order to design continuous systems in your structure. In fact, these two spreadsheets have a column titled "Bending Shape" which includes all the bending shapes that you will find in this table.

Group: Shared Data: VDBase.mdb

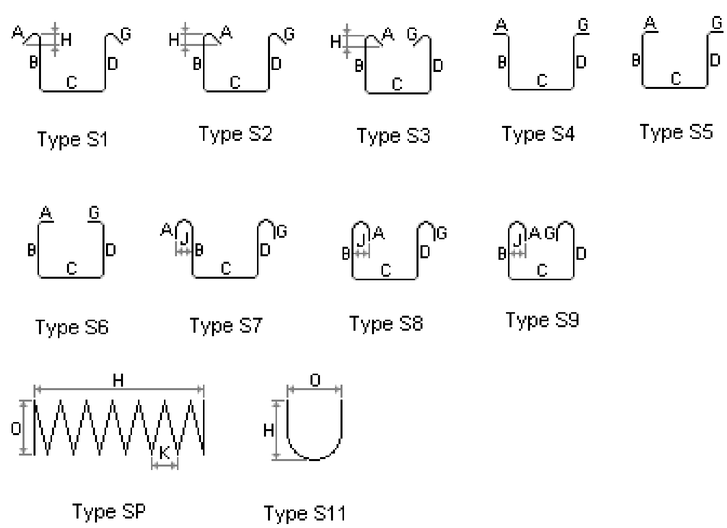
Column	Description	Editing
ID	Automatically calculated	No
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Image	Image of the bending shape	No
Bend number	Type of bend. See figures below	Double-click
Alias	Rebar bending type: C: tie E: stirrup F: hoop L: L bent bar R: bent-up bar U: U bent bar Y: miscellaneous bent bar	Double-click

Column	Description	Editing
Modified Alias	Some bending shapes may possess two aliases.	Double-click
Number of Vy planes	Number of stirrup legs that may be used in the calculation of the section shear capacity, according to strong axis.	No
Number of Vx planes	Number of stirrup legs that may be used in the calculation of the section shear capacity, according to weak axis.	No
Family	Bending shapes family: 1. Transverse Reinforcement: Closed tie Cross-tie Hoop Stirrup 2. Longitudinal Reinf. 1 plan 3. Longitudinal Reinf. 2 plans	No
Category	Type of reinforcement: main or transverse	No
Availability	If this bending shape is available, choose option [x].	No

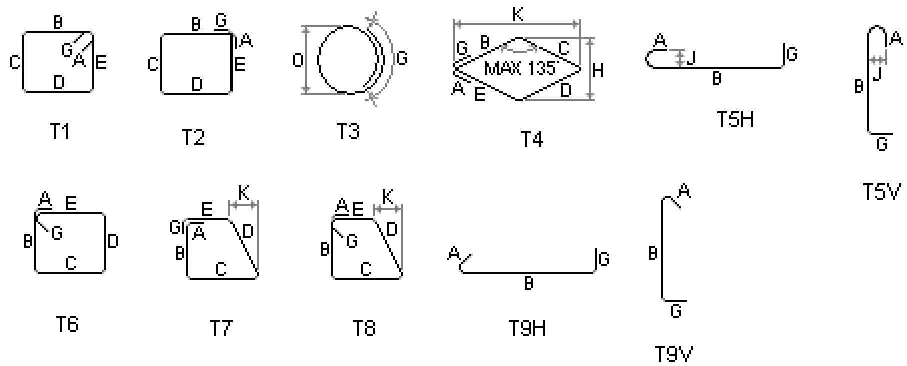
Types 1 to 26 and type L represent standard bending shapes for longitudinal reinforcement bars. They are shown below:



Types S1 to S11, SP included, represent bending shapes for transverse reinforcement bars for various concrete members. You will notice that the following bending shapes are open ties only:



Types T1 to T9 are closed ties (including hoop reinforcements, cross-ties and seismic cross-tie) for transverse reinforcements that must carry only shear stresses present in concrete members. These bending shapes are used in seismic design. They are the following:



See also

Types of Rebars

Rebars Steel Grades

Cables

Cable Steel Grades Spreadsheet

Define the steel grade that you will use in your prestressed concrete project. Select **Cables/Steel Grades** under **Common** menu.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Grade number (12 alphanumerical characters).	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Fu	Tensile strength for this grade	Single click
εa	0.008. See the note below	Single click
σ1	See the note below.	Single click
ε1	See the note below	Single click
k1	See the note below.	Single click
α	0.98. See the note below.	Single click

According to commentary C.8.4.3.2 "Stress-strain relationship" of S6-00 Code concerning strands:

For a low-relaxation 7 wire prestressing strand:

If $\varepsilon_p \leq 0{,}008$

We can say that:

$$f_p = \varepsilon_p \cdot E_p$$

If $\varepsilon_p > 0{,}008$

For a Grade 1760 Strand:

$$f_p = 1749 - \frac{0.433}{\varepsilon_p - 0.00614} < 0.98 \cdot f_{pu}$$

In VisualDesign:

$$f_p = \sigma_1 - \frac{k_1}{\varepsilon_p - \varepsilon_1} < \alpha \cdot f_{pu}$$

So: $\sigma_1 = 1749$

$k_1 = 0.433$

$\varepsilon_1 = 0.00614$

$\alpha = 0.98$

<u>N.B. Be careful with units. MPa are used in formulas.</u>

For a Grade 1860 Strand:

$$f_p = 1848 - \frac{0.517}{\varepsilon_p - 0.0065} < 0.98 \cdot f_{pu}$$

You will find the following values in the **Cable Steel Grades** spreadsheet:

$\sigma_1 = 1848$

$k_1 = 0.517$

$\varepsilon_1 = 0.0065$

$\alpha = 0.98$

See also

Strands spreadsheet

Strands Spreadsheet

Define the type of strands that will be included in your prestressed concrete project. Select **Cables/Strands** under **Common** menu.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Type	Choose a type of strand among the drop-down list: Low relaxation, Smooth high strength bars, Deformed high strength bars, Normal relaxation	Double-click
Diameter	Enter the strand diameter. (This value is used for screen display only.)	Single click
Area	Enter the strand area	Single click
Material	Choose the strand steel grade among the drop-down list. To add or modify steel grades, select the **Steel Grades** spreadsheet under **Common/ Cables** menu.	Double-click

See also

Prestressed Concrete Module
Post-tensioning Mechanisms
Cable Groups Spreadsheet
Cable Layouts spreadsheet
Cable Steel Grades Spreadsheet

Post-tensioning Mechanisms

Define the post-tensioning mechanisms that will be used in your prestressed concrete project. Select **Cables/Post-tensioning Mechanisms** under **Common** menu. Two types are available: with sheath or through external deviator.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Type	Choose a type of post-tensioning mechanism: Sheath or External deviator.	Double-click
Wobble friction, K	Enter the wobble friction coefficient per meter of strand length.	Single click
Friction coefficient, Mu	Enter the friction coefficient Mu.	Single click
Sheath diameter	Enter the sheath diameter (used for screen display only).	Single click
Length of deviator	Enter the length Ld of external deviator. This value is used to calculate prestress losses (friction between cable and deviator).	Single click
Deviator curvature	Enter the deviator curvature, ρd, for screen display only.	Single click

See also

Prestressed Concrete Module

Project Configuration (Prestressing tab)

Strands spreadsheet

Cable Group spreadsheet

Cable Layout spreadsheet

Bolts

Bolt Steel Grades Spreadsheet

Define the bolt steel grades that you will be using in the design or verification of bolted connections. In the **Common** menu, select heading **Bolts**/ **Steel Grades**.

Group: Shared Database VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Enter a number for this bolt steel grade (Up to 12 alphanumerical characters).	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click

See also

Bolts spreadsheet

Bolts Spreadsheet

Go to **Common** menu and select heading **Bolts**/ **Bolts**. For each type of bolts (metric and imperial) listed in this spreadsheet, you will find its diameter, nominal area, hole diameter, steel grade and corresponding tensile strength Fu.

Group: Shared Database VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Enter a number describing this bolt (Up to 12 alphanumerical characters).	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Nominal Area	Bolt nominal area	Single click
Diameter	Bolt diameter	Single click

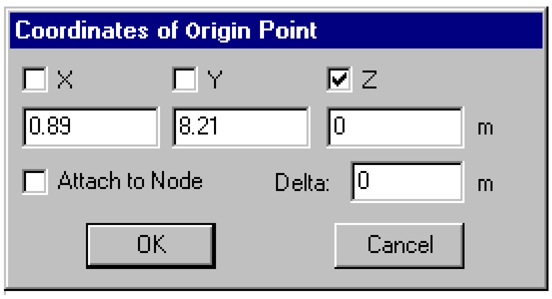
Column	Description	Editing
Hole Diameter	Required hole diameter for this bolt.	Single click
Head thickness	Thickness of the bolt head.	Single click
Head diameter	Diameter of the bolt head	Single click
Fu	Tensile strength for this type of bolt.	Single click
Material	Bolt steel grade	Double-click

Coordinates

Coordinates Dialog Box and Axis System

With VisualDesign™, you can work in 2D or 3D Cartesian system.

A **Coordinates** dialog box appears when you use commands **Add, Move** and **Rotate**. General axes accompany this dialog box. This dialog box allows you to point coordinates or define new ones. It can be entitled **Origin Point** or **Insertion Point**.



When defining an origin point or insertion point, coordinates can be selected by pointing and clicking or by typing in the **Coordinates** dialog box.

Shortcut keys:

x, **y** or **z**: Activates box *X*, *Y* or Z

a: Activates option *Attach to Node*

The coordinates are displayed as the pointer is moved.

Note. To select coordinates for one axis at a time, activate the respective axis check-off box.

"Delta" Option

When selecting coordinates using the pointer, values of coordinates can be varied in increments (e.g. 0.1m.). To do so, click on "Delta" and type in the value.

Note. To select coordinates one axis at a time, activate the respective axis check-off box.

"Attach to Node" Option

With the "Attach to node" activated, locating the pointer near a node and clicking with the left mouse button will attach the pointer to the node and select its coordinates in the dialog box.

Defining Coordinates

To store the coordinates of a point, use one of the following methods:

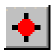
- With the left mouse button depressed move your cursor and press the "X", "Y" or "Z" keys on your keyboard when you want to select x, y or z values in the **Coordinates** dialog box. The respective check-off boxes will be activated and the values indicated. To store the coordinates and quit the dialog box, press the "OK" button.
- Enter manually the x, y, and z values in the **Coordinates** dialog box after you have activated the check-off boxes of these values. In activating those boxes, you freeze all moves from the cursor. To store the coordinates and quit the dialog box, press the "OK" button.

Note that if you want to move the cursor along one axis at the time, you can do so by blocking the two other axes.

Note. When working in 3D, VisualDesign™ offers the XY plane by default, blocking the z-axis. To unblock it, activate the corresponding check-off box.

Nodes

The Node Element



The "Node" icon of Elements toolbar

The node is an element located at the junction of one or more members, floors or plates.

There are two types of nodes: the normal node and the support node.

The node has its own icon on the Elements toolbar. It can be added or moved with the **Add** or **Move** icons of **Edit** toolbar. Furthermore, the Node icon must be selected to move or add a support.

See also

Nodes spreadsheet
Node Characteristics
Inactive Nodes
Forces on Nodes

Inactive Nodes

An inactive node is not linked to anything. We recommend that you delete inactive nodes that may be present in your structural model.

VisualDesign warns you that there are inactive nodes in your project when you launch a static analysis or a design. It is written in the **Analysis** or **Design** dialog box.

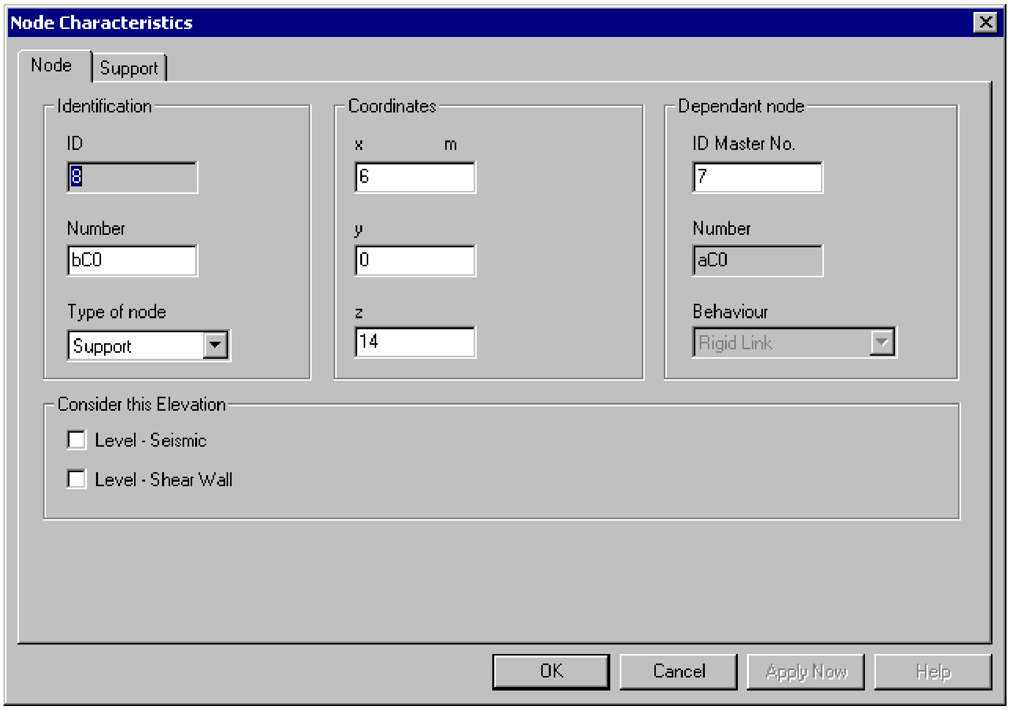
To find and delete these nodes, use the function **Select/Inactive Nodes** in **Edit** menu. Then, select them and press the [Delete] key.

Node Characteristics Dialog box

While in the "Structure" mode, access to the **Node Characteristics** dialog box by double-clicking on a node or a support. The dialog box will include the **Node** tab and **Support** tab if a support was selected. If nodes were selected, only the **Node** tab will appear.

To modify many nodes and/or supports in one click, select many of them and press the **Properties** icon. Enter values in blank fields.

The table below shows the definition of headings composing the **Node** tab.



Heading	Description
Identification	
ID	Identification number associated with a node (automatically calculated)
Number	Number of the node
Type of node	Specify if the node is a normal node or a support node
Coordinates	Enter the x, y, and z coordinates of the node
Dependant node	
Master node ID	Enter the ID of the master node associated with this node. The node becomes a slave node to this master node.
Number	Number associated to the master node ID.
Behaviour	This shaded field indicates the behaviour of the master node towards the slave nodes: n/a, Rigid links, Translation.
Consider this Elevation	VisualDesign automatically seeks for levels in the structural model (horizontal plates or floors) and calculates the dead load of each level. Activate options if there is no horizontal element indicating a level for VisualDesign, if you want to obtain results for particular levels.

Heading	Description
Level - Seismic	Activate this box to include this level into seismic analysis. You should specify a level near supports to minimize the non-participating mass that is distributed to supports. One node per level is sufficient.
Level - Shear Wall	Activate this box to obtain results at this level in the shear wall if there are openings. One node per level is sufficient.

(1) This box can be activated if you own the Dynamic Analysis module or Reinforced Concrete Design module (analysis of a shear wall).

See also

The Support tab
Inactive Nodes
Dynamic Analysis Module
Shear Wall
Reinforced Concrete Design module

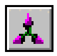
Nodes Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the node number.	Single click
Type	Type of node: normal or support.	Double-click
X Coord.	X-coordinate	Single click
Y Coord.	Y-coordinate	Single click
Z Coord.	Z-coordinate	Single click
Master Node ID	ID of the master node associated to this node.	Single click
Linked DOF	Degree of freedom between the master and slave node (n/a, rigid link, translation).	No
Level - Seismic	Activate option [x] to consider this node as a level in the seismic analysis.	Double-click or Space bar
Level - Shear Wall	Activate option [x] to consider this node as a level for shear wall design, if there is an opening in the wall.	Double-click or Space bar

Supports - General

The Support Element



The "Support" icon of Elements toolbar

The support is a node that is generally located at the base of a column, for example, and receives the loads that are transmitted by the columns to the foundations.

The support has its own icon on the Elements toolbar. A support can be added or moved by joining the Node icon to those of the **Add** or **Move** icons of the **Edit** toolbar.

See also

Support Characteristics Dialog Box
Support’s DOF
Support Orientation
Support in Analysis
Spring Supports
Support Release
Supports Spreadsheet
Spring Supports Spreadsheet
Released Supports Spreadsheet

Support in Analysis

There must be at least one support for each sub-structure. A structure might have sufficient supports, however, if one or more supports do not have rotational restraint (i.e. Fixed Mx, My or Mz), it will create structural instability. If this is the case, VisualDesign™ will warn you that a null pivot have been found in the stiffness matrix.

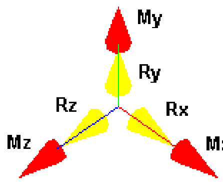
See also

Null pivot

Support's Degrees of Freedom and Behaviour

By default, translation degrees of freedom (Rx, Ry, Rz) are represented by yellow cones and rotation degrees of freedom (Mx, My, Mz), by red cones.

When the degree of freedom is released in one direction, the attached cone disappears.



By default, a new support is blocked in all directions (Rx, Ry, Rz, Mx, My, and Mz). However, it is possible to release one or more restraints in the **Support** tab of the **Node Characteristics** dialog box. Directions indicated in this tab are according to the support local axis system.

Behaviour of Spring Supports

A support can behave like a spring in one direction or more, to model the soil acting below a foundation slab or along walls, if you do not own the Foundation Design Module. In this case, you shall specify the support as a spring and enter the stiffness of the soil, K. Then, specify the tributary area of the soil acting on each spring support.

If finite elements are present and fixed to spring supports, the tributary areas of the soil acting on each spring support, in each direction, can be automatically calculated with this function Calculation of Tributary Areas (**Structure** / **Tools** menu). Refer to topic Spring Supports.

Secant Modulus K (Foundation Supports)

If you own the Foundation Design Module, VisualDesign can automatically calculate the soil rigidity. Select the option *Secant Mod. K* in the **Support** tab, for some degrees of freedom, or for all of them, and select a foundation model. The rigidity of supports will be calculated and written in the shaded fields next to each degree of freedom.

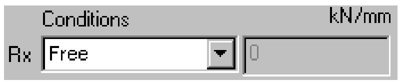
Support Axial Release

The support can be released in one direction at a time, negative or positive, to simulate uplift. It can also be considered inactive if released, meaning that the support degrees of freedom will be free, so the shear force will not be considered. Refer to topic Released Support.

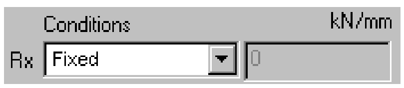
Here is how to block or unblock a support displacement or rotation in one direction, or to create spring behaviour:

Restraint and behaviour

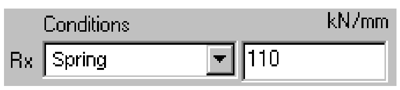
Displacement



The support is free to move along the local x-direction.



The support cannot move along the local x-direction.



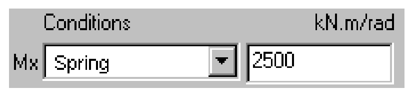
The support is elastic along the local x-direction. The linear rigidity constant of the spring must be entered according to displayed units. To modify units, open the Spring Supports spreadsheet and use the contextual menu.

The secant modulus K of this spring support is automatically calculated according to the stratigraphical profile, when a foundation model is selected in the **Support** tab.

Rotation

The rotation of support is allowed around local x-axis.

The support cannot rotate around local x-axis.



The elastic torsional rotation is allowed around the local x-axis. The constant for torsional stiffness must be entered according to displayed units. To modify units, open the Spring Supports spreadsheet and use the contextual menu.

The secant modulus K for torsional rotation is automatically calculated according to the stratigraphical profile, when a foundation model is selected in the **Support** tab.

See also

Supports Spreadsheet
Spring Supports
Released Supports

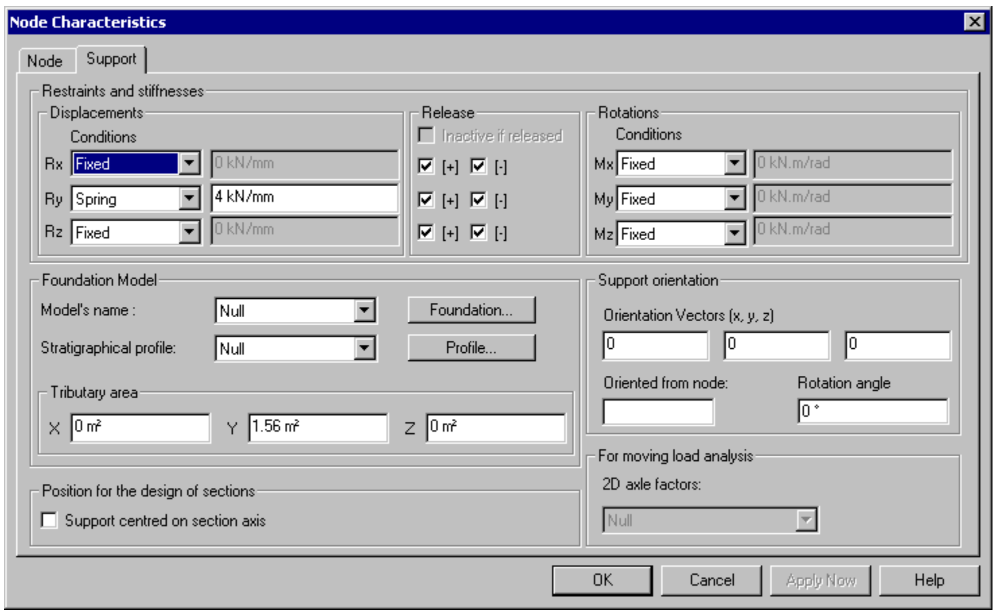
Static Analysis with Axial Release

When supports or members are released, you must run a static analysis considering axial release. This type of analysis, which is linear, must be activated in the **Analysis** tab of **Project Configuration** dialog box. You cannot run a design, which is a non-linear type of analysis, if there are members or supports with axial releases.

Support Characteristics Dialog Box

Support Characteristics

While in the "Structure" mode, activate the Support icon and access to the **Support** tab of **Node Characteristics** dialog box by double-clicking on a support. If the node element is activated on Elements toolbar, The **Node** tab will also be part of the **Node Characteristics** dialog box.



To modify many supports in one click, select many of them and press the **Properties** icon. Enter values in blank fields.

The table below shows the definition of headings composing the **Support** tab.

Heading	**Description**
Restrictions and Rigidities	Refer to Support's Degrees of Freedom
Support Release	Refer to Support Release
Foundation Model and Stratigraphical Profile	
Foundation	Select the name of foundation model to be assigned to this support. If the support was modeled using option *Secant Modulus K*, the spring stiffness will be automatically calculated and posted next to each support degree of freedom that uses this option.

Heading	Description
Stratigraphical Profile	For Culvert Generation and Abutments, Piers, and Retaining Walls Generation Modules: the stratigraphical profile is automatically selected for spring supports, which model the soil stiffness.
Tributary Area	Spring Supports only: Enter the tributary area of soil acting on the spring support. For spring supports with finite elements only: Call up the function Calculation of Tributary Areas. For Culvert Generation and Abutments, Piers, and Retaining Walls Generation Modules: The tributary area of soil is automatically calculated.
Support Orientation	Refer to Orientation of Support Node
Moving Load Analysis	Select the 2D axles factor that will be applied to this support. Refer to 2D Axle Factors for Supports
Position of Support During Design	For a support that is linked to a member having lateral rigid extensions, refer topic to Position of Support

See also

Support in Analysis

Support Results

Position of Support

Definition of Foundation Models

Orientation of a Support Node

The support can be oriented in a specific direction, identified by the user in the **Support Characteristics** dialog box. The support orientation is defined by its local axis z (z') that is oriented according to vectors x, y and z or towards a node specified by its number in **Node Orientation** field.

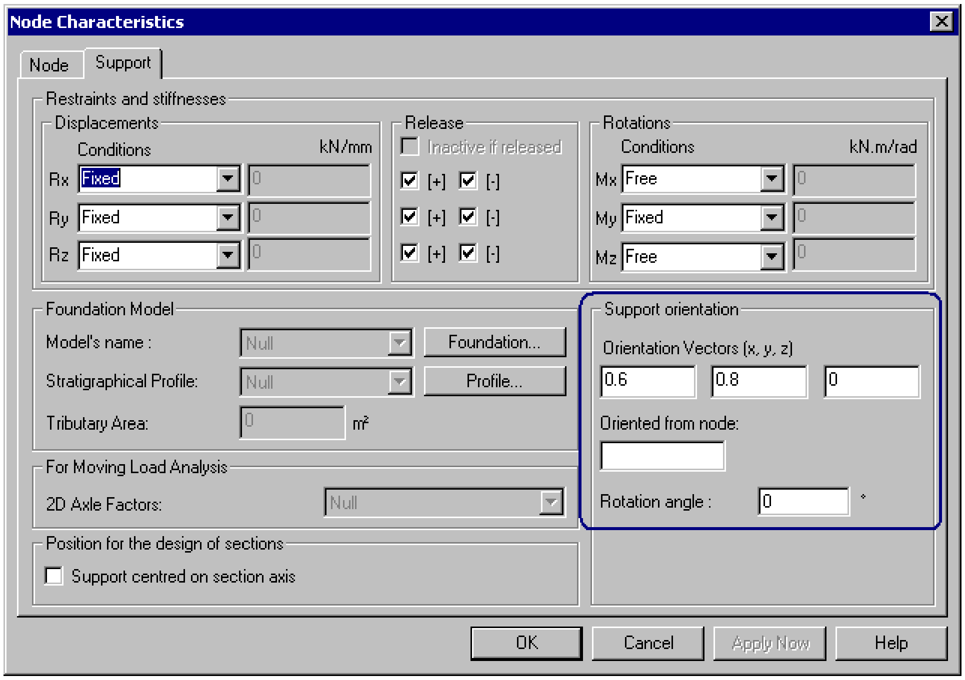
Orientation Vectors

The orientation of a support is always done through orientation vectors x, y and z.

Orientation Node

The support can be oriented according to a node. In this case, the user enters the node number in the appropriate field. Then, if the dialog box is open after orientation is done, orientation vectors will appear in the dialog box.

Note. If the orientation node is moved, the orientation of support local axis system will also change.



Rotation of the support

It may also be rotated about its local "z" axis by specifying an angle of rotation in the **Rotation** section. The direction of rotation follows the Beta angle convention for members.

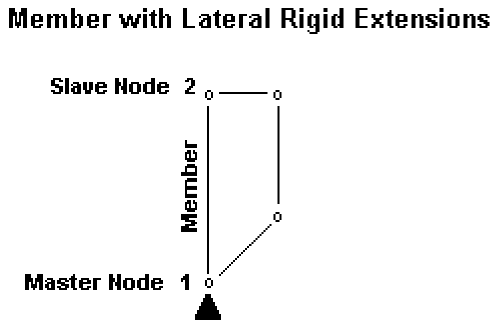
When the support node has no specified orientation, its local axis system is identical to the global axis system.

Position of Support during a Design

If a support is attached to a vertical member having lateral rigid extensions, do as follows:

Activate this box in the **Support Characteristics** dialog box to keep the support centred on the member axis in order to avoid offset and bending moment in the member. This particular case can happen when the steel shape grows bigger during a steel design.

Furthermore, this support node must be a master node and the end of the member, which is linked to this node, must be specified as a slave node. Look at the figure below.



See also

Rigid Extensions

Spring Supports

Spring Supports

Spring supports are mostly used to model the soil behaviour beneath a footing or next to a wall.

Spring supports are automatically generated when modeling culverts or piers, abutments and retaining walls using these generation modules. They are also modeled along piles when using the Foundation Design module.

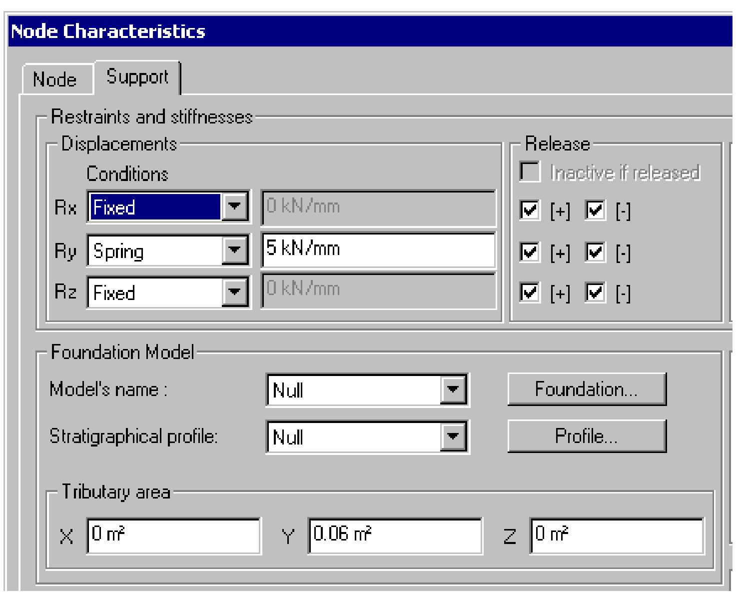
Soil stiffness

Open the **Support** tab of the **Node Characteristics** dialog box. To model spring supports, Rx, Ry, and/or Rz boxes must be checked and shaded. The soil stiffness, K, must be specified in the blank fields next to theses boxes, as shown below. This K value depends on the secant modulus of the soil, Ks, and on the tributary area of soil acting on a spring support. (K= Ks*A)

(References: Bowles, 5e Ed., page 303. The secant modulus of the soil, Ks, can be calculated using the Young modulus of the soil and the parameter μ (mu), which are supplied in VisualDesign's soils spreadsheet. Footing or wall dimensions and the thickness of the stratum are also required among others.

Tributary Area of Spring Supports

If spring supports are modeled with finite elements (plates), such as slabs or walls, use the function **Calculation of Tributary Areas** to quickly obtain the tributary areas of each spring support, in each direction. This function is available in **Structure** menu / **Tools**. The calculated values will be indicated in the **Support** tab and in the **Spring Supports** spreadsheet.



Releasing Spring Support for Uplift:

Sometimes, it is necessary to release spring supports, depending on the type of structure and its behaviour under particular loads. Uplift is one of these cases. Spring supports must be released towards the y-direction (if it corresponds to gravity axis). When a spring support is released, the soil will no longer act on the support if uplift occurs and reaction Ry will be null.

Static Analysis with Axial Release

When supports or members are released, you must run a static analysis with axial release. This type of analysis, which is linear, must be activated in the **Analysis** tab of **Project Configuration** dialog box. You cannot run a design, which is a non-linear type of analysis, if there are members or supports with axial releases.

Secant Modulus K for Spring Supports Assigned to Foundations Models

If you own the Foundation Design module, please refer to Foundation Supports.

Tributary Areas for Spring Supports

Use this tool, which is located in **Structure** menu / **Tools**, to automatically calculate tributary areas in the x-, y- and z-direction, for spring supports that are associated to plate elements.

Procedure:

- Activate the Structure activation mode.
- Select spring supports.
- Go to **Structure / Tools** and select **Calculation of Tributary Areas**.

Calculated areas will be written in the **Support** tab (**Node Characteristics** dialog box).

N. B. For the Generation of Abutments, Piers & Retaining Walls module, spring support tributary areas are automatically calculated from the stratigraphical profile data and are indicated in the **Support** tab.

Support Release

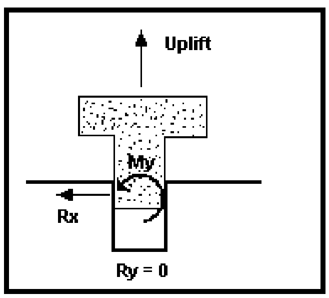
Support Release

VisualDesign allows you to release a support in the x, y and z direction (positive or negative). When a support is released in one direction (positive or negative), the corresponding reaction will be zero.

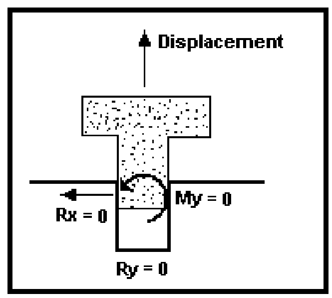
Spring supports can be released also.

Example 1: If this support is uplifted, the reaction Ry will be zero but Rx and bending moment My will not be zero.

Support released for y-



Inactive support if released for y-

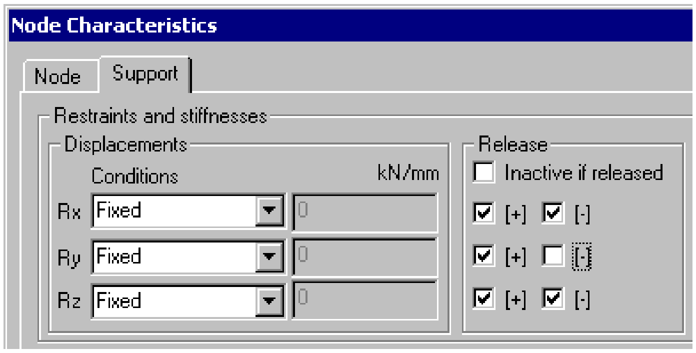


Procedure:

Go to **Project Configuration** *Analysis* tab and activate the analysis *With Release*. Increase the number of iterations to 15 or more (this number can be reduced after an analysis have been ran.)

Select the support and open the **Node Characteristics** dialog box. In the **Support** tab, you must first tick off the Rx or/and Ry boxes to indicate that the support cannot move. Then, in the "Support Release" zone, you will notice that the boxes corresponding to the positive or negative displacement are all checked. This status means that the support is normal (with no release).

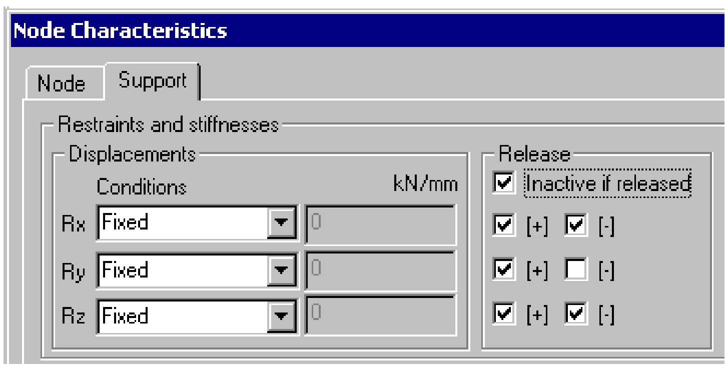
To activate a displacement in a positive or negative direction, uncheck one of the following boxes: Rx+, Ry+, Rz+ or Rx-, Ry- or Rz-. It is not allowed to release the support in the positive and negative direction at the same time.



In the dialog box shown above, the support release is activated in the negative direction of y-axis. This means that only the positive reaction Ry (upwards) will be considered in the analysis.

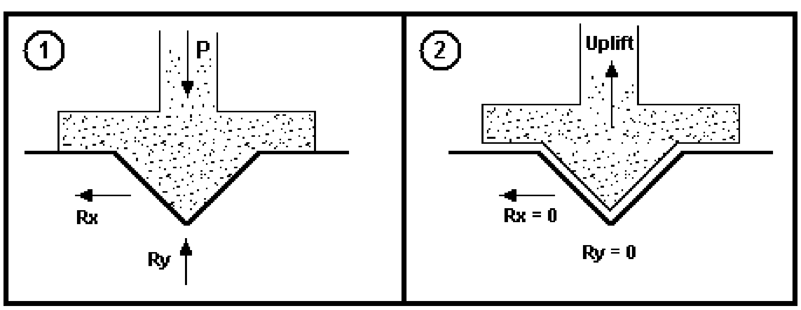
Inactive Support if Released

A support becomes inactive if all its degrees of freedom are free. To do so, activate option *Inactive if released* in the **Support** tab.



__Example 2__: If this support is uplifted (case 2), all reactions will be zero.

Inactive support if released for y-



Foundation Supports

Secant Modulus K

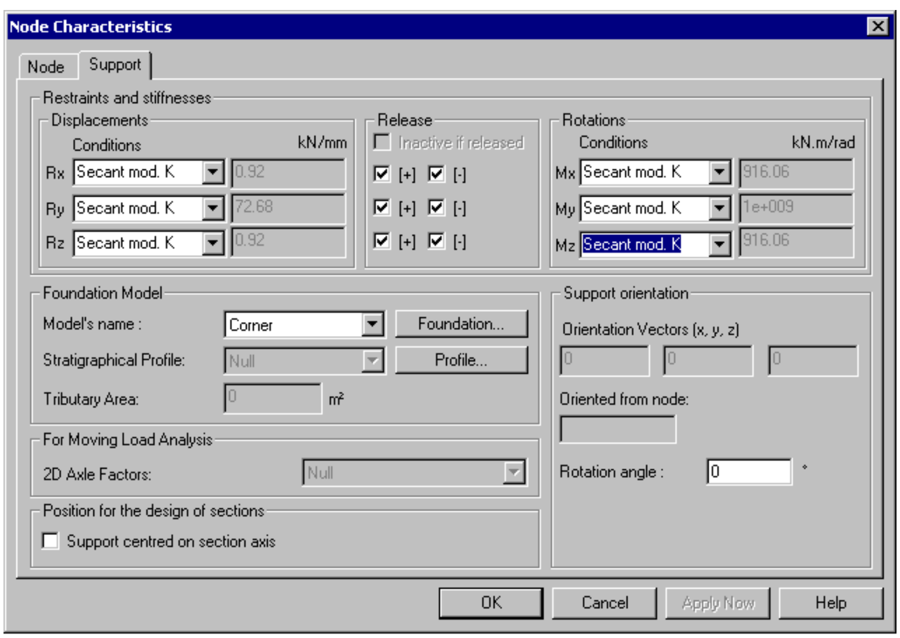
If you own the Foundation Design module, VisualDesign can automatically calculate the linear and torsional rigidity of spring supports. It is done using the soils properties composing the stratigraphical profile, which is specified in a foundation model. Lateral and vertical displacements of the footing or slab can be obtained from these calculated values.

The secant modulus replaces the *Interaction* status that considered the soil/structure interaction in previous compilations (231 and less).

Procedure:

- Define a shallow foundation model.
- Select supports and press the Properties icon.
- In the **Support** tab, choose option *Secant modulus K* for Rx, Ry, Rz, Mx, My, and/or Mz.
- Assign supports to foundation models.

The linear and torsional rigidity of the support will be calculated and written in the shaded field next to each degree of freedom corresponding to *Secant modulus K.*

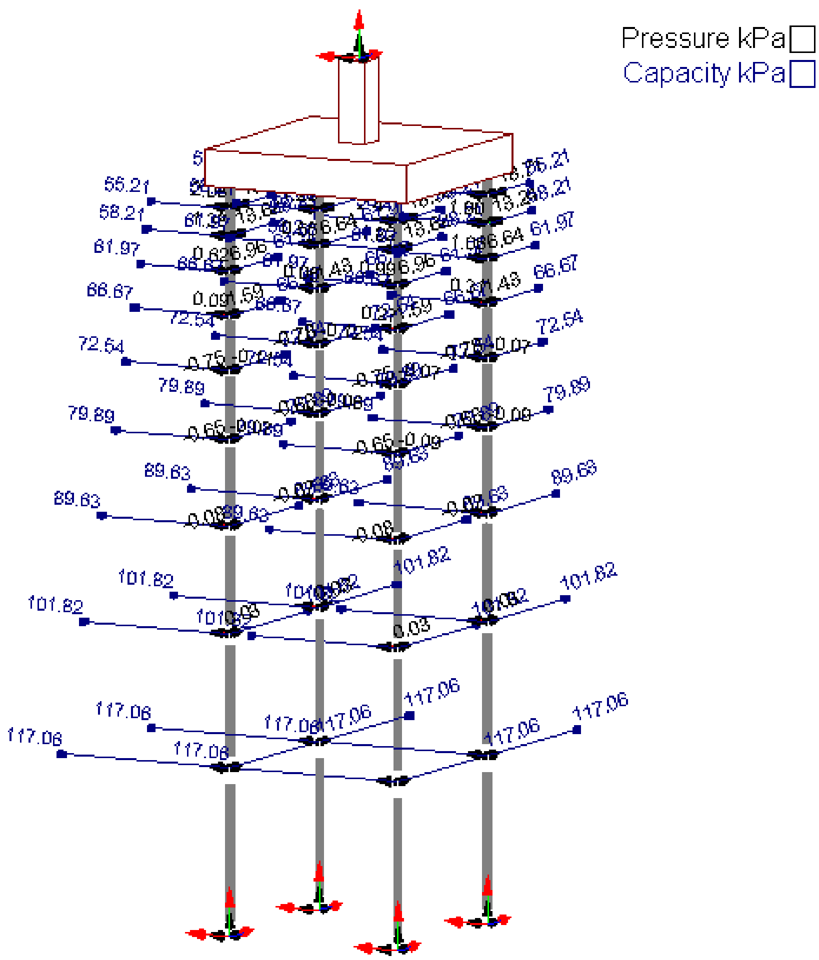


Pressure and Capacity of Spring Supports along Piles

Once that a pile foundation analysis is done, you can display pressures acting on each spring support located along piles along with the capacity of spring supports.

This option is available in the **Results** tab of **View Options** dialog box. Check the "Pressure and Capacity" box in the *Supports* section. A coloured legend will also appear to differentiate pressure diagram (orange) from the capacity diagram (blue). If you display numerical values, they will be the same colour as their respective diagram.

The displayed values correspond to the envelope minimum and maximum forces.



- Load Combination : DL1

Supports Spreadsheet

Supports Spreadsheet

This spreadsheet is split into three tabs: **Standard**, **Spring**, and **Released**.

Here is a description of the **Standard Supports** Spreadsheet columns:

The Standard tab

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the support number.	Single click
Rx	Displacement restraint in the x-dir.: Free, Fixed, Spring, or Secant Mod. K.	Double-click
Ry	Displacement restraint in the y-dir.: Free, Fixed, Spring, or Secant Mod. K.	Double-click
Rz	Displacement restraint in the z-dir.: Free, Fixed, Spring, or Secant Mod. K.	Double-click
Mx	Rotation restraint around x-axis: Free, Fixed, Spring, or Secant Mod. K.	Double-click
My	Rotation restraint around y-axis: Free, Fixed, Spring, or Secant Mod. K.	Double-click
Mz	Rotation restraint around z-axis: Free, Fixed, Spring, or Secant Mod. K.	Double-click
Orientation Vector X	Orientation vector, which orients the support according to global x-axis. Refer to Orientation of Support.	Single click
Orientation Vector Y	Orientation vector, which orients the support according to global y-axis.	Single click
Orientation Vector Z	Orientation vector, which orients the support according to global z-axis.	Single click
Angle	Angle of rotation of the support, around the local z-axis.	Single click
Foundation	Foundation model associated to the support.	Double-click

Column	Description	Editing
Axle Factor	Axle factor for moving load analysis. This column appears for owners of the Moving Load Analysis module	Single click
Centred on axis of section	For a design: If the support is linked to a member having lateral rigid extensions, select option [x]. Refer to Position of Support	Double-click or space bar

See also

Support DOF

Released Support Spreadsheet

Spring Supports Spreadsheet

Spring Supports Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the support number	Single click
Kx	Linear rigidity of spring support in the x direction, according to secant modulus K.	Single click
Ky	Linear rigidity of spring support in the y direction, according to secant modulus K.	Single click
Kz	Linear rigidity of spring support in the z direction, according to secant modulus K.	Single click
Krx	Torsional rigidity of spring support around the x axis, according to secant modulus K.	Single click
Kry	Torsional rigidity of spring support around the y axis, according to secant modulus K.	Single click
Krz	Torsional rigidity of spring support around the z axis, according to secant modulus K.	Single click
Profile	Stratigraphical profile below the spring support.	Double click
Tributary Area	Tributary area of soil acting on the spring support.	Single click

See also

Secant Modulus K for a Foundation Support

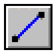
Released Supports Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the support number.	Single click
Release Rx+	Activate this option ([x]) to get a null Rx+ reaction if the support moves towards the negative direction of x-axis.	Double-click or Space bar
Release Rx-	Activate this option ([x]) to get a null Rx- reaction if the support moves towards the positive direction of x-axis.	Double-click or Space bar
Release Ry+	Activate this option ([x]) to get a null Ry+ reaction if the support moves towards the negative direction of y-axis.	Double-click or Space bar
Release Ry-	Activate this option ([x]) to get a null Ry- reaction if the support moves towards the positive direction of y-axis.	Double-click or Space bar
Release Rz+	Activate this option ([x]) to get a null Rz+ reaction if the support moves towards the negative direction of z-axis.	Double-click or Space bar
Release Rz-	Activate this option ([x]) to get a null Rz- reaction if the support moves towards the positive direction of z-axis.	Double-click or Space bar
Inactive if released	Activate this option ([x]) if you want the support inactive if it is released, meaning that the degrees of freedom will be free.	Double-click or Space bar

Members - General

The Member



The "Member" icon of Elements Toolbar

A member must be located between two nodes and must be split at the junction of other members.

The Member Dialog Box
The Member tab
Member Incidence
Geometry
Orientation of Beta Angle
Pre-tensioned Members
Member End Conditions
Member Properties
Member Usage
Member with a Linear Behaviour
Effective Stiffness
Types of sections
The Composite Beam tab
The Filled HSS tab
The Connection tab
The Behaviour tab

Beta Angle Convention

With VisualDesign™, member's vector ij corresponds to local axis z (z'). According to the CISC convention, the x' axis is considered as the strong axis and the y' axis, the weak one.

The x' axis is parallel to the horizontal plan of the global axes system while the y' axis always has a positive component on the vertical axis of the global axes system.

EXCEPTION CASE:

When z' is parallel to the gravity axis, x' can take an infinity of directions relatively to the horizontal global axes system. In fact, y' being also parallel to the horizontal plan of the global axes system, it does not have any component on the vertical axis which could help to determine x' orientation.

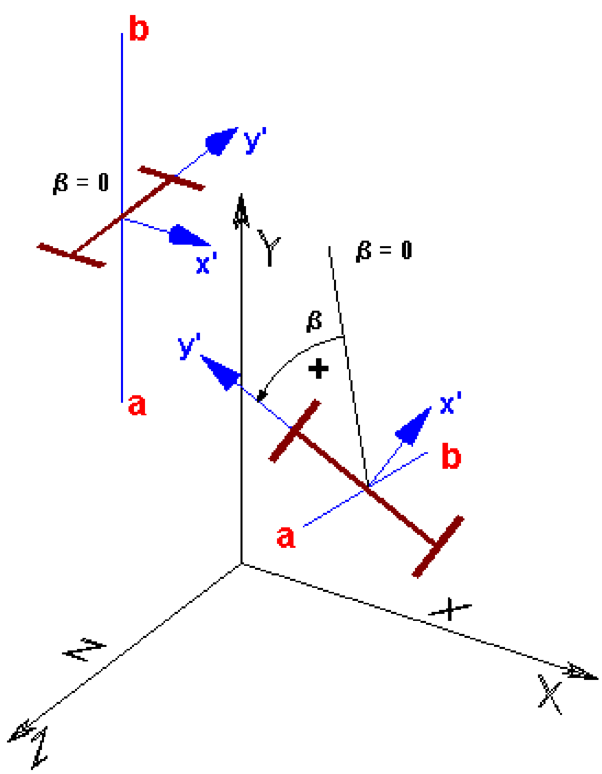
In this particular case, x' will then be parallel to the x-axis. Otherwise, x' will be parallel to y when the gravity axis is X.

Convention of beta angle beta and Strong Axis

According to this convention, we will have for the strong axis:

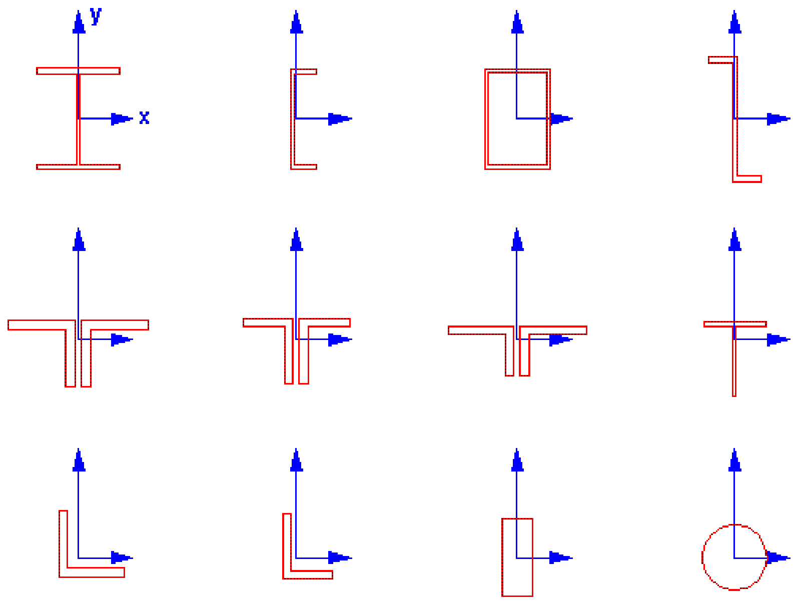
Ix	Inertia
Mx	Bending moment
Vy	Shear
rx	Gyration radius

Beta angle convention when Y is the gravity axis

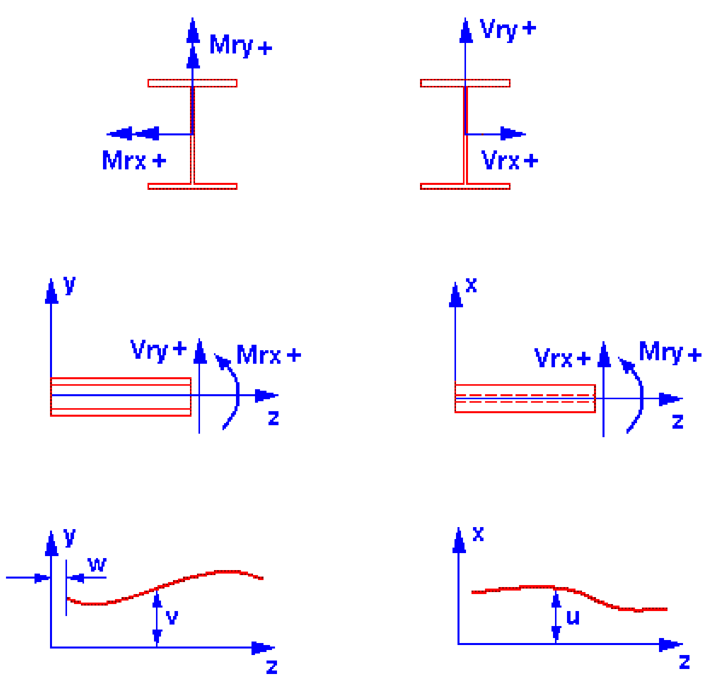


Convention - Forces in members

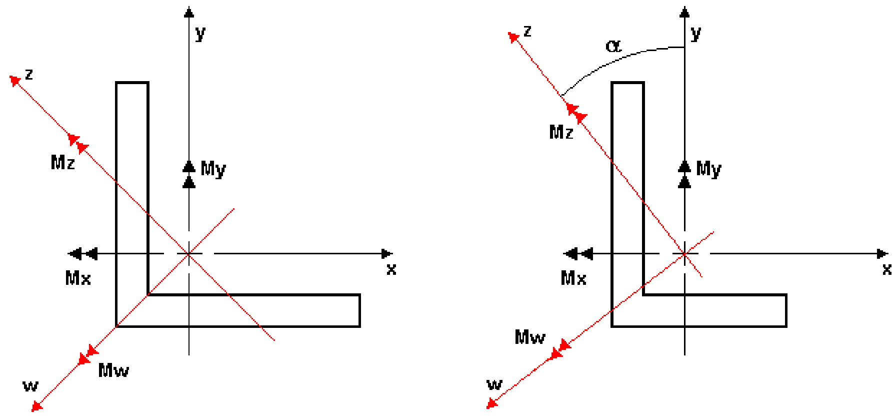
Sections' strong and weak axes:



The resistance of sections (Mr and Vr, ...) and results (internal stresses and deflections) are given in accordance to the local axes system (x, y, z). Local z-axis is longitudinal to the member.



For steel angles, Mw and Mz values are always transformed into orthogonal axis system once that the design is completed.



See also

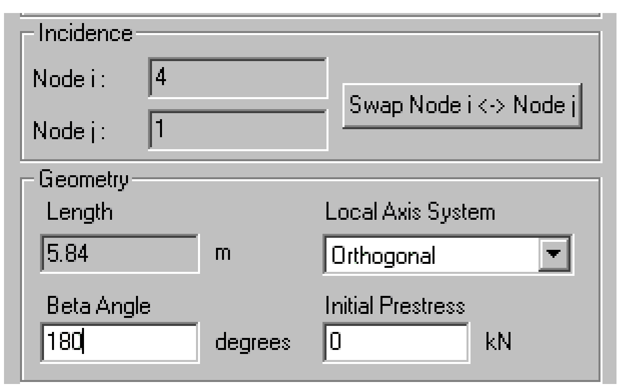
Major/minor and Orthogonal Axis Systems

Major/minor and Orthogonal axis system for steel angles

If you chose single steel angles and plan to run a design, we strongly recommend that you switch to a minor/major axis system once that the steel angle is properly positioned in space.

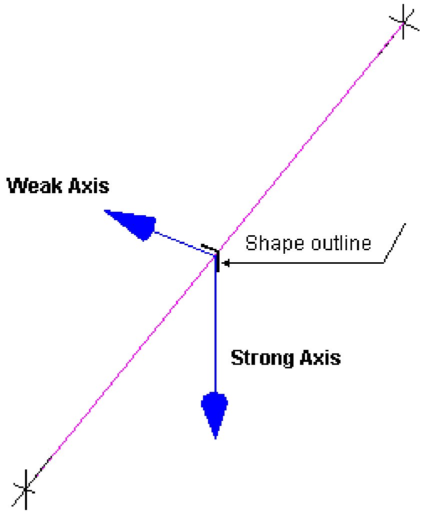
When modeling:

Use an orthogonal axis system in order to position the steel angle. The field "Local Axis System" of the **Member** tab (**Member Characteristics** dialog box) allows you to specify the orientation of beta angle. Select "Orthogonal" and enter a beat angle. Display the shape of the section through the **Attributes** tab (View Options) to make sure that it is correctly oriented.



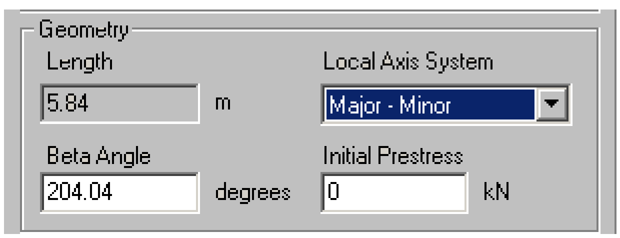
If you cannot orient the steel angle properly, use the "Invert Node i ↔Node j" option shown above.

Orthogonal Axis System and 180 deg. Beta Angle



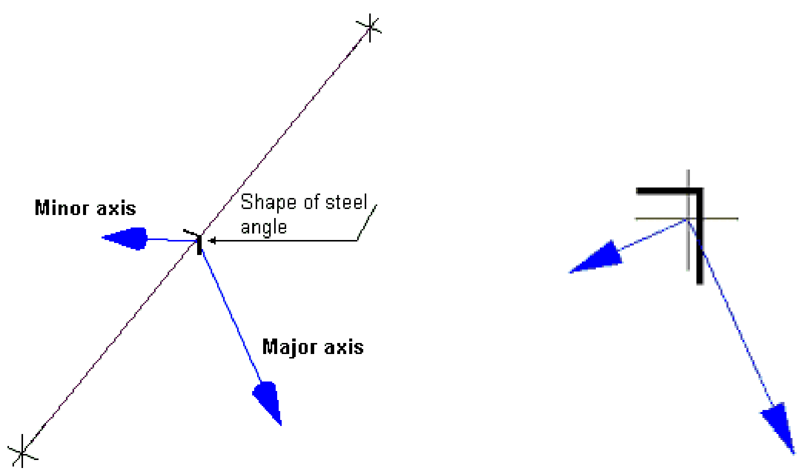
For a Steel design:

Once that the steel angle is correctly positioned, select all of them and switch to a major/minor. You will notice that the beta angle is automatically readjusted.



Major and minor axis will be displayed as follows:

Major/minor axis system



Remark Beta angle orientation can vary during cycles of design (when steel angle changes) in order to maintain the orientation of orthogonal axes.

See also

Convention – Forces in Member

Steel Design Criteria

Member Incidence

The member incidence is set when creating one. Node i is the first node selected and node j, the second. The member ij vector defines the local z (z') axis.

Display the member local axis system to look at the member incidence. The z-axis is always pointing towards node j.

It is possible to change member incidence by reversing nodes i and j order through the **Member** dialog box. To do so, press the button "Invert Node i ↔ Node j". Notice that when member incidence is modified, so are the member orientation and loads applied to the member.

Pre-tensioned Members

Select the **Member** tab of **Member Characteristics** dialog box and enter a pre-tension on the selected member.

If you are using some pre-tensioned members in your project, VisualDesign™ considers this tension within each load combination separately when analyzing the structure. So, **do not overlap (by hand) results** because you will add extra forces and results will be incorrect.

- Example: Bridge with pre-tensioned cables. Your load combinations are:
- First load combination: Dead load only
- Second load combination: Live load only
- Third load combination: Dead load + Live load.

VisualDesign™ will calculate as follow:

- Dead load with pre-tensioned members;
- Live load with pre-tensioned members;
- (Dead load + Live load), including pre-tensioned members.

And: (Dead load+pre-tension) + (Live load+pre-tension) ≠ (Dead load+Live load+pre-tension)

So, let VisualDesign™ combine load cases for you.

Member End Conditions

In the **Member** dialog box, you are asked to set the member end conditions for bending, torsion and axial behaviour. Note that Mx corresponds to bending in the strong axis, and My, the weak axis.

Following are the potential types of end conditions for internal forces:

Internal Forces	Symbols	Definition
Bending Mx	+------------+	Rigid – Rigid
	+------------O	Rigid – Hinge
	O------------+	Hinge – Rigid
	O------------O	Rigid – Hinge
Bending My	+------------+	Rigid – Rigid
	+------------O	Rigid – Hinge
	O------------+	Hinge – Rigid
	O------------O	Hinge – Hinge
Torsion Mz	+------------+	Rigid – Rigid
	+------------O	Rigid – Hinge
	O------------+	Hinge – Rigid
Axial Fz (1)	<->[ ]<->	Tension/Compression
	<-[ ]->	Tension only
	->[ ]<-	Compression only

Note 1

To model bracings with tension-only behaviour, select an tension-only axial end condition (<-[ ]->) and select option "Diagonal[Xt] " as member usage.

For compression-only member, select a compression-only axial end conditions (->[ ]<-) and run an analysis with release (activate this type of analysis in the **Analysis** tab of **Project Configuration** dialog box.).

If you are planning a dynamic analysis and a ductile steel design according to section 27 of S16-01 Standard, refer to section *Seismic Steel Design*, in this chapter.

See also
The Member Dialog Box

Analysis tab
Split Functions and Member End Conditions
Seismic Steel Design (S16-01)

Released Members

If you want some members to act as compressed members only (->[ ]<-), specify such conditions in the "End Conditions" section of **Member Characteristics** dialog box.

Then, you must activate option *Analysis with Release* in the **Analysis** tab of **Project Configuration** dialog box. Increase the number of iterations to 15 or more (this number can be modified once that the static analysis is completed).

This type of analysis is a linear type of analysis, but is an iterative one: When VisualDesign will encounter members that are specified with released conditions, it will correct the members' degrees of freedom and launch another linear analysis until it reaches convergence.

See also

Analysis tab (Project Configuration)
Released Supports
Type of Static Analyses

Member with a Linear Behaviour

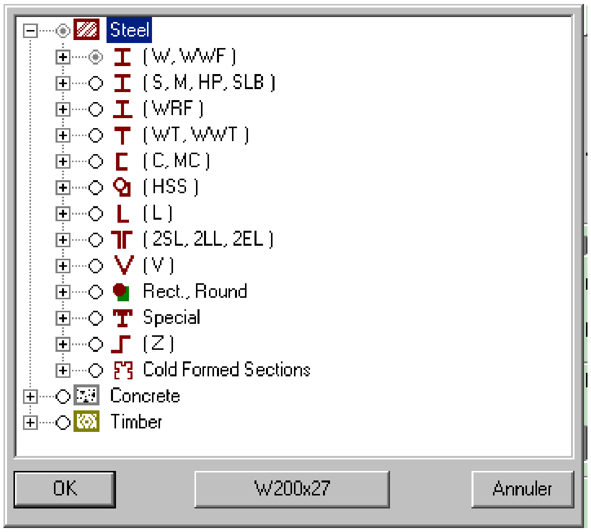
It is possible to define a member having a linear behaviour at all times even in a non-linear analysis. To do so, select option "Linear only" among the *Behaviour* drop-down list box of the Member tab.

This functionality is useful to model the small members that are located between bridge supports and pier supports. With a linear behaviour, these members will not create horizontal components (usually created in a non-linear analysis). Consequently, only axial forces will be transferred and the convergence will be faster than before.

Selection of a shape

Press this icon to call up the shape selection tree. This selection tree is composed of roots "Timber", "Steel" and "Concrete". Each one of these roots includes a list of available shapes for the chosen material.

To choose a shape, expand the root up to leafs and activate the radio button corresponding to a shape. Then, click OK.



Look at the shape properties dialog box by clicking the button displaying the name of the shape.

HSS Design Thickness

The design thickness of a selected HSS shape must be specified in the **Member** tab (**Member Characteristics** dialog box). HSS properties are adjusted with respect to this design thickness.

The display of such shapes on screen and its nomenclature in the design brief is as follows: HS305x254x9.5 (0.9t).

Note Dead load is calculated without considering the thickness reduction. Consequently, dead load applied on the structure and bill of materials are adjusted to usual HSS (full wall thickness).

See also

Member tab

Selection of Shapes

HSS spreadsheet

Definition of Sections

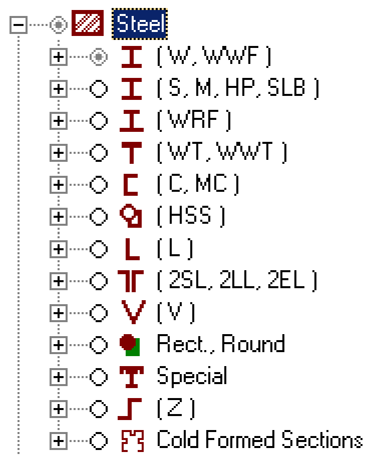
In the "Section Properties" of **Member** tab (**Member** dialog box), you must specify the shape, material and composition of the section.

When you will press the ***I Beam*** icon, a *tree* composed of shapes will appear. If the shape that you wanted to choose is not part of this tree, create it inside the appropriate spreadsheet. Then, the tree will include your shape.

Procedure:

To create new timber sections, go to the Rectangular or Round spreadsheets. Enter dimensions and choose a timber material among the drop-down list box. These new sections will be part of the Shapes *Selection tree*.

The table below shows the section and material that you must choose, and the parameters you must specify in a particular tab in order to run an analysis and/or a design for the chosen type of section.

Type of Section	Section	Properties	Action
Steel Beam		Material: Steel Composition: Standard	Project Configuration: Modify default parameters in **Steel** tab. Member: Specify design parameters in the **Steel Design** tab.
Composite Beam (steel-concrete)	All I, S, and WRF shapes	Material: Steel Composition: Composite beam The member is effective at construction stage 1, by default.	Define a slab Activate construction stages in the **Composite Beam** tab (Project Configuration) Member: Specify composite parameters in **Composite Beam** tab (Member dialog box) Define construction stage load combinations.

Type of Section	Section	Properties	Action
Composite Beam (concrete-concrete)	Solid sections, T-Section, AASHTO, and NEBT only. Concrete [T] Rectangular and Round Special [L(t,w)] [AASHTO] [NEBT]	Material: Concrete Composition: Composite beam The member is effective at construction stage 2, by default.	Define a concrete slab. Modify default parameters in **Concrete Design** tab (Project Configuration) Member: Specify design parameters in **Composite Beam** tab and **Concrete Design** tab.
HSS (Filled with concrete)	[HSS] Square HSS Round HSS Rectangular HSS Pipe	Material: Steel Composition: Filled HSS	Design: choose a concrete material in the **Filled HSS** tab.
Reinforced concrete	Concrete [T] Rectangular and Round Special [L(t,w)] [AASHTO] [NEBT] N.B. Use L (t, w) sections as edge beams.	Material: Concrete Composition: Standard	Modify default parameters in **Concrete Design** tab (Project Configuration) Member: select the **Concrete Design** tab.
Timber	Timber Rectangular and Round Round Rectangular [] SP MT GL GL: glue laminated	Material: Timber (See Timber Properties) Composition: Standard	Member: select the **Timber Design** tab.
Aluminium	All appropriate sections.	Material: Aluminium Composition: Standard	Analysis only.

Member Usage

Member usages are required for some modules only because VisualDesign needs to recognize particular members to properly design them.

Usages are selected in the "Usage" selection tree included in the **Member** tab of **Member Characteristics** dialog box. This selection tree is composed of three roots: *Wood Truss*, *Ductility* and *Towers*. To select a usage, expand the root and branches and double click the chosen usage.

TOWER DESIGN MODULE

The following usages must be assigned to members that are composing a tower: Upright, Guy, Horizontal, Diagonal, Horizontal Secondary, Diagonal Secondary, Vertical Secondary, Stabilizer, and Internal Bracing.

N.B. Usage must also be specified when wind loads and ice loads are applied to tower members.

TIMBER DESIGN MODULE

The usage for vertical and diagonal members within a wood truss must be set to "Secondary". Usage is not required for other wood members.

SEISMIC DESIGN AS PER CAN/CSA-S16-01 – STEEL DESIGN MODULE & DYNAMIC ANALYSIS MODULE

VisualDesign recognizes members that will dissipate energy during an earthquake through their usages. Usages are specific to types of braced systems and corresponding ductility.

Quick Selection of Members according to Usage

Go to **Edit / Select / Usage** and select a type of usage in the dialog box. Members having the same usage will be selected on screen.

Select Members according to Usage

Effective Stiffness

If you do not possess the *Reinforced Concrete Design* module, you can analyze your structure all the same using the cracked section of concrete elements in your structure. This function is part of the **Member** tab that is included in the **Member Characteristics** dialog box.

Enter the ratio or percentage for the reduction of the section gross inertia to obtain an effective stiffness as specified per code. If you do not wish to reduce the stiffness, enter a value of 1.0.

Effective bending stiffness on strong axis:

- Effective EIxx = Ratio of bending stiffness * EIxx

Effective bending stiffness on weak:

- Effective EIyy = Ratio of bending stiffness * EIyy

Effective shear and torsion stiffness:

- Effective GJ = Ratio of shear stiffness * GJ
- Effective G*A*kx = Ratio of shear stiffness * G*A*kx
- Effective G*A*ky = Ratio of shear stiffness * G*A*ky

See also the topic Shear Energy. VisualDesign™ takes into account the shear energy in the deflection calculation.

Effective axial stiffness:

- Effective EA = Ratio of axial stiffness * EA

See also
Member Dialog Box

Member Dialog Box

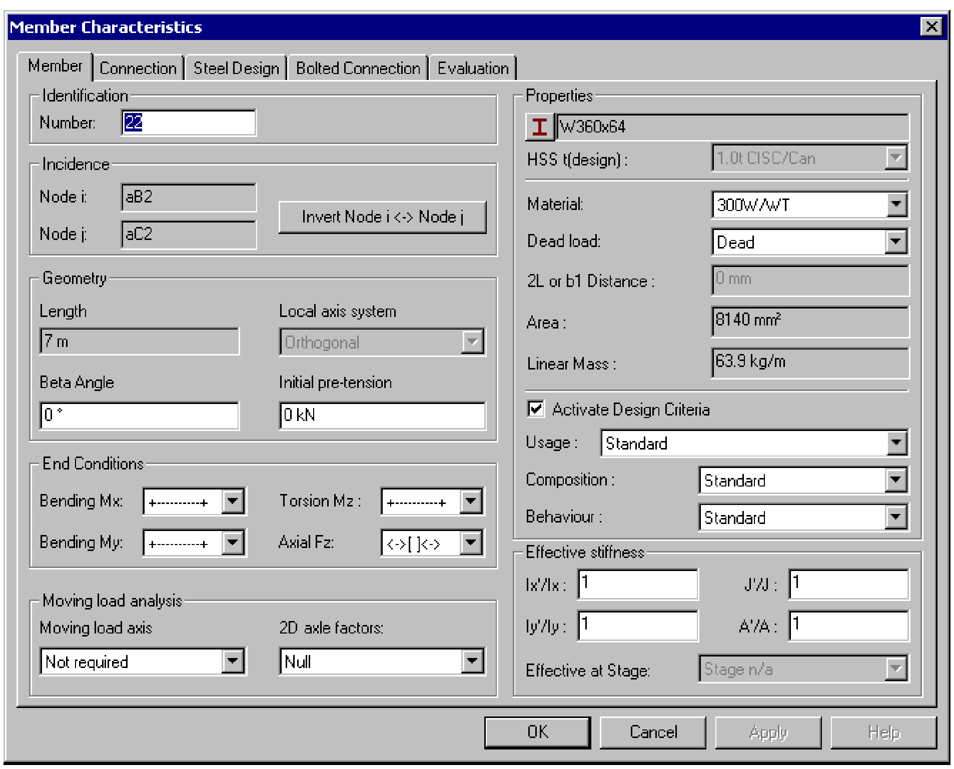
Member Dialog Box - General

While you are working in the "Structure" activation mode, you can access the **Member Characteristics** dialog box by double-clicking on a member (after having activated this type of element on Elements toolbar) or by clicking on a member and pressing the **Properties** icon.

This dialog box may be composed of several tabs. The most important one is the **Member** tab, which is the "master" one. In fact, this tab contains basic data about the member such as material, shape, composition, behaviour, activation of design criteria, etc.

Specific information is entered using the **Connection** tab, **Composite Beam** tab, **Filled HSS** tab, and **Behaviour** tab. If design criteria were activated in the **Member** tab, there may be a **Steel Design** tab and **Bolted Connections** tab appearing in the dialog box, a **Concrete Design** tab, if he owns the corresponding modules (Steel Design or Reinforced Concrete Design Module), the **Evaluation** tab (Bridge Evaluation module), or the **Timber Design** tab (Timber Design module).

Member tab



The table below shows the description for each topic included in the **Member** tab.

Field	Description
Identification	
Number	The member number.
Incidence	
Node i, Node j	Node i and node j numbers. Please refer to Member Incidence
Invert Node i ⇔ Node j	Click this button to invert the node incidence. Warning: Applied loads will be inverted too.
Geometry	
Length	The member calculated length (shaded).
Local axis system	Steel angle only: Select an orthogonal axis system (standard) or Minor/Major axis system. Refer to Orthogonal and Major/Minor Axes System
Beta angle	The shape position in space. Refer to Beta Angle Convention
Initial pre-tension	Indicate the magnitude of initial pre-tension, if any. Refer to Initial Pre-tension
End Conditions	Specify member end conditions at node i and j. Symbols are: o : hinged + : fixed (continuous beams or columns) Refer to End Conditions
Properties	
Icon 	Click the icon and choose a shape among the selection tree. See also topic Selection of Shape.
HSS t (design)	If a HSS shape has been selected, please specify the design thickness that will be considered during design or verification: 1.0t (ICCA/Can), 0.9t (ICCA/US) or 0.93t (AISC/US).
Material	Select a material among the selection tree.
Dead load	Select the dead load case that automatically considers the member self-weight in generated load combinations.
2L Distance or b1	2L: Gap between back-to-back angles b1: Dimension of the flat side of a V-section.
Area	Area of section.

Field	Description
Linear mass	Linear mass of this member.
Activate design criteria	Check this box to activate design criteria. The **Steel Design** tab, **Concrete Design** tab or **Timber Design** tab will appear in this dialog box according to the chosen shape material.
Usage	Specify the member usage, if required. Refer to topic Member Usage.
Composition	Select the shape composition in the drop-down list box: Standard, Composite Beam, or Filled HSS. The **Composite Beam** tab or **Filled HSS** tab will appear in the dialog box. Select the tab and complete it.
Behaviour	Specify the member behaviour in the drop-down list box: Standard, Elastoplastic, Off-load or Linear only. If the elastoplastic behaviour is selected, the **Behaviour** tab will appear in the dialog box. Select this tab and complete it.
Effective Stiffness	Refer to Effective Stiffness
Ix'/Ix	For a cracked section, specify the effective inertia of this member, on its strong axis, as specified by code. Otherwise, enter a value of 1.0 (100% effective).
Iy'/Iy	For a cracked section, specify the effective inertia of this member, on its weak axis, as specified by code. Otherwise, enter a value of 1.0 (100% effective).
J'/J	For cracked section, enter the effective torsion and shear stiffness of the section. Otherwise, enter a value of 1.0 (100% effective).
A'/A	Specify the effective axial stiffness of this member. Otherwise, enter a value of 1.0 (100% effective).
Effective at stage	Select the construction stage where the composite member will be effective. Refer to Construction stages.
Moving load analysis	
Moving load axis	If the member is part of a moving load axis, select it in the list box.
2D axle factors	If you modeled in 2D, select the axle factors to be assigned to this member.

Rigid Extensions and Member Eccentricities

Alignments ey & ex

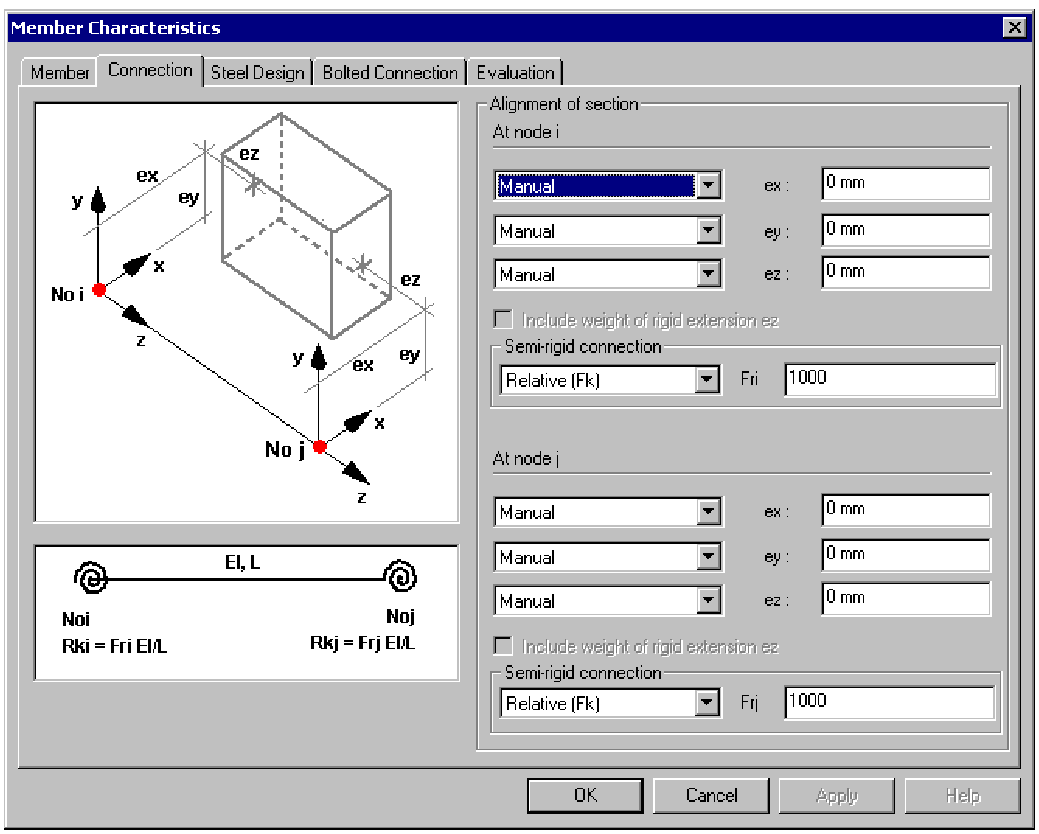
The section may be positioned vertically, longitudinally and transversely to the axis passing through the member end nodes. Make sure that eccentricities ex and ey are the same for a given member.

Rigid Extensions ez

All concrete members and prestressed concrete members must have rigid extensions because VisualDesign needs them to calculate the required rebars development lengths. Rigid extensions are specified in the **Connection** tab of **Member Characteristics** dialog box.

VisualDesign automatically calculates rigid extensions when the function **Automatic Calculation of Rigid Extensions** is called up (available in **Structure / Tools**).

The Connection tab



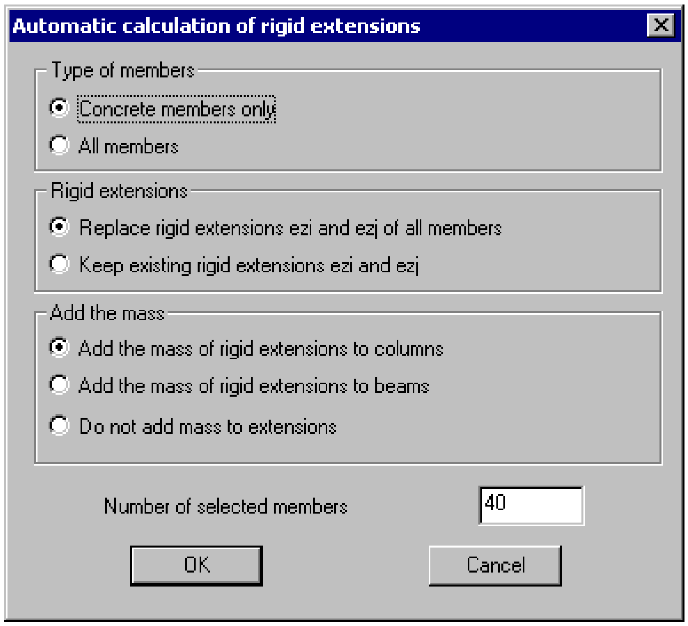
The table below shows the description for each topic included in the **Connection** tab.

Field	Description
Alignment of section	
Alignment x-direction	Choose an alignment from the drop-down list box: "Manual", "Left" or "Right". Example: By choosing "Left", the section is positioned left to the node and the eccentricity value "ex" is automatically calculated and written in this field. This value is negative and equal to half of the section width.
ex	If you choose "Manual", the value of eccentricity "ex" will be equal to zero. The section will be centred on the node, in the x direction.
Alignment y-direction	Choose an alignment from the drop-down list box: "Manual", "Above" or "Below". Example: By choosing "Above", the section is positioned above the node and the eccentricity value "ey" is automatically calculated and written in this field. This value is positive and equal to half of the section height.
ey	If you choose "Manual", the value of eccentricity "ey" will be equal to zero. The section will be centred on the node, in the y direction.
Alignment z-direction	Specify the length of rigid extension in the drop-down list box "Manual" or select option "Free edge". If you choose "Free edge", the end of the section will be positioned at the face of the support. For example, the end of a beam will be positioned at the face of the column.
ez	If you choose "Manual", the value for rigid extension "ez" will be equal to zero. The end of the section will be positioned on the node, meaning that there is no rigid extension. If there is no transverse element, enter the length of rigid extension.
Include weight of rigid extensions	Check the box to include the mass of the rigid extension "ez" at node i and j, for columns OR beams.
Semi-rigid connection	Refer to topic "Semi-rigid Connections".

Automatic Calculation of Rigid Extensions

The tool **Automatic Calculation of Rigid Extensions**, located in the **Structure/Tools** menu, calls up a dialog box that will help you model steel or concrete member rigid extensions by calculating them automatically.

This functionality will automatically create rigid extensions at the face of each support, for concrete members only or for all members of your structure. It can also replace the already defined rigid extensions or keep the old ones. More, you can choose to automatically add these weights to columns or to beams or not considering any rigid extension weight in your project.



See also

The Connection tab

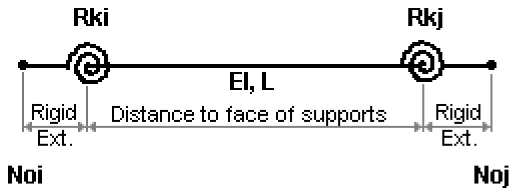
Rigid Extensions

Modeling Valid Rigid Extensions

Semi-Rigid Connections

Semi-rigid connections (or partial connections) can be modeled at member ends i and j through the **Connection** tab (**Member Characteristics** dialog box). When the member has rigid extensions, partial connections will be located at the interior ends of rigid extensions, as shown below with spirals. Therefore, it is always located at the face of support (beam or column).

Considering a beam of length L, of stiffness EI/L, with rigid extensions, and semi-rigid connections of stiffness equal to Rki and Rkj, we will obtain this model:



Where

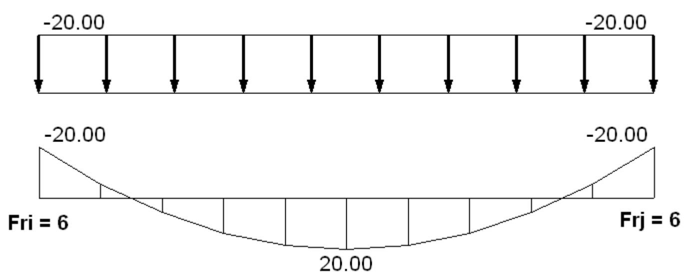
$Rk_i = Fr_i \; EI/L$

$Rk_j = Fr_j \; EI/L$

WARNING!

To model such partial connections, appropriate stiffness factors must be specified. Before using this option, read the theory about semi-rigid frames.

This example shows a beam of 4m long with fixed ends, with a uniform load of -20kN/m. If stiffness factors are equal to 6 (Fri = Frj = 6), the positive bending moment and negative bending moments will be equal.



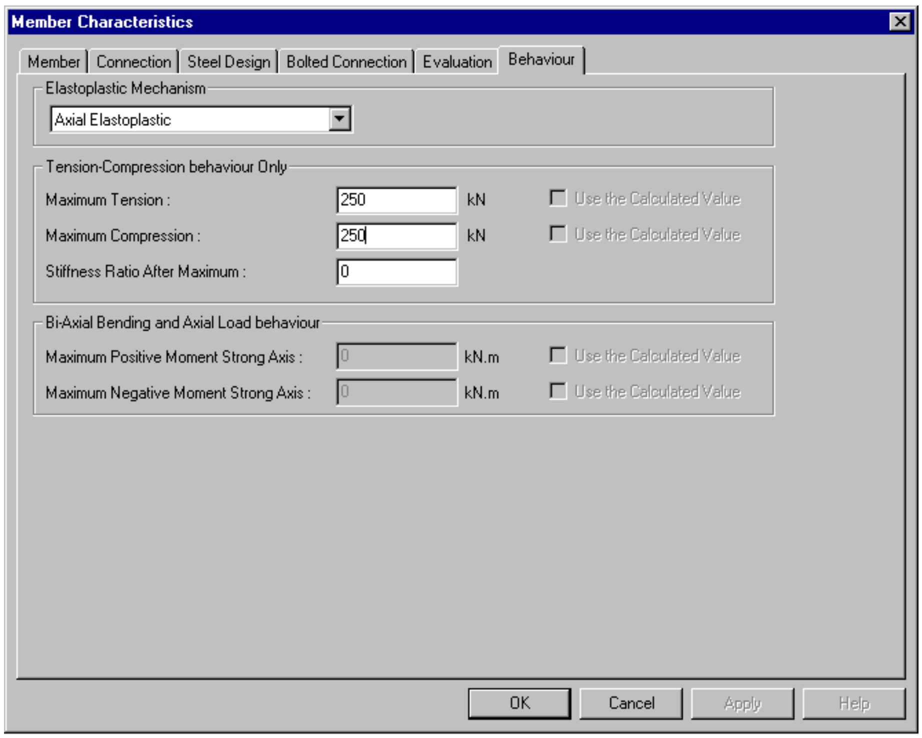
Note:

The calculated value of Rki or Rkj depends on the length of the member. If you split up a member, the partial rigidity will be adjusted to account for the partial rigidities that have been defined at the ends of original member.

If you split up the beam showed in the above example in two segments of 2m each, the factor Fri (beam at the left) and the factor Frj (beam at the right) will be adjusted to 6 * 2/4 = 3 because the stiffness of the two beams, Rki and Rkj, must the same as before the splitting.

The Behaviour tab

You have access to the **Member** dialog box by double-clicking on a member while in the "Structure" mode. To define a member as elastoplastic, choose option *Elastoplastic* in the "Behaviour" field. Then, select the **Behaviour** tab and complete the information.



See the table below for definition of headings included in the **Behaviour** tab.

Field	Description
Elastoplastic mechanism	Choose an *Axial Elastoplastic* or *Elastic* behaviour in the drop-down list box.
Tension-Compression behaviour only	Only for an axial elastoplastic analysis using friction dampers (PALL). See Non-linear Time History Analysis.
Maximum Tension	Specify the allowed maximum tension in member or check the **Use the Calculated Value** box to use the calculated tension capacity of the member.
Maximum Compression	Specify the allowed maximum compression in the member or check the **Used Calculated Value** box to use the calculated compression capacity of the member.

Field	Description
Stiffness Ratio After Maximum	Specify the stiffness ratio after it reached plasticity.

Bi-axial Bending and Axial Load Behaviour

Maximum Positive Moment Strong Axis	Specify the allowed maximum positive moment relative to the strong axis of the member or check the **Use Calculated Value** box to use the calculated resisting moment of the member.
Maximum Negative Moment Strong Axis	Specify the allowed maximum negative moment relative to the strong axis of the member or check the **Use Calculated Value** box to use the calculated resisting moment of the member.

See also

Member Dialog Box

Non-linear Time History Analysis with friction dampers

Members Spreadsheet

Members Spreadsheet

Access this multi-spreadsheet by selecting **Members** under **Structure** Menu. The first spreadsheet, which is "Members", is the main one because it contains basic characteristics about the member connections, composition, and behaviour and this is where you activate the member design criteria (concrete, steel or wood).

If you own all other modules, you will find the following tabs in the spreadsheets: Composite Beams, Concrete Design, Steel Design, Timber Design, Bolted Connections and Evaluation

Members spreadsheet (Master)

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Node i Number	Node number at origin of member	No
Bending Mx	Member end conditions in the plane of the strong axis	Single click
Bending My	Member end conditions in the plane of the weak axis	Single click
Torsion Mz	Member end conditions for torsion.	Single click
Axial Fz	Member axial relaxation	Single click
Node j Number	Node number at the end of member	No
Shape	12 alphanumeric characters describing the member shape. Expand the Shape *tree* composed of all the available shapes and choose one by activating a radio button.	Double-click
Material	Choose the member material in the Material selection *tree*.	Double-click
HSS t (design)	If a HSS shape was selected, indicate the design thickness that will be considered during the design or verification, in the list box.	Double-click

Column	Description	Editing
Usage	Select the member usage among the drop-down list box. See topic Member Usage.	Double-click
Gap between Angles or b1	2L section: Gap between two angles. b1: dimension of the V section part that is not bend.	Single click
Local axes	Indicate if the local axes system is oriented according to an orthogonal or major-minor axes system.	Double-click
β angle	Rotation angle	Single click
Length	Member length	No
Pre-tension	Initial pre-tension in the member (positive in tension). N.B. This pre-tension will be applied to all load combinations.	Single click
Behaviour	Choose the member behaviour: Standard, Elastoplastic, Off-load or Linear only.	Double-click
Composition	Choose the member composition: Standard, Composite Beam or Filled HSS.	Double-click
Design Code	Choose a design code: Steel or Reinforced concrete, (Timber and Aluminium are not yet available).	Double-click
Effective Inertia – Bending strong axis	For a cracked section, specify the effective inertia of this member, on its strong axis, as specified by code. Otherwise, enter a value of 1.0.	Single click
Effective Inertia – Bending weak axis	For a cracked section, specify the effective inertia of this member, on its weak axis, as specified by code. Otherwise, enter a value of 1.0.	Single click
Effective Stiffness for Torsion/Shear	For cracked section, enter a ratio to reduce the torsional and shear stiffness of the section. Otherwise, enter a value of 1.0.	Single click
Effective Inertia – Axial	Enter the ratio to reduce the axial stiffness due to cracked section. Otherwise, enter a value of 1.0.	Single click
Moving Load Axis	If the member is located on a moving load axis, select the one among the list box.	Double-click
2D Axle Factor	Select the axle factor that will be applied to this member for moving load analysis.	Double-click

Connection Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Alignment exi	Choose the type of alignment: "Manual", "Left" or "Right" of node i.	Double-click
exi	Enter eccentricity exi	Single click
Alignment eyi	Choose the type of alignment: "Manual", "Above" or "Below" node i.	Double-click
eyi	Enter eccentricity eyi	Single click
Rigid Extension ezi	Choose option *Manual* and enter the rigid extension length at node i or choose option *Free Face*. The latter will create a rigid extension from node to free face of column or beam.	Double-click
ezi	If you selected option *Manual*, enter rigid extension length ezi (according to member local axes, negative or positive).	Single click
Alignment exj	Choose a type of alignment: "Manual", "Left" or "Right" at node j.	Double-click
exj	Enter eccentricity exj	Single click
Alignment eyj	Choose a type of alignment: "Manual", "Above" or "Below" at node j.	Double-click
eyj	Enter eccentricity eyj	Single click
Rigid Extension ezj	Choose option *Manual* and enter the rigid extension length at node j or choose option *Free Face*. The latter will create a rigid extension from node to free face of column or beam.	Double-click
ezj	If you selected option *Manual*, enter rigid extension length ezj (according to member local axes, negative or positive).	Single click
Stiffness Node i	Specify the stiffness of connection at node i: Absolute (Rk), Relative (Fk) or According to connection.	Double-click

Column	Description	Editing
Rki	Factor that represents the absolute stiffness of connection at node i. The default value of 1 kN.m/rad is corresponding to an absolute stiffness.	Single click
Fki	Factor that represents the relative stiffness of connection at node i. The default value is 1000 meaning that the connection is 100% rigid.	Single click
Stiffness Node j	Specify the stiffness of connection at node j: Absolute (Rk), Relative (Fk) or According to connection.	Double-click
Rkj	Factor that represents the absolute stiffness of connection at node j. The default value of 1 kN.m/rad is corresponding to an absolute stiffness.	Single click
Fkj	Factor that represents the relative stiffness of connection at node j. The default value is 1000 meaning that the connection is 100% rigid.	Single click
+Mass Noi	If "ezi" is different from 0, include the weight of the rigid extension at node i by activating this option [x]	Double-click or Space bar
+Mass Noj	If "ezj" is different from 0, include the weight of the rigid extension at node j by activating this option [x]	Double-click or Space bar

Composite Beam Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	No
Composition	Shaded field indicating that this member is a composite beam.	No
Composite Slab	Choose a composite slab previously defined in the Composite Slabs spreadsheet.	Double-click
Stud	Choose a type of stud (previously defined in the Studs spreadsheet).	Double-click
Effective b	Effective width of composite slab.	Single click

Column	Description	Editing
Actual b	Real width of composite slab.	Single click
%Qr	Fraction of shear connection (ex: 100%, 70%, 40%) of studs.	Single click
Nos. Studs / Row	Specify the number of studs per row for the design or verification of fatigue.	Single click
Dead Load Slab	Choose option [x] to add the slab dead load to the structure dead load. *Do not activate this option for a project with construction stages.*	Double-click or Space bar
Composite Properties	Consider the composite section properties for analysis by choosing option [x].	Double-click or Space bar
With Reinf. If Mfx -	Consider the steel reinforcement in the slab located at negative moments by choosing option [x]. As a result, the position of neutral axis will be modified.	Double-click or Space bar
With Reinf. If Mfx +	Consider the steel reinforcement in the slab located at positive moments by choosing option [x]. As a result, the position of neutral axis will be modified.	Double-click or Space bar
Composite Beam with construction stages only		
Solid Concrete at Stage	Composite effects will be effective at this construction stage.	Single click
Composite Beam with or without construction stages		
Face Number	Choose the face on the beam where stresses will be calculated.	Double-click
Position x	Enter the position of this point on the beam according to the figure below.	Single click
Linear Mass	Linear mass of transformed section.	No

Composite Beams Spreadsheets - Short-term and Long-term

These two spreadsheets include the section-transformed properties considering a ratio "n" (= Es/Ec) respectively for short-term and long-term, as specified in the **Composite Beam** tab of **Project Configuration**.

The default values of "n" for short-term and long-term deformations are respectively of 1.0 and 3.0. These values will be used for those not owning the *Steel Design* module.

Users owning the *Steel Design* module can modify the default values for "n" in the **Composite Beam** tab of **Project Configuration**.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	No
Composition	Shaded field indicating that this member is a composite beam.	No
Neutral Axis	Position of neutral axis in the composite section.	No
yt	Distance from neutral axis to top fibre.	No
yb	Distance from neutral axis to bottom fibre.	No
Es/Ec	Ratio n: Modulus of elasticity of steel divided by modulus of elasticity of concrete	No
Ix	Inertia of transformed section, on strong axis.	No
J	Torsional constant of the transformed section.	No
Area	Area of the transformed section.	No
Sx (ct)	Elastic section modulus at the top of concrete slab, on strong axis.	No
Sx (cb)	Elastic section modulus at the bottom of concrete slab, on strong axis.	No
Sx (st)	Elastic section modulus at the top of steel shape, on strong axis.	No
Sx (sb)	Elastic section modulus at the bottom of steel shape, on strong axis.	No

Behaviour Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Behaviour	Choose an axial elastoplastic or elastic behaviour in the drop-down list box.	Double-click

Maximum Tension	Specify the allowed maximum tension in member.	Single click
Maximum Compression	Specify the allowed maximum compression in the member.	Single click
Maximum Stiffness Ratio	Specify the stiffness ratio after it reached plasticity.	Single click

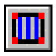
Bill of Materials (Members)

Group: Structural data

Column	Description	Editing
Material	The member material	No
Section	Name of the section	No
Length	Total length of all members having the same section	No
Weight	Weight of all members having the same section	No
Surface	Surface of all members having the same section	No
Volume	Volume of concrete members	No

Floors - General

The Floor Element



The "Floor" icon of Elements Toolbar

VisualDesign does not design or verify floors and joists. The **Composite Slabs**, **Steel decks** and **Studs** spreadsheets are used to model steel/concrete composite beams.

VisualDesign takes the loads applied to surfaces (floors or joist) and transfers them to the beams surrounding the floor area according to the type of floor (Two-Way, One-Way or Joist).

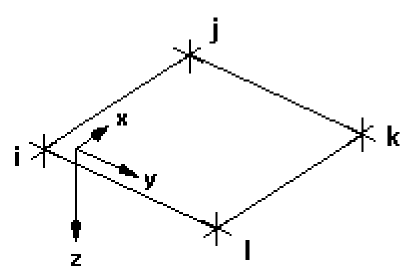
Restrictions

- The nodes that delimit the floor element must be located in the same plan.
- The floor might be an asymmetrical quadrilateral.
- The distance between two corners may be covered par many beams.

If you want to obtain information (forces, bending moments…) about the transfer of concrete floor bending moments to adjacent beams, model the floor with plates and run a finite element analysis.

It is possible to view the floor local axes system by activating this option from the "Attributes" tab of the **View Options** dialog box. The local axes are then displayed inside the floor, near node i.

Floor Local Axes System



See also

Floor Characteristics

Types of Floors

The Diaphragm Effect

Floor spreadsheet

Slabs Spreadsheet
Steel Decks Spreadsheet
Activate Element
Displaying Floor Characteristics

Rigid Diaphragm for Floors

A diaphragm action can be specified in the Floor Characteristics dialog box and in the corresponding spreadsheet.

RECOMMENDATIONS

Be careful when using this option. If girders are composed of a truss, never activate the diaphragm effect. The diaphragm action will create a composite effect between the chords and the floor because the centre of gravity of the truss is too low compared to the floor. Thus, this system will be too rigid. If girders are composed of usual I-beams, this effect is less critical.

BRIDGE DESIGN:

Rigid floors should not be used to model a bridge, for the same reason explained above. Furthermore, the diaphragm effect will act opposite to temperature effects and finally, prestressing effects will be absorbed within the diaphragm.

CONCRETE FLOORS

You can use this option for concrete floors.

STEEL DECK AND CONCRETE SLAB

You can use this option for storeys and roof.

SEMI-RIGID DIAPHRAGM

If the roof is composed of joists and steel decks, it is recommended to create a semi-rigid diaphragm with thin triangular plates. These plates should be modeled using all nodes located on this level.

VERTICAL FLOORS

Don't activate this option if you modeled vertical floors to distribute loads!

TRUSSES

Be careful when using this option. If girders are composed of trusses, the diaphragm action will create a composite effect between top chords and floors because the centre of gravity of the truss is too low compared to the floor. Thus, this system will be too rigid. If girders are composed of usual I-beams, this effect is less critical.

Joist Floor

When creating a joist floor, first make sure that you have clearly defined the members on the entire periphery of the floor. If you have missed one side of the floor, you will get a warning message indicating that there is one adjacent beam missing for the floor.

The space between joists (delta) proposed by default is 600 mm. The quantities of joists, the first and last spaces between joists are calculated according to the specified direction and the delta. However, you can decide to enter a different quantity of joists, or specify a different space value (spacing, first or last), and other boxes will then be interactively calculated according to joist orientation.

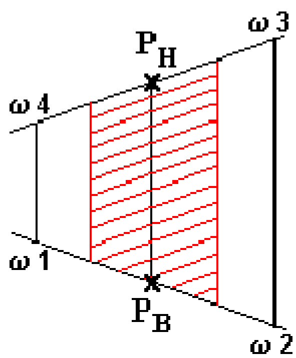
See also

Floor Characteristics

Load Transfer for Joist Floors

Loads Transfer for Joist Floor

Distributed loads are transferred to the joists according to their corresponding tributary surfaces, assuming a zero half-span shear. Consequently, joists loads are transferred along adjacent beams according to the user's geometric selection.



See also

Floor Characteristics

Two-Way Slab

This type of floor transfers concentrated loads on adjacent beams. Beams must surround the floor outline. If you have missed one side of the floor, you will get a warning message at the beginning of the analysis.

See also

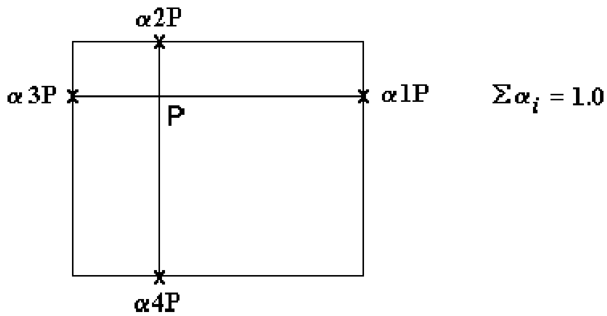
Floor Characteristics

Creating a Two-Way Slab

Load Transfer for Tw0-Way Slabs

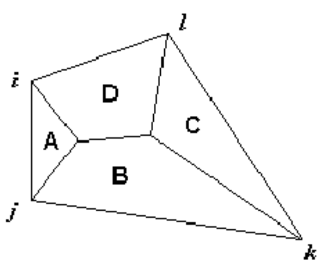
Loads Transfer for Two-Way Slabs

A concentrated load that is located on a two-way slab is distributed along the four sides of the floor without diffusion of load. Its distribution is carried out according to the floor uniform stiffness.



As for distributed loads, they must be applied on the entire surface. The distribution is carried out according to the distribution lines indicated in the figure below.

The sum of the loads is respected, although the distribution is approximated.



See also

Floor Characteristics

Creating a Two-Way Slab

Creating a Two-Way Slab

To create a Two-Way slab, select the *Two-Way Slab* option in the **Floor Characteristics** dialog box.

See also

Floor Characteristics

One-Way Slab

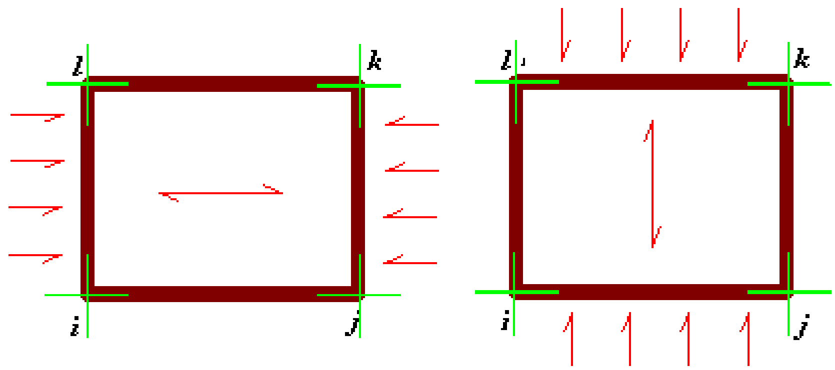
This type of floor allows the transfer of loads on two opposite sides, according to the user's choice. These sides are determined by the slab direction, as specified in **Floor Characteristics** dialog box. If you forgot to model a beam supporting this type of floor, a warning message will appear at the beginning of the analysis.

See also

Floor Characteristics
Creating a One-Way Slab
Load Transfer for One-Way Slab

Loads Transfer for One-Way Slab

Concentrated loads are distributed with diffusion along the appropriate sides, according to the simply supported beam approach.



Distributed loads are transferred along the appropriate sides according to the slab orientation and according to the slab stability when simply supported.

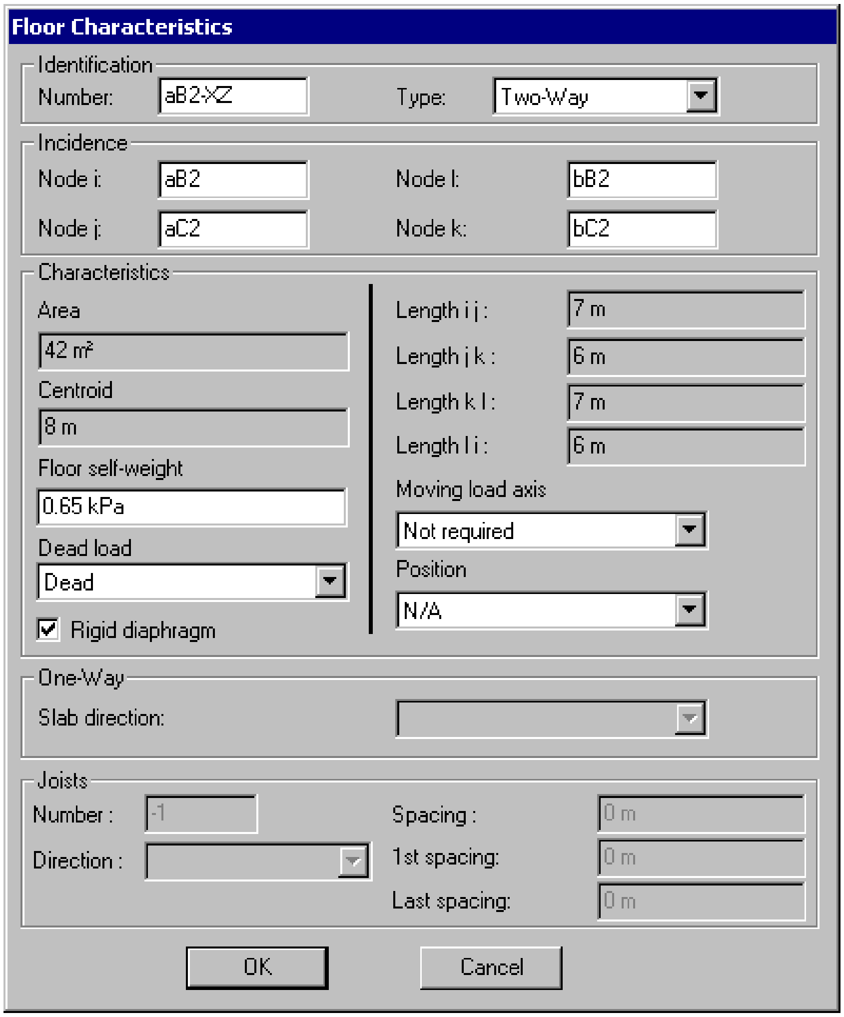
Creating a One-Way Slab

To create a one-way slab, select the *One-way slab* option as type of floor in **Floor Characteristics** dialog box. Then select slab direction (jk and li or ij and kl).

Floor Characteristics Dialog Box

Floor Characteristics Dialog Box

While in the "Structure" mode, you have access to the **Floor Characteristics** dialog box by double-clicking on a floor or by selecting many and pressing the **Properties** icon.



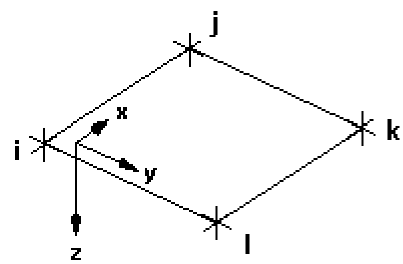
If the floor self-weight is specified in this dialog box, VisualDesign will integrate it in the dead load case that will be selected here. To learn more about it, please refer to the **Dead load** tab of **Loads Definition** dialog box.

See the table below to know the definition of headings composing the dialog box.

Field	Description
Identification	
Number	Floor number
Type	Select the type of floor in the drop-down list box: Two Way slab, One-Way slab or Joists. If this slab is a one-way slab, specify the bearing direction in the field "Slab direction".
Incidence	Number of nodes i, j, k, and l located at the floor corners.
Characteristics	
Area	Calculated area of the floor.
Centroid	Calculated centroid of this floor, relatively to zero point.
Floor self-weight	Specify the floor dead load here or use the Load Definition dialog box in order to apply the floor dead load graphically as other type of loads.
Dead load	Select the type of dead load that will consider the floor self-weight.
Rigid diaphragm	Check this box if you want the floor to behave as a rigid diaphragm. Refer to Diaphragm Effect.
Length ij, jk, kl, li	Calculated lengths between nodes that are composing this floor.
Moving Load Axis	
Axis	If a moving load axis runs on one side of the floor, specify which one.
Position	Specify the side where is located the moving load axis.
One-Way	
Slab direction	Specify the bearing direction for this one-way slab.
Joist	
Number	Joist floor number
Direction	Select the direction of the joists
Spacing	Specify the distance between joists

Field	Description
First Spacing	Distance to first joist.
Last Spacing	Distance between last joist and spandrel beam.

Floor Local Axes System



Floors Spreadsheet

Floors Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the floor.	Single click
Node i number	Node number at corner i	No
Node j number	Node number at corner j	No
Node k number	Node number at corner k	No
Node l number	Node number at corner l	No
Type	Type of floor: two way slab, one way slab, or joist	Double-click
Self-weight	Self-weight of this floor.	Single click
Dead load	Dead load case that will integrate the floor self-weight.	Double-click
Lij	Length between nodes i and j	No
Ljk	Length between nodes j and k	No
Lkl	Length between nodes k and l	No
Lli	Length between nodes l and i	No
Area	Area of the floor	No
Diaphragm action	Choose option [x] if the floor acts as a diaphragm.	Double-click or Space bar
Slab Direction	Direction of the one-way slab (supported at jk and li, supported at ij and kl, n/a)	Double-click
Joists Direction	Choose the direction of beams, joists	Double-click
Number of Joists	Validated upon exiting spreadsheet.	No
Delta	Regular spacing between joists	Single click
First Spacing	Validated when exiting the spreadsheet	No
Last Spacing	Validated when exiting the spreadsheet	No

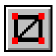
Column	Description	Editing
Centroid	Calculated centroid of this floor relatively to zero point.	No
Moving load axis	Select the moving load axis number, if applicable.	Double-click
Location	Select the side of the floor that is part of the moving load axis or choose N/A.	Double-click
Moving Load Axis	Select the moving load axis number.	Double-click

See also

Types of Steel decks
Slabs spreadsheet

Plates - General

Rectangular and Triangular Plates



The "Plates" icon of Elements toolbar

Rectangular (24 DOF) and triangular plates (18 DOF) are available in VisualDesign™. This icon activates both types. The software uses an isoparametric formulation that includes a component for in-plane translation stiffness and a component for rotation stiffness in the normal direction of the plane element.

For finite element analyses, it is recommended to create groups of plates for surfaces and shear walls. A common local axis system is assigned to each group of plates. Therefore, the interpretation of results is much easier.

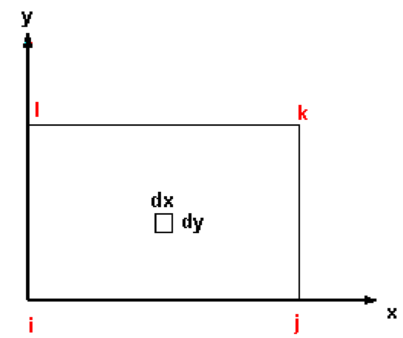
To split up plates, use function **Multiple Split**. If the plates are already loaded, you can split up plates along with their loads.

See also

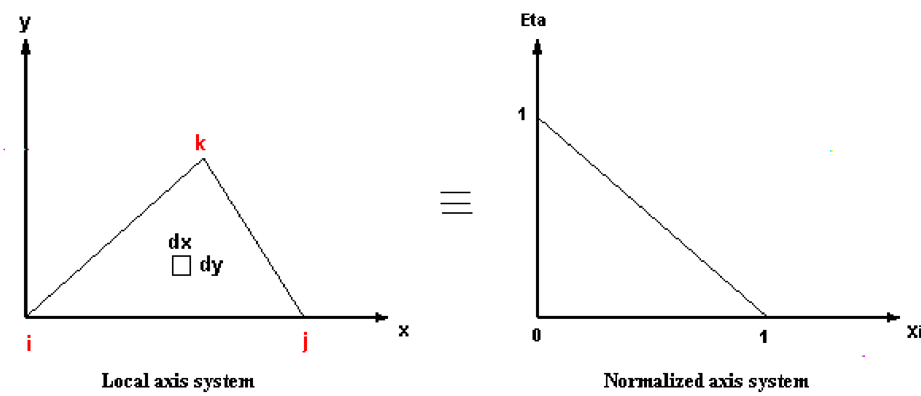
The Plate Characteristics dialog box
Splitting Plates Into Multiple Parts
The Plates Spreadsheet
Nodes Numbering Convention for Plates
Plane Stresses Convention
Principal Stresses Convention
Interpreting Results
Groups of plates - Surfaces
Groups of plates - Shear wall

Numbering Convention for Plates

RECTANGULAR PLATES



TRIANGULAR PLATES



See also

Plane Stresses Convention

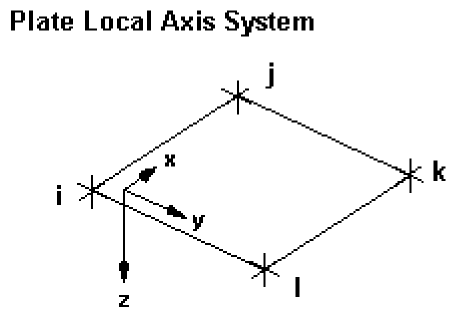
Principal Stresses Convention

Interpreting Results

Displaying the Plate Local Axis System

- Go to the **Attributes** tab of **View Options** dialog box and activate the option *Local Axis system* in the Plates section.

Point (0,0,0) corresponds to the plate's Node *i*.

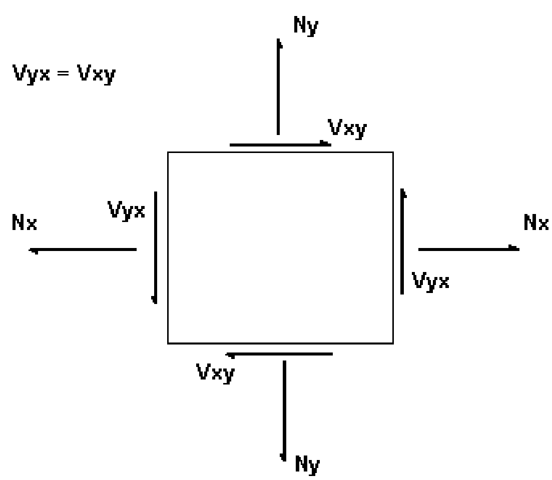


See also

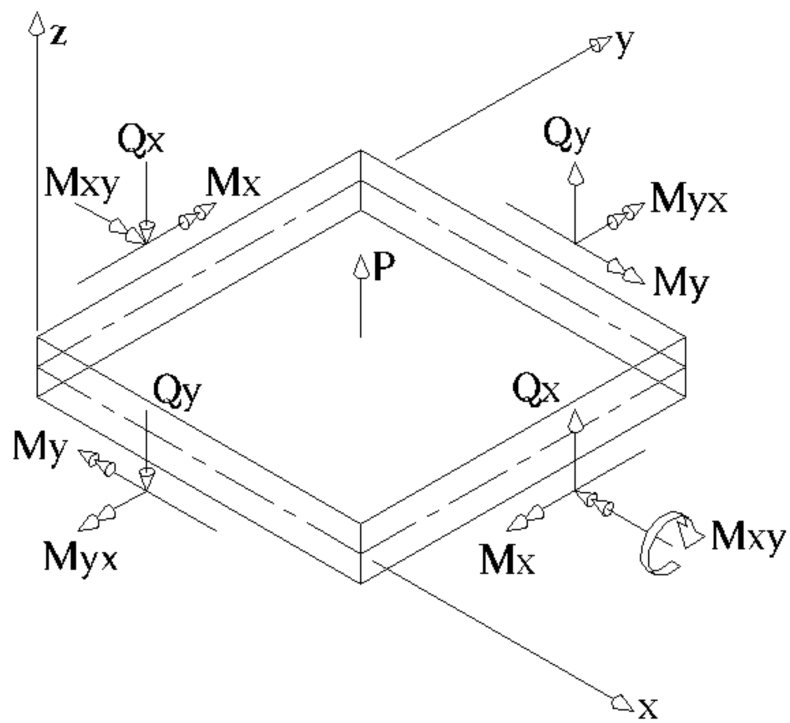
The Attributes Tab

Display Plate Characteristics

Convention for Plane Stresses



THE SIGN CONVENTION IS ILLUSTRATED BELLOW:

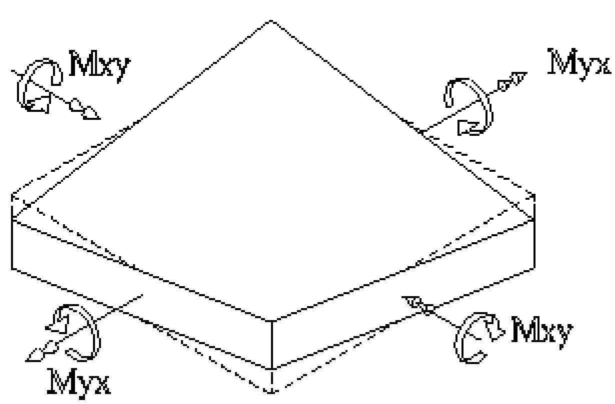


CALCULATION OF QX AND QY

$$Qx = \frac{\partial Mx}{\partial x} + \frac{\partial Mxy}{\partial y}$$

$$Qy = -\frac{\partial My}{\partial y} - \frac{\partial Mxy}{\partial x}$$

Myx and Mxy bending moments are:



The dotted line shows the original position of the plate.

See also

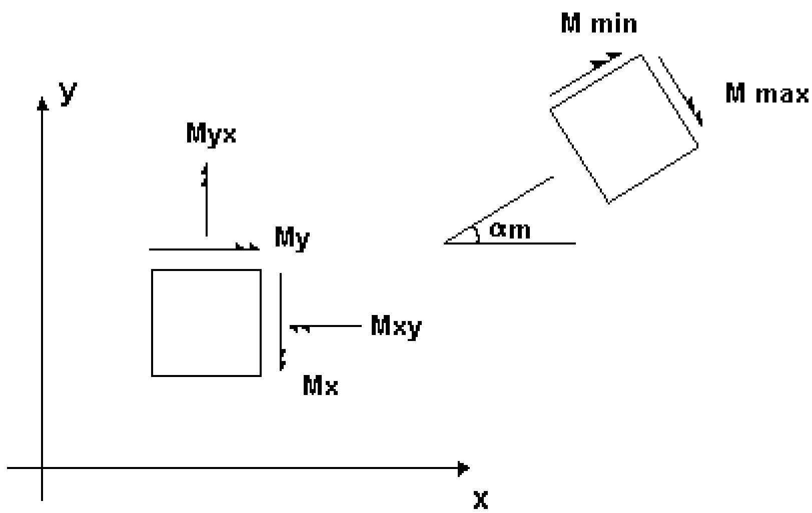
Numbering Convention

Principal Stresses Convention

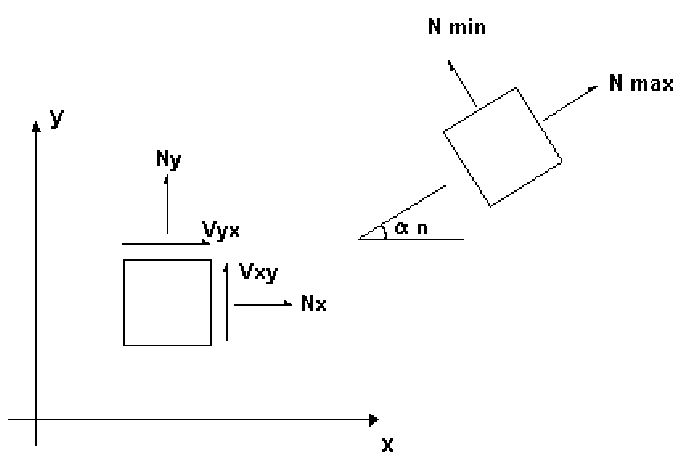
Interpreting Results

Convention for Principal Stresses

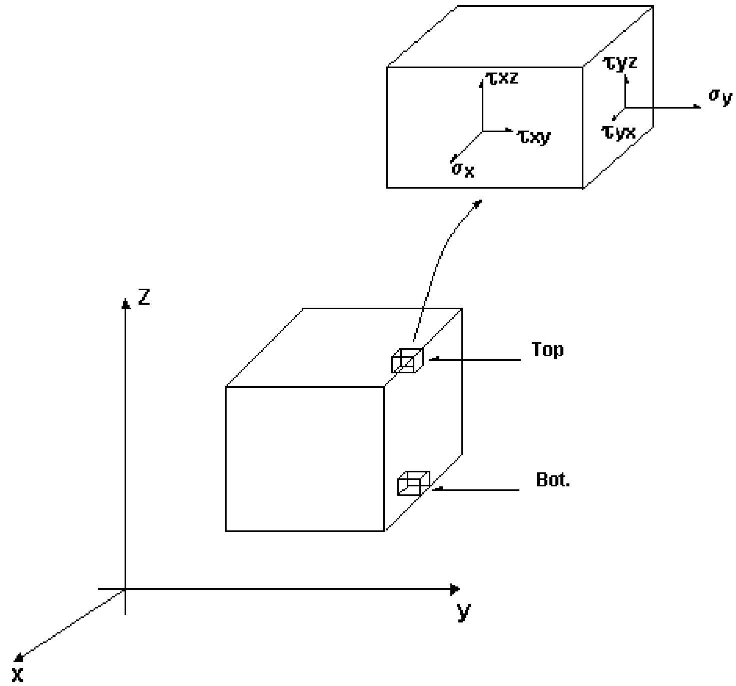
Convention for bending moments



Convention for axial stresses



Convention for principal strains



See also

Numbering Convention

Plane Stress Convention

Interpreting Results

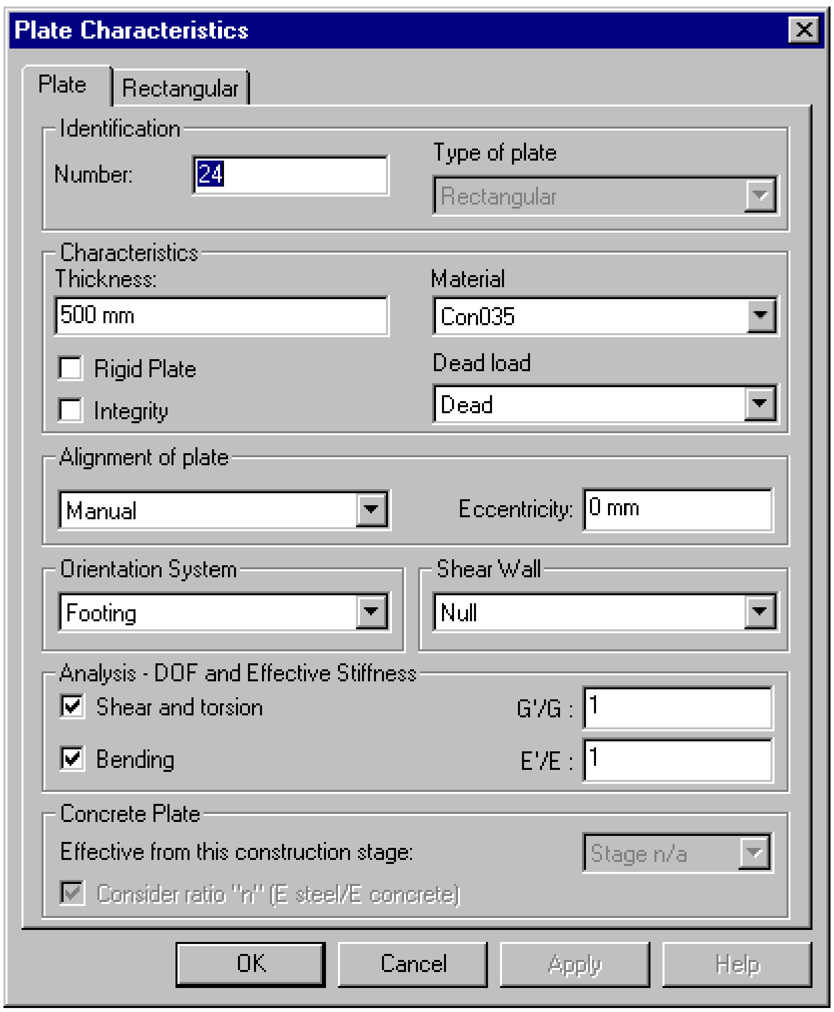
Plate Characteristics Dialog Box

Plate Characteristics Dialog Box

While in the "Structure" mode, you have access to the **Plate Characteristics** dialog box by double-clicking on a rectangular or triangular plate or by selecting any plates and choosing the **Properties** function of **Edit** menu.

The **Plate Characteristics** dialog box can be composed of two or three tabs: The **Plate** tab, which is the main one, the **Rectangular** and **Triangular** tabs according to the selected type of plates.

The General tab



See the table below to know the definition of headings composing this tab:

Field	Description
Identification	
Number	Number of the plate.
Type of plate	Shaded field that informs the user on the plate geometry: Rectangular or Triangular.
Characteristics	
Thickness	Enter the thickness of the plate.
Material	Select the plate material in the drop-down list box.
Dead load	Select the type of dead load that corresponds to the plate self-weight.
Rigid Plate	Activate this box if the plate is very rigid (100x). **Warning!** Use this option if the plate cannot deform such as a plate located below a concrete column or above.
Integrity	Activate this option if this rigid plate must include integrity reinforcement and VisualDesign will place extra reinforcement. This type of plate could be located above a concrete column..
Alignment of plate	Select an alignment in the list box: "Manual", "Above" or "Below". If a manual one is selected, the default eccentricity is 0.0, meaning that the centreline of the plate is in line with end nodes.
Eccentricity	If you selected a manual alignment, enter the eccentricity relative to the centreline of the plate and according to local axis system.
Orientation System	Select the group that orients this plate if it is part of a surface. Refer to Groups of Plates - Surfaces.
Shear Wall	Select the group that orients this plate if it is part of a shear wall. Refer to Groups of Plates - Shear walls.
Analysis – DOF	
Shear and Torsion	Check this box to consider the shear and torsion of the plate in the analysis.
G'/G	Specify the ratio (< 1.0 or <100%) for effective stiffness in shear and torsion.
Bending	Check this box to consider the bending of the plate in the analysis.

Field	Description
E'/E	Specify the ratio (< 1.0 or <100%) for effective inertia in bending.
Concrete Plate	
Effective from Stage:	Indicate from which construction stage this plate will be considered effective (solid) for finite element analysis of concrete plates.
Consider Ratio "n"	Check this box to consider ratio "n" (E steel/E concrete), as defined in the **Composite Beam** tab of **Project Configuration**. If this option is not activated, ratio "n" will be equal to 1 for long-term and short-term effects.

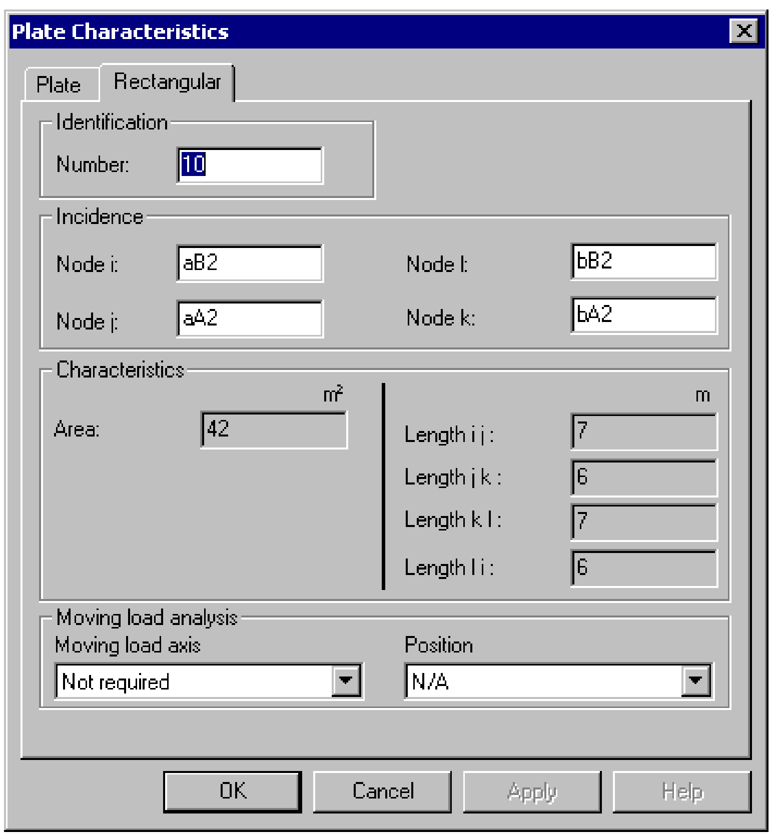
See also

Interpreting plates analysis results

Composite Beam tab (Project Configuration)

The Rectangular tab

In the "Structure" activation mode, you have access to the **Rectangular** tab of **Plate Characteristics** dialog box by double-clicking on a rectangular plate or by selecting many and choosing the **Properties** function of **Edit** menu.

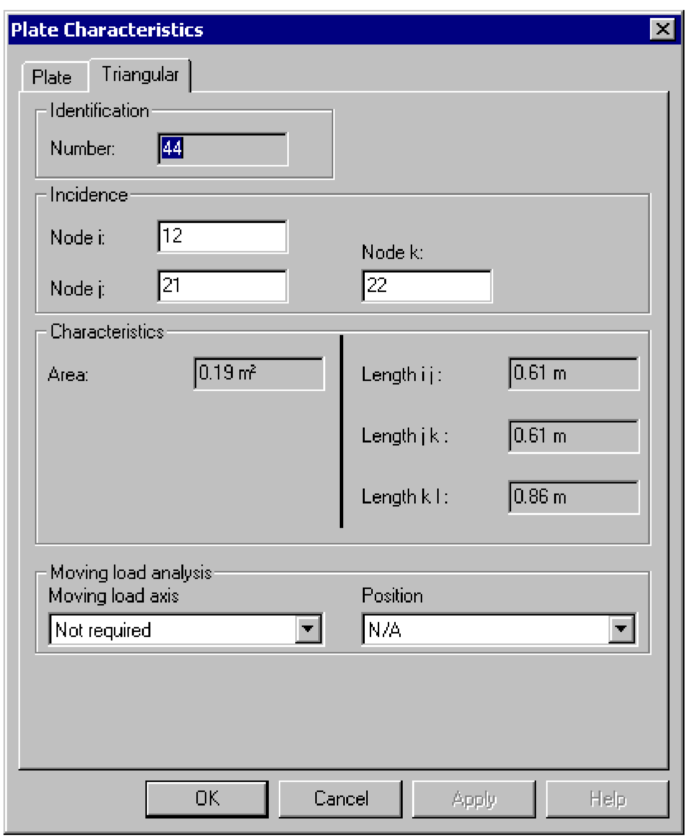


See the table below to know the definition of the **Rectangular** tab.

Heading	Description
Identification	Plate ID number.
Incidence	Nodes i, j, k and l numbers.
Characteristics	
Area	Calculated plate area.
Length ij, jk, kl and li	Calculated lengths of each side of the plate.
Moving Load Axis	
Axis	If a moving load axis runs on one side of the plate, specify the one.
Position	Specify the side of the plate where runs the moving load axis.

The Triangular tab

While in the "Structure" mode, you have access to the **Triangular** tab of **Plate Characteristics** dialog box by double-clicking on a triangular plate or by selecting many and choosing the **Properties** function of **Edit** menu.



See the table below to know the definition of headings composing the **Triangular** tab.

Heading	Description
Identification	Plate ID number.
Incidence	Node i, j, and k numbers.
Characteristics	
Area	Calculated plate area.
Length ij, jk, ki	Calculated lengths of each side of the plate.
Moving Load Axis	
Moving Load Axis	If a moving load axis runs on one side of the plate, specify which one.
Position	Specify on which side of the plate runs the moving load axis.

See also

Interpreting plates analysis results

Triangular Plates spreadsheet

Composite Beam tab (Project Configuration)

The Plates Spreadsheet

The Plates Spreadsheet

While in the Structure activation mode, go to menu **Structure** and select **Plates**. This spreadsheet is a multi spreadsheet that includes general data about plates and specific data about triangular and rectangular plates (the **Triangular** tab and **Rectangular** tab). The first page (the **Plate** tab) is the master one. Use this page to add or delete plates. Sorting can be done in either pages and it is saved when switching to one page to another.

Here is a description of fields included in the master page:

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Geometry	Rectangular or Triangular plate	Double-click
Orientation System	Group of this oriented plate that is part of a surface. Refer to Groups of Plates - Surfaces.	Double-click
Shear Wall	Group of this oriented plate that is part of a shear wall. Refer to Groups of Plates - Shear walls.	Double-click
Thickness	Plate thickness	Single click
Material	Plate material	Double-click
Dead load	The type of dead load that represents the plate self-weight.	Double-click
Rigid Plate	Activate option [x] to consider the plate as very rigid, such as a plate located under or above a concrete column. (The plate stiffness will be multiplied by 100).	Double-click or Space
Integrity	Activate this option [x] if this rigid plate must include integrity reinforcement and VisualDesign will place extra reinforcement. This type of plate could be located above a concrete column.	Double-click or Space bar
Bending	To consider bending in the stiffness matrix of the plate, choose option [x].	Double-click or Space bar

Column	Description	Editing
Stiffness Ratio Bending	According to chosen units, specify the stiffness ratio or percentage that will be considered for bending.	Single click
Shear Torsion	To consider shear and torsion in the stiffness matrix of the plate, choose option [x].	Double-click or Space
Stiffness Ratio Shear	According to chosen units, specify the stiffness ratio or percentage that will be considered for shear and torsion.	Single click
Vertical alignment	Alignment of this plate: "Manual", "Above" or "Below". If a manual one is selected, the default eccentricity is 0.0, meaning that the centreline of the plate is in line with end nodes.	Double-click
Eccentricity	Eccentricity of the plate relatively to its centreline.	Single click
For a project with composite elements only		
Effective from Construction Stage:	Indicate from which construction stage this plate will be considered effective (solid) for finite element analysis of concrete plates.	Double-click
Consider Ratio "n"	To consider ratio "n" (E steel/E concrete), as defined in the **Composite Beam** tab **of Project Configuration**, choose option [x]. If this option is not activated, ratio "n" will be equal to 1 for long-term and short-term effects.	Double-click or Space bar

See also

Triangular Plates Spreadsheet

Rectangular Plates Spreadsheet

Groups of Plates - Surfaces

Groups of Plates - Shear Walls

Composite Beam tab (Project Configuration)

Rectangular Plate Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	No
Node i number	Node number at corner i	Single click
Node j number	Node number at corner j	Single click

Column	Description	Editing
Node k number	Node number at corner k	Single click
Node l number	Node number at corner l	Single click
Lij	Length between nodes i and j	No
Ljk	Length between nodes j and k	No
Lkl	Length between nodes k and l	No
Lli	Length between nodes l and i	No
Area	Calculated area of the plate	No
Centroid	Centroid of this plate relatively to the axis of gravity of the project. This coordinate can be useful to select plates located at a level when there are many of them in a building.	No
Moving load axis	Select the moving load axis number.	Double-click
Location	Select the side of the plate that is part of the moving load axis or choose N/A.	Double-click

Triangular Plate Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	No
Node i number	Node number at i	Single click
Node j number	Node number at j	Single click
Node k number	Node number at k	Single click
Lij	Length between nodes i and j	No
Ljk	Length between nodes j and k	No
Lki	Length between nodes k and i	No
Area	Calculated area of plate	No

Column	Description	Editing
Centroid	Centroid of this plate relatively to the axis of gravity of the project. This coordinate can be useful to select plates located at a level when there are many of them in a building.	No
Moving load axis	Select the moving load axis number.	Double-click
Location	Select the side of the plate that is part of the moving load axis or choose N/A.	Double-click

Bill of Materials (Plates)

The bill of materials for plates is available in the **Results** menu, under **Bill of Materials**.

Group: Structural Data

Column	Description	Editing
Material	Material of the plate	No
Thickness	Thickness of the plate	No
Weight	Weight of all plates composed of the same material.	No
Surface	Area of all plates composed of the same material.	No

Groups of Plates

Groups of Plates - Surfaces

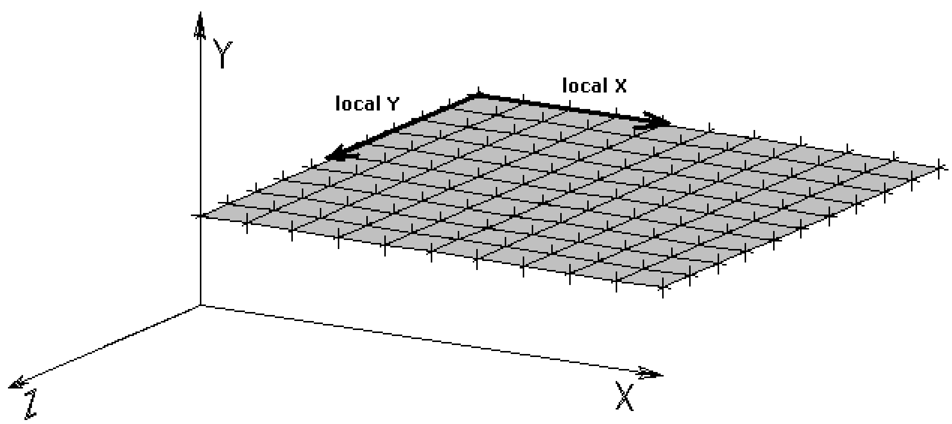
Access this spreadsheet by selecting **Plates - Surfaces** under the **Structure/Groups** menu.

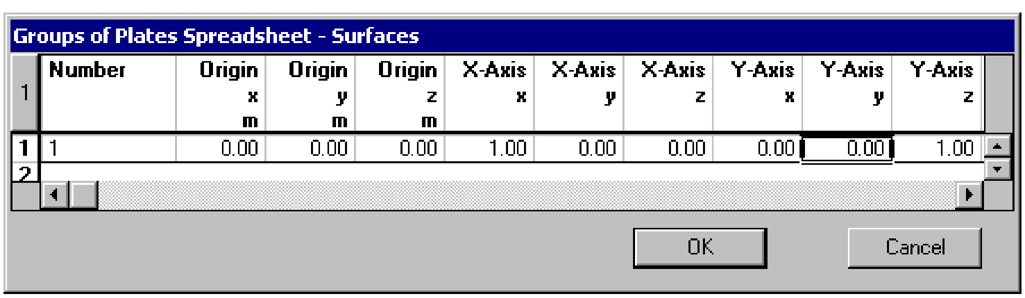
This spreadsheet is useful to interpret finite elements results. A local axis system is assigned to a group of horizontal or vertical plates that are composing a slab or a simple wall (not shear walls). Results will be homogeneous and so it will be easier to read results.

Design of 2-way slabs

Plates are oriented through direction vectors that are normalized by VisualDesign™.

A group of plate must be assigned to the plates that are composing the concrete slab. A specification must be selected and concrete covers must be specified.





Groups of Plates Spreadsheet - Surfaces

	Number	Origin x m	Origin y m	Origin z m	X-Axis x	X-Axis y	X-Axis z	Y-Axis x	Y-Axis y	Y-Axis z
1	1	0.00	0.00	0.00	1.00	0.00	0.00	0.00	0.00	1.00
2										

OK Cancel

Description of spreadsheet columns:

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Group number or name	Single click
Origin x	Global x-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
Origin y	Global y-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
Origin z	Global z-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
X-Axis x	Local x-axis component of the group projected on the global x-axis.	Single click
X-Axis y	Local x-axis component of the group projected on the global y-axis.	Single click
X-Axis z	Local x-axis component of the group projected on the global z-axis.	Single click
Y-Axis x	Local y-axis component of the group projected on the global x-axis.	Single click
Y-Axis y	Local y-axis component of the group projected on the global y-axis.	Single click
Y-Axis z	Local y-axis component of the group projected on the global z-axis.	Single click
Concrete Design		
Specification	Design of 2-way slab: Select the concrete specification.	Double click
Top Cover	Specify the concrete cover at the top of the slab.	Single click
Bottom Cover	Specify the concrete cover at the bottom of the slab.	Single click
Lateral Covers	Specify the lateral concrete covers of the slab.	Single click

See also

The Plates Spreadsheet
Design of 2-way slabs
Numbering Convention of nodes
Plane Stresses Convention
Principal Stresses Convention
Interpreting Results

Groups of Plates - Shear Walls

This spreadsheet is available in **Structure/Groups** / **Plates – Shear Wall.**

This spreadsheet is required for a shear wall design. Common local axes are assigned to plates that are composing a wall section, which can be of any form (T, C, or cubic). The longitudinal axis of the shear wall must be parallel to gravity axis (up or down) according to the right hand rule. Otherwise, VisualDesign will not be able to build the vertical continuous system, which is represented by a vertical fictitious member.

VisualDesign normalizes vector directions.

Groups are assigned to plates through the Plates spreadsheet or **Plate Characteristics** dialog box.

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Group number or name	Single click
Strong axis x	If the strong axis of the shear wall is pointing towards the positive global x direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click
Strong axis y	If the strong axis of the shear wall is pointing towards the positive global y direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click
Strong axis z	If the strong axis of the shear wall is pointing towards the positive global z direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click

See also

Modeling and designing shear walls

Bolted Connections

Bolted Connection Definition Spreadsheet

Select **Bolted Connections** in **Structure** menu and specify the required parameters for connections and bolts that will be used for the design.

Go to topic **Bolt Layouts** for more information about variables that are part of this spreadsheet.

Group: Structural data

Column	Description	Editing
Specification ID	Automatically calculated	No
Connection Number	Connection number (Up to 16 alphanumerical characters)	Single click
Connection Model	Choose a connection model among the list box. See topic Bolted Connection Models	Double-click
Model 5 and 9 Other section	Additional section used as connector for models 5 and 9.	Double-click
Nos. of bolts	Total number of bolts for this connection	Single click
Bolt	Choose the bolts that are used in this connection.	Double-click
Bolts Layout	Choose the bolts layout: In line, Staggered A, or Staggered B. See topic Bolts Layout.	Double-click
Nos. of transverse lines	Specify the number of transverse lines for this connection.	Single click
Nos. of longitudinal lines	Specify the number of longitudinal lines for this connection.	Single click
Nos. of planes Bolts	Specify the number of planes considered for bolts. *This parameter is different from shear planes.*	Single click
Nos. of planes Member	Specify the number of planes considered for member(s).	Single click
Nos. of planes Plate	Specify the number of planes considered for plate(s).	Single click
Intercepted Threads	If bolt threads are intercepted in this connection, choose option [x].	Double-click or Space Bar

Column	Description	Editing
Member em	Longitudinal distance measured from free edge of steel angle to the first bolt in a row.	Single click
Member p	Longitudinal distance measured centre-to-centre between two bolts.	Single click
Member g1	Transverse distance measured from outside edge of bent angle to the nearest bolt row.	Single click
Member g2	Transverse distance measured centre-to-centre between two consecutive bolt rows.	Single click
Plate ep	Longitudinal distance measured from free edge of plate to the first bolt in a row.	Single click
Plate t	Thickness of the plate used in the connection.	Single click
Plate Fu	If a plate is part of this connection, specify its tensile strength.	Single click

See also

Bolts Steel Grades
Bolts spreadsheet
Connection Models
Bolts Layout
Checking or designing bolted connections

Chapter
3

MODELING & EDITING ELEMENTS

TABLE OF CONTENTS

Chapter 3 Modeling & Editing Elements

Creating a Structural Model

Modeling Strategy

When modeling a structure, it is important to work with strategy in order to minimize errors and time for inputs. Here are recommended steps to model a structure in the most productive way:

- Identify reference spatial coordinates;
- Always work according to functionalities that you will be using. You must master the many **Split** functions (Multiple split, Split according to node, Split at exact position, Split with a pin connection and Split with a rigid connection) and the **Copy/Paste** function (translation, rotation and mirror) before modeling a structure;
- Before splitting a member, specify design criteria such as the building code that you will be using, the type of shape and others. End conditions are also <u>very</u> <u>important</u> before you split a member (hinged or fixed ends);
- Adjust beta angle at this step. (If can use the **Rotate** function also);

  If you cannot place the member in the right direction in space with the beta angle (it can happen for single symmetrical shape), use option "Invert Node I ↔ Node j" in the **Member Characteristics** dialog box.
- When these steps are completed, split members. Members will keep original characteristics (beta angle, end conditions and design criteria).
- When members are split, define load titles and types (live, dead add., snow, etc.), load combinations, load factors and envelopes. Then, apply load on the structure directly by double-clicking on elements on the screen (in the "Load Case" activation mode).
- Use VisualDesign **View Options** to verify the structural model and loads that you applied on it.
- If you are planning to run a modal analysis, model a rough structure in order not to create local vibration modes (Ex.: mezzanine, walkway or footbridge) but do not forget to transfer these dead loads on nodes.

See also

Systematic Process from Input to Output
Practical example - 2D Frame

Structure Modeling Process from Input to Output

This systematic process includes a brief description of the different stages in creating a project with VisualDesign™.

VDBase.mdb Database

This database includes data on materials, shapes, reinforcement, cables, studs, steel decks and soils. Users are allowed to add new data so it can become a customized database. In addition, it can be shared among users: copy the database in VisualDesign™ "Sections" directory on your workstation.

P.S. This database will not be overwritten if you update VisualDesign™.

Modeling

Before beginning with the modeling, configure your project. Select **Project Configuration** under **File** menu.

To quickly model a structure (nodes, supports, members and floors), use the **Building Generator** function. You will find this tool in the **Structure/Generator** menu.

You can create elements separately by using the spreadsheets in the **Structure** menu or directly on your screen with the **Add** function (**Edit** menu).

Definition of Loads

Once that the structure is modeled, define load case titles and types. Select the **Loads Definition** spreadsheet under **Loads/Load Cases** menu and insert the number of lines corresponding to the number of load case type that you need in your project. At each line, double-click in the "Type" cell to specify the type of load.

Then, activate the Load Case mode and choose a type of load in the drop-down list box of Activation toolbar.

Now you are ready to apply loads on your structure.

P.S. You must define at least one load combination to run VisualDesign™.

Generation and Definition of Load Combinations

A quick way to create load combinations is through the **Load Combination Generator**. This tool, available under **Loads/ Load Combinations/ Automatic Generation** menu, will generate load combinations according to a selected code or Standard.. Load factors can be modified. When the generation if finished, you can deactivate load combinations that you do not wish to analyze.

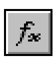
If you prefer to define load combinations yourself, select the Combinations dialog box (**Loads/Load Combinations/Definition)**. This dialog box also includes a **Load Factors** tab in which you will specify the load factor for each load that is part of the load combination.

Analysis – Design - Verification

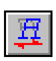
There is no limit to the number of elements included in a model, neither concerning the number of load cases or load combinations. VisualDesign™ cannot analyze a structure that has no support nodes. If a user forgot to assign a type of shape or material to members, a warning message will be posted on the screen. In addition, the user must define at least one load combination to run VisualDesign™.

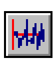
Structures can be analyzed, checked or designed. To check or design a structure according to a code, you must define a steel (or concrete) specification. Specifications spreadsheets are available under the **Structure/Specifications** menu. They also include other parameters corresponding to design criteria.

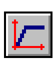
Available analyses are:

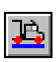
 **Static Analysis**

 **Dynamic Analysis**

 **Spectral Analysis**

 **Time History Analysis**

 **Non-linear Time History Analysis**

 **Moving Load Analysis**

 **Analysis and Design**

Results

VisualDesign™ automatically activates the Load Combination mode once that the analysis is done. You must choose a load combination title among the drop-down list box of Activation toolbar.

To look at an envelope results, activate the Envelope mode and select an envelope title among the drop-down list box of Activation toolbar.

Results may be viewed in many ways:

- Select the **Results** tab or **FE Results** (Finite Elements) tab of **View Options** dialog box. Check the box that corresponds to the results (numerical or graphical) that you want to see on the screen;

- With the mouse, double-click on any element to call up the results dialog box of this element;
- Select one of results spreadsheets in the **Results** menu to consult load combination results (nodes, members, plates, etc.) or envelope results.

Code and Element Alphanumerical Number

VisualDesign™ gives a code number (ID) to all newly created elements. The user cannot modify this number.

More, each element has an alphanumerical number. While the automatic generation is in process, the user sets the first value of the numbering in the X, Y, and Z-axis. The application displays the elements numerical values according to the selected gravity axis.

Selected Gravity Axis	Numbering (Automatic Generation)
X	(Y, Z, X)
Y	(X, Z, Y)
Z	(X, Y, Z)

It is possible to modify the alphanumerical numbering using the structure spreadsheets. To do so, select the numbers to be changed, right click to get access to the contextual menu. Select the **Auto-numbering** function.

The spreadsheets will not accept identical alphanumerical numbers for elements of the same category.

When elements are created individually using the **Add** function, VisualDesign™ gives them a default code number (ID) as alphanumerical number.

Note. The **Copy/Paste** function may generate identical alphanumerical numbers under certain circumstances (e.g. when duplicating elements or parts of a structure). These identical numbers are shown in the nodes spreadsheets. The "Auto-numbering" option of the spreadsheet contextual menu may be used to re-number nodes.

See also

The Add Function

Generators

The Building Generator

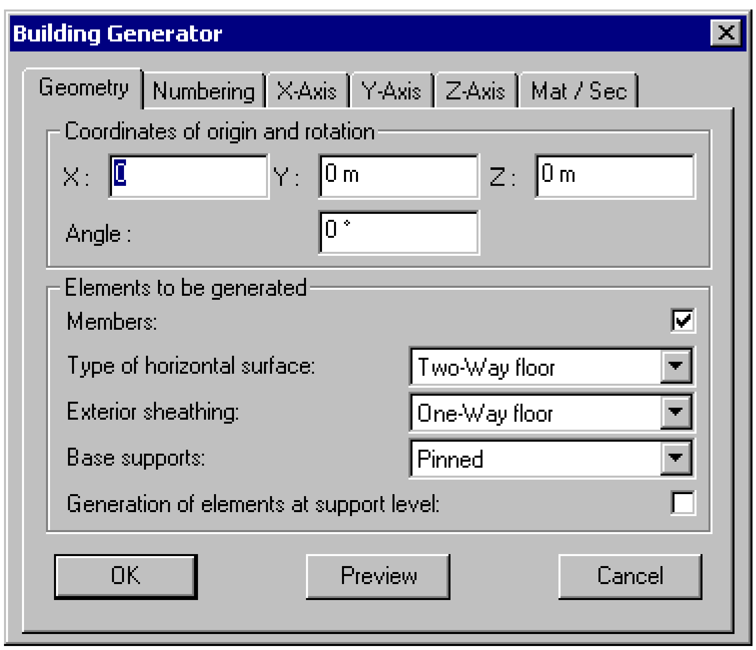
The "Building Generator" icon of Tools toolbar

Use the **3D Building Generator** function (**Structure/Gnerators)** menu to automatically generates a building according to specified dimensions, material and shapes included in the following tabs: **Geometry**, **Numbering**, **Axis X**, **Axis Y, Axis Z** & **Mat/Sec**.

When the function is called up, the **Building Generator** dialog box opens on screen.

The Geometry tab

Select the **Geometry** tab of the **Building Generator** dialog box to specify parameters needed for the generation of the structure. The gravity axis corresponds to the global y-axis, as specified in the **Preferences** tab of **Project Configuration**.



See the table below for the definition of parameters included in this tab.

Field	Description
Coordinates of origin and rotation	
X, Y, Z	Generate the structure from the starting coordinates entered in these fields.
Angle	Angle of rotation of the structure relative to the X axis, if Y is the axis of gravity; or relative to the Z axis, if X is the axis of gravity; and finally, relative to the Y axis, if Z is the axis of gravity. Note: Positive clockwise.
Elements to be generated	
Members	Members are automatically generated when this check box is activated.
Type of horizontal surface:	Select the type of horizontal surface to be generated: Plates, 2-way floor, one-way floor, joist floor or None. Note: if the selected surface is a floor, the member check box cannot be disabled.
Exterior sheathing:	Select the type of vertical surfaces (sheathing) to be generated around the building: Plates, 2-way floor, one-way floor, joist floor or None. (Wind loads can be applied to these surfaces afterwards.)
Base supports	If you wish to automatically generate supports at base level, choose "pin" or "fixed", otherwise, choose "none".
Generation of elements at support level	When this check box is activated, the members, plates and floors are generated starting from the first level (supports) of the structure.

If you want to look at the structure without exiting the dialog box, press the "Visualize" button. If it is OK, press the OK button. You will then exit the dialog box.

The Numbering tab

Select the **Numbering** tab to specify some parameters needed to generate the structure's element numbering.

See the table below for the definition of parameters included in this tab.

Field	Description
Starting Values	
X, Y, Z	Begins the nodes numbering according to the values indicated in these edit boxes. The numbering increases in the positive direction of the axes.
Separators	
Between X and Z	Inserts a separator character, indicated in this box, between the X and the Z numbering.
Between Z and Y	Inserts a separator character, indicated in this box, between the Z and the Y numbering.
Member, Floor, Plate	Inserts a separator character, indicated in this box, in the numbering of members, plates and floors.

The X-Axis tab

Select the **X Axis** tab of **Building Generator** dialog box to specify the distance between bays that will be generated in the x direction.

Group: Structural data

Column	Description	Editing
Delta	Distance centre to centre between two axes in the x direction.	Single click

Press the "Preview" button to visualize the generated mesh. If it is correct, press the OK button.

The Y-Axis tab

Select the **Y Axis** tab of **Building Generator** dialog box to specify the distance between the bays (or stories, if y is the gravity axis) that will be generated in the y direction.

Group: Structural data

Column	Description	Editing
Delta	Distance centre to centre between two axes in the y direction.	Single click

Press the "Preview" button to visualize the generated mesh. If it is correct, press the OK button.

The Z-Axis tab

Select the **Z Axis** tab of **Building Generator** dialog box to specify the distance between bays that will be generated in the z direction.

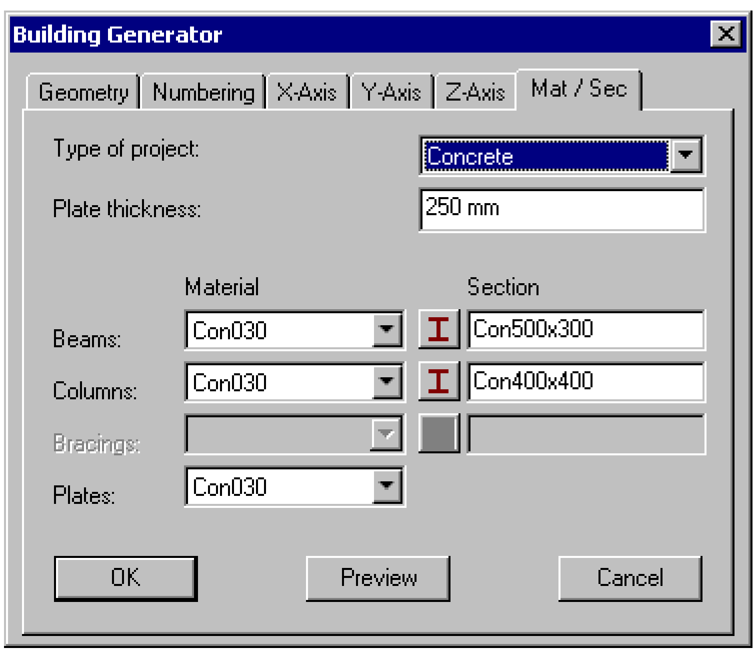
Group: Structural data

Column	Description	Editing
Delta	Distance centre to centre between two axes in the z direction.	Single click

Press the "Preview" button to visualize the generated mesh. If it is correct, press the OK button.

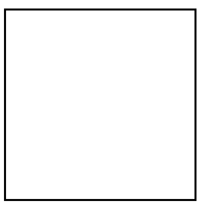
The Mat/Sec tab

Select the **Mat / Sec** tab of **Building Generator** dialog box and specify the material and shape that will be assigned to generated members. Default values are supplied depending on the type of generated structure (steel, concrete, timber or aluminium).



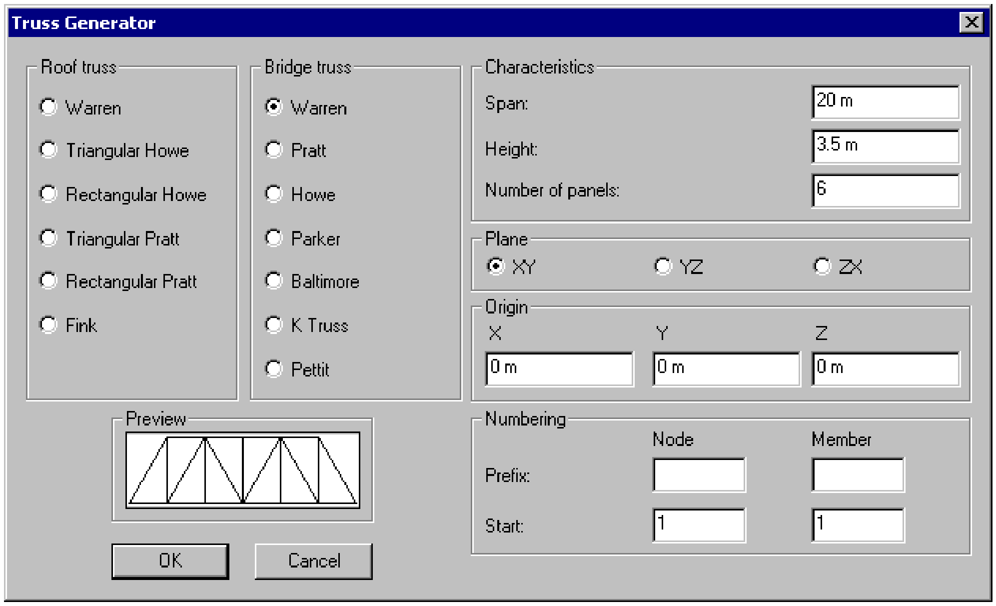
Field	Description
Type of Project	Select one option (*Steel*, *Concrete*, *Wood*, *Aluminium*) among the drop-down list box that describes the main material used in the building. Appropriate selection trees will be available for each element listed below. Choose option *Null* if you do not want to assign materials and shapes to members.
Plate thickness	Specify the thickness for the generated plates.
Material / Shape	
Beams and Columns	Click on the arrow and select a material in the drop-down list box. Then, click on the icon that represents a I beam and select a shape in the selection tree.
Plates	Click on the arrow and select a material in the drop-down list box.

The Truss Generator



The "Truss Generator icon of Tools toolbar

Press this icon on Tools toolbar or select this function under **Structure/Generators** menu to have access to a dialog box that generates pre-existing trusses.



Customized Options

CivilDesign is happy to put its programming expertise, intellectual abilities and innovative approach at your service by creating customized applications to optimize your operations and increase the reliability of results.

We can develop an automated customized structure <u>just for you</u> to make your design quicker than ever! We can also upgrade and improve your house made programs into Windows environment!

Save time and money: Contact CivilDesign today!

Phone: (450) 674-0657

Toll Free 1-800-724-5678

E-mail: custom@civild.com

Modeling Tools

Axis Transformation

The "Axis Transformation" icon of Tools toolbar

Generate circular or parabolic curves from existing segments of your structure, using the **Axis Transformation** function under **Structure/Tools** menu.

You must at least select three nodes to be allowed to use this function. N.B. If nodes were not merged with others when generating a particular curve, nodes will still be selected.

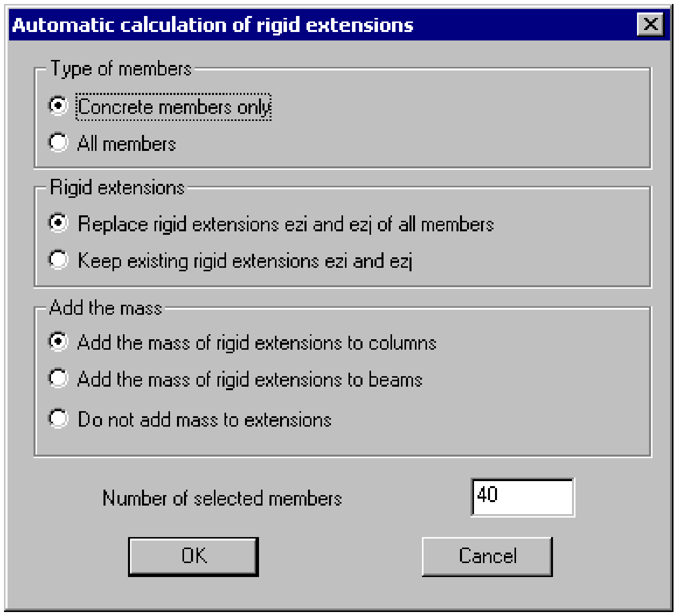
Description of the dialog box:

Field	Description
Type of Arc	Allows to choose the type of arc to generate: Linear-Po fixed: Build a linear segment between the Po and the Pn with Po fixed. Linear-Pn fixed: Build a linear segment between Po and Pn with Pn fixed. -Parabolic: Build a parabolic arc between Po and Pn. -Half Circular arc: Build a half circular arc between Po and Pn. -Circular arc: Build a circular arc between Po and Pn.
Calculation of Coordinates	Two options are offered for the curve generation: Nodes stay perpendicular to the axis: In this case, nodes only move vertically. To obtain equidistant segments: In this case, nodes are moved to obtain equal segments.
Parameters	Two types of curves can be generated: Symmetrical or Linear Shape. This case allows a rise to the curve. Non-symmetrical Shape. This case allows generating a parabolic curve using a known equation.
Axis Rotation	See Convention of Angle beta

Automatic Calculation of Rigid Extensions

The tool **Automatic Calculation of Rigid Extensions**, located in the **Structure/Tools** menu, calls up a dialog box that will help you model steel or concrete member rigid extensions by calculating them automatically.

This functionality will automatically create rigid extensions at the face of each support, for concrete members only or for all members of your structure. It can also replace the already defined rigid extensions or keep the old ones. More, you can choose to automatically add these weights to columns or to beams or not considering any rigid extension weight in your project.



See also

The Connection tab

Rigid Extensions

Modeling Valid Rigid Extensions

Auto-Hinge Function

This function, available in **Structure / Tools**, adds hinges along selected continuous columns while working in the Structure mode. Hinges will be placed according to a specified length.

This length may represent the maximum length that is manufactured or the capacity of a truck for transport.

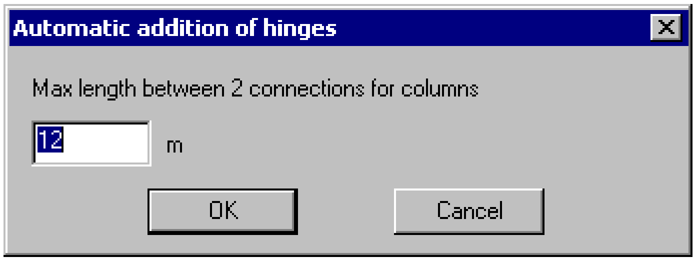
RESTRICTIONS:

We recommend using this function for columns only.

This function can be applied to all materials, except concrete.

Using the Auto-Hinge Function

- Activate the Structure mode.
- Make sure that a steel specification is assigned to columns that will be grouped before using the function **Auto Group** (**Structure / Groups**).
- Select continuous columns that you want to split with hinges.
- Go to **Structure / Tools** and select **Auto Hinge**.
- Enter a maximum length in the following dialog box and press OK.



- To look at created hinges, activate the option "End conditions" in the **Attributes** tab of **View Options** dialog box.
- Use function **Auto Group**, located in menu **Structure / Groups**. VisualDesign will automatically group these columns according to the positions of hinges.

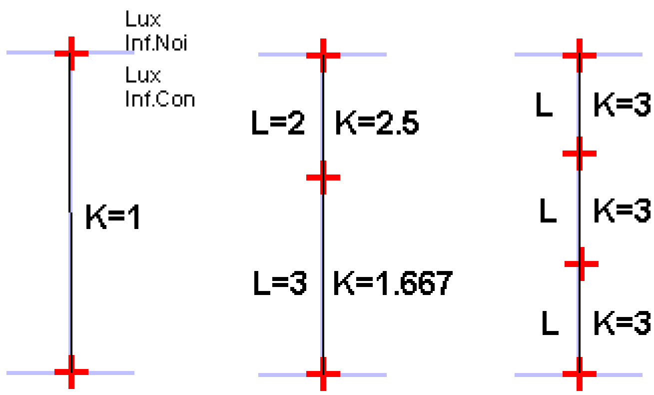
Automatic Calculation of Kx, Ky, Kt and Kz

This function, available under the **Structure/Tools** menu calculates the effective length factor (K) for a restrained member located between two joints. The user has to specify K factors for particular cases.

In fact, the program verifies if lateral bracing has a minimum stiffness to prevent the buckling of the member that is attached. For example, if VisualDesign™ found that some lateral members have not a sufficient stiffness, members that are attached to those members will be design with a greater KL/r. The calculation of the ideal stiffness is based on the theory of Theodore V. Galambos *Guide to stability design criteria for metal structures*, 4th edition, 1988, pages 55 to 57.

Kx and Ky values are generally equal to 1. If elements are continuous and make up a sole column between two floors, VisualDesign™ calculates new K values, for each element, in order to obtain KL value equal to the total length.

Example:



For truss crossing members connected at their centre, the program considers a K factor of 1, on half of their total length (distance between end connection and bolt connection). If you wish to consider the total length of crossing members in the case where the two bracings are in compression at the same time, you must create a group for these members and specify Kx and Ky factors in the **Steel Design** tab of **Member Characteristics** dialog box.

Automatic calculation of Kt:

The Kt factor is used for the computation of buckling due to torsion for single symmetrical shapes 2L, WRF, V, WT and cold-formed sections. The automatic calculation of Kt will be done according to the selected option in the **Steel** tab of **Project Configuration** dialog box.

Automatic calculation of Kz:

The Kz factor is used to calculate the buckling of single steel angles in an orthogonal axis system for the design of towers and antennas. The automatic calculation of Kz will be done according to the selected option in the **Steel** tab of **Project Configuration** dialog box.

See also

Steel Design Criteria
Steel Design Results
Steel tab (Project Configuration)
The Tower Design Module

Tributary Areas for Spring Supports

Use this tool, which is located in **Structure** menu / **Tools**, to automatically calculate tributary areas in the x-, y- and z-direction, for spring supports that are associated to plate elements.

Procedure:

- Activate the Structure activation mode.
- Select spring supports.
- Go to **Structure / Tools** and select **Calculation of Tributary Areas**.

Calculated areas will be written in the **Support** tab (**Node Characteristics** dialog box).

N. B. For the Generation of Abutments, Piers & Retaining Walls module, spring support tributary areas are automatically calculated from the stratigraphical profile data and are indicated in the **Support** tab.

See also

The Support tab
The Spring Supports spreadsheet

Selection Windows

Activate Selection Window

VisualDesign™ allows one of two selection window modes: the **Extended Window** and the **Restricted Window**. Both windows allow you to select an element or group of elements. Whichever mode is used, the whole element is selected.

The difference between the two selection windows appears when you use the pointer to plot a window around an element or a group of elements.

Whichever mode you chose, you must always specify the type (member, node, plate etc.) of element that you want to select. The list of available elements appears both in the **Edit** Menu, under the "Elements" heading, and on the Elements toolbar.

See also

Extended Window

Restricted Window

Selecting Mixed Elements

Using the Pointer (in Extended Window mode or Restricted Window mode)

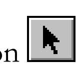
Extended Window

The "Extended Window" icon of Cursor toolbar

When you use the **Extended Window**, all objects touching the selection window and inside it will be selected.

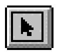
To simultaneously select elements or groups of the same type, keep the [Ctrl] key down while clicking on objects.

Selecting Objects using the Extended Window

- Indicate the type of element you want to choose by activating one of the buttons on the Elements toolbar.
- Then, do one of the following:
  - Click the icon  on Cursor toolbar.
  - Go to **Edit / Activate Window** and choose **Extended Window**.
- With the help of the mouse pointer, trace a window around the element(s) that you wish to select while keeping the left mouse button pressed.

Only elements belonging to the activated category within or touching the window will be chosen.

Restricted Window



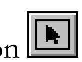
The "Restricted Window" icon of Cursor toolbar

When using the **Restricted Window**, only elements *completely located inside this window* will be selected if the cursor moves from left to right. The selection window is displayed with dotted lines.

If the cursor moves from right to left, this selection mode changed to an **Extended Window** mode. The window selection is then displayed with continuous lines.

To simultaneously select elements or groups of the same type, keep the [Ctrl] key down while clicking on objects.

Selecting Objects with the Restricted Window

- Indicate the type of element that you wish to select by pressing one of the buttons on the Elements toolbar or by activating one of the options under **Activate Elements** on the **Edit** menu.
- Then, do one of the following:
  - Click the icon  on Cursor toolbar.
  - Go to **Edit / Activate Window** and choose **Restricted Window**.
- Use the mouse pointer, ***from left to right***, and draw a selection window around the element(s) that you wish to select. Only elements that are completely contained within the window will be selected.
- To change to an extended selection mode, draw the selection window from ***right to left***.

Only elements that are completely contained within the window will be selected.

Using the Pointer in Extended or Restricted Window

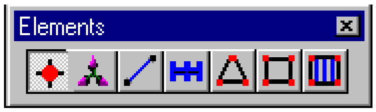
To	Using the Pointer
Select an object	Click once on the object with the pointer
Selecting mixed elements	Keeping the [Shift] key down, draw a window to select all elements.
Select a group of objects of the same type	Click on one of the objects keeping the [Ctrl] key down while clicking on the next objects
Withdraw one object from the selection	Keeping the [Ctrl] key down, click once on (or near) the selected object with the pointer
Cancel all the selection	Click on one of the objects without keeping the [Ctrl] key down
Cancel all the selection while selecting a new object	Click on one object not part of the selection without keeping the [Ctrl] key down

See also

Activate Selection Window
Selecting Mixed Elements

Activate Elements

The Elements Toolbar



The Element toolbar of VisualDesign Main Window

Before selecting an element on your screen or activating a function on **Edit** toolbar, you must activate a type of element on Elements toolbar. It may be a node, support, member, continuous system, plate or floor.

Activating an Element

- Do one of the following:
  - Select one of the icons on the Elements toolbar and go to **Edit** / **Activate Elements** and choose a type of element.
  - Use the short-cut keys: [Ctrl]+1 for activating nodes, [Ctrl]+2 for activating supports, [Ctrl]+3 for activating members, [Ctrl]+4 for activating continuous systems, [Ctrl]+5 for activating triangular plates, [Ctrl]+6 for activating rectangular plates, and [Ctrl]+7 for activating floors.

Select Elements

The Select Function

The **Select** function of **Edit** menu allows selecting specific elements in your model such as:

- Columns
- Personalized Selections of Mixed Elements (1)
- Beams
- Inclined Members
- Rectangular plates
- Triangular Plates
- Floors
- Horizontal Continuous Systems
- Vertical Continuous Systems
- Members according to Usage
- Inactive Nodes
- Incomplete Modeling

This submenu also includes functions **Cancel** and **Invert selection**.

When elements are selected, they are highlighted on screen.

Personalized Selections of Mixed Elements

This submenu also includes functions to create personalized selections of miscellaneous elements. These selections can be called back at anytime while working in the project, helping you to quickly edit and consult results. Functions are:

- Create a Selection
- Choose Selection(s)
- Update Current Selection
- Edit selections

Selecting Mixed Elements

To select different types of elements, do as follows:

- Use the [Shift] key while drawing a window to select elements, either in the **Restricted Window** or **Extended Window** selection mode. All elements (nodes, supports, members, plates, and floors) will be selected if they are displayed through the **View Options**. (Tip: Use this function and make the structure, then, create a selection of mixed elements. Refer to Personalized Selections of Mixed Elements.)
- Activate the element icon on Activation toolbar and select elements of the same type. Then, activate another element and select other elements while keeping the [Ctrl] key down. Note that the **Delete** function applies to both selections.

See also

Extended Window

Restricted Window

Select Identical Items in a Spreadsheet

Spreadsheet Contextual menu: Access the spreadsheet's contextual menu in any open spreadsheet by clicking the right mouse button.

Function **Selection of Contiguous Identical Items** is useful to group elements, to look at results, to modify parameters, etc.

Example:

Open the Steel Design Results spreadsheet and sort member shape W310x39. Then, use function **Selection of contiguous identical items** to select all W310x39. Click OK. These members will be highlighted on the screen. Go to **Edit** menu and select **Create a Selection** or press icon . Give a name to this selection. You can call back this selection any time.

See also

Selection of Contiguous Identical Items

Sort in a spreadsheet

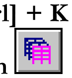
Create a selection

Personalized Selections of Mixed Elements

This functionality includes four commands located in **Edit** / **Select** menu and is useful to create and save selections of miscellaneous elements (nodes, members, plates, and floors). A colour can be assigned to each selection and can be displayed on screen or disabled. The same element can be part of many selections.

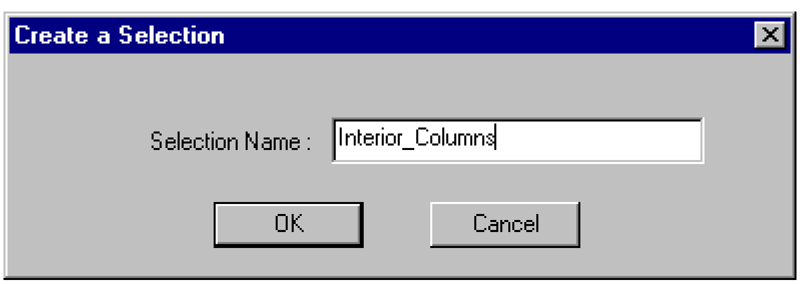
Quickly select mixed elements by pressing the [Shift] key, in either the **Restricted** or the **Extended** window selection mode, while drawing a window with the cursor. All elements will be selected if they are displayed on screen through the **View Options**.

Look at the table below to have a description of each function.

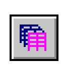
Function	Description	Shortcut /Icon
Create a selection	Creation of a new selection from selected elements on the screen.	**[Ctrl] + L**
Choose a selection	Choose one selection or more in the **Selections** dialog box. Selections will be highlighted on screen.	**[Ctrl] + K or** **Icon** 
Update current selection	Choose one selection, add or withdraw elements, and update the final selection using this command.	
Edit selections	Assign a colour to a selection, delete a selection or modify a selection title in the **Selections** spreadsheet.	

Create a selection

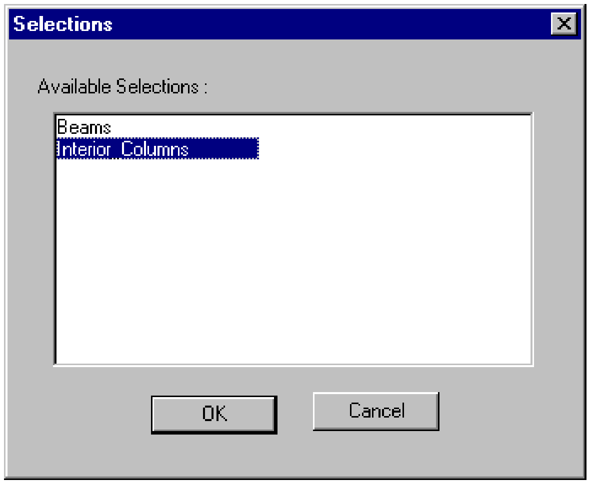
- Activate an element on Elements toolbar and, while you keep the [Ctrl] key down, click on elements. Then, activate another type of element and, while you keep the [Ctrl] key down again, click on other elements.
- Use shortcut keys [**Ctrl**]**+L** or go to **Edit / Select** / **Create a Selection**. In the following dialog box, give a name to this selection. Press OK.



Choose a Selection:

- To choose selections, press this icon  on Edit toolbar or use the shortcut keys **[Ctrl] + K** or go to **Edit / Select** / **Choose a Selection**. Click on selection titles and press OK. All selections will be highlighted on the screen.

N. B. You can choose more than one title among the list.



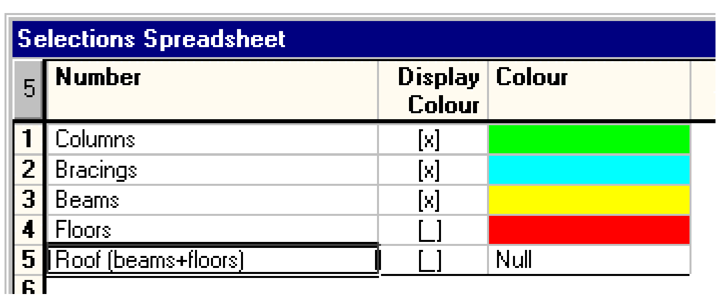
Update Current Selection:

- Choose <u>one</u> selection (**Edit / Select / Choose a Selection**).
- Keep the [Ctrl] key down while you click on elements that want to add or withdraw from selection.
- Go to **Edit** menu and select **Select / Update Current Selection**.

Edit Selections:

This function can be called up to assign a colour to each selection, to disable the colour, or to erase a selection or modify titles.

- Go to **Edit / Select / Edit Selections**.
- In the **Selections** spreadsheet, double-click in the Colour cell and select one. Then, to activate the display, double click in the next cell to get this symbol: [x].
- To delete a selection, select the line and press [Delete].
- To modify the title, double click in the *Number* cell and enter a new title.



Selections Spreadsheet

5	Number	Display Colour	Colour
1	Columns	[x]	
2	Bracings	[x]	
3	Beams	[x]	
4	Floors	[ ]	
5	Roof (beams+floors)	[ ]	Null
6			

Notes:

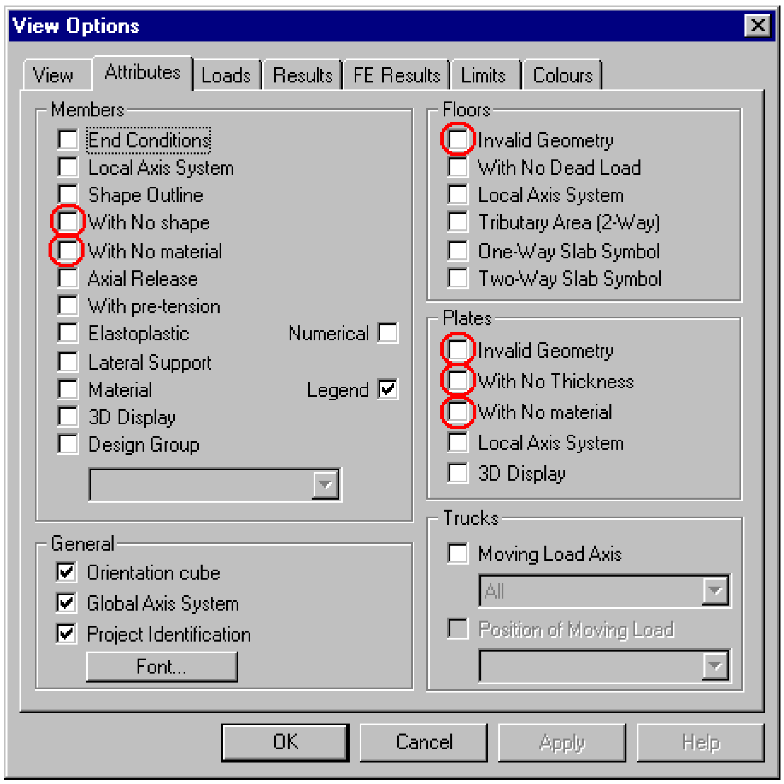
If an object is part of more than one selection, the colour that will be displayed is the colour that was assigned to the first selection that includes this object.

If split functions are used on elements that are part of a selection, the newly created elements will be part of the selection that included original elements.

Incomplete Modeling

This tool, available in **Edit** menu / **Select Elements**, selects elements with incomplete or inadequate modeling such as members with no assigned shape or material, floors and plates with invalid geometry and plates with no assigned material or thickness.

In fact, this tool activates the following options, which are listed in the **View Options**' **Attributes** tab:



When invalid elements are selected using this function, click on each one in the Structure activation mode or select many of the same type and call up the default spreadsheet by pressing the shortcut keys **[Ctrl]+H**. The appropriate spreadsheet (members, Floors or Plates) will be displayed on screen. Complete the missing information such as materials, shapes or thickness).

Concerning invalid geometry for plates and floors, VisualDesign will notice them only if an analysis is launched. A warning message will appear on screen to inform you that it detected invalid geometries. Then, use the function **Incomplete Modeling** to locate and select them. Correct node coordinates for these surfaces.

Invert Selection

This function, available in **Edit** menu under command **Select**, is useful to select elements that are not selected on the screen. The element icon must be activated beforehand on Elements toolbar.

Shortcut keys: **[Ctrl]+I**

Example:

- Activate the *Member* icon;
- Select columns on the 4th floor and use the **Mask** function;
- Select columns located on the contour and do some modifications;
- Go to **Edit** menu and select **Invert selection** to select the story interior columns.

See also
Mask
The Mask Function (Tricks)

Cancel Selection

This function, available in **Edit** menu under command **Select**, is useful to cancel any selection of elements on the screen.

Shortcut key: [Escape]

Properties of Elements

Properties

The "Properties" icon of the Edit toolbar

With the **Properties** function, you may view or edit the characteristics of an object or the common characteristics of many selected objects while working in the "Structure" activation mode.

While in the "Load Case" mode, and with the load case chosen in the activation toolbar, use the **Properties** function to view and edit the loads applied on selected elements.

Use the **Properties** function while working in "Load Combination" mode, to obtain analysis results for the load combination title selected on the Activation toolbar.

In "Envelope" mode, use the **Properties** function to obtain the analysis results for the envelope title selected in the Activation toolbar.

In the "Design Results" mode, the **Properties** function will allow you to display the **Design Results** Spreadsheet.

The Reinforced and Prestressed Concrete Design modules:

If you are working in the "Structure" mode, the **Properties** function gives access to the **Continuous Systems** spreadsheet.

Activate the "Rebar Placement" activation mode and click on a continuous system. Press the **Properties** function to open the *Rebar Placement* window.

See also

Multiple Modifications of Elements in the "Structure" mode
Multiple Modifications of Applied Loads in the "Load Case" mode
Consulting Load Combination Results
Consulting Envelope Results
Consulting or Modifying Characteristics of a Selected Element

Modifying Characteristics of Selected Elements

- Use one of the following procedures:
  - Click on the "Structure" icon of Activation toolbar.
  - Go to **Edit/Activate Mode /Structure**.
- Select an object by using one of the following procedures:
  - Click the "Properties" icon from Edit toolbar.
  - Select the **Edit/Properties** menu.
  - Use the shortcut key [Ctrl]+T

A dialog box including the object characteristics appears on the screen. Characteristics can be examined or modified.

Consulting or Modifying Characteristics of a Selected Element

- Use one of the following procedures:
  - Click on the "Structure" icon of Activation toolbar.
  - Go to **Edit/Activate Mode /Structure**.
- Select an object by using one of the following procedures:
  - Click the "Properties" icon from Edit toolbar.
  - Select the **Edit/Properties** menu.
  - Use the shortcut key [Ctrl]+T

A dialog box including the object characteristics appears on the screen. Characteristics can be examined or modified.

Applying Multiple Modifications to Elements in the "Structure" mode

- Activate the "Structure" mode using one of the following procedures:
  - Click on the "Structure" icon of Activation toolbar.
  - Select the **Edit/Activate Mode/Structure**.
- Select elements of the same category while pressing down the [Ctrl] key.

- Access the **Properties** dialog box using one of the following procedures:
  - Click on the "Properties" icon  of Edit toolbar.
  - Select the **Edit/Properties** function.

You get access to the **Member Characteristics** dialog box. Empty fields indicate that values will be applied to all selected elements.

- Type in values, then press OK.

Add and Delete Elements

Add an Element

An element addition is possible only while working in "Structure" mode. To get access to this mode, select **Edit/Activate Mode/Structure.**

It is possible to add one or more elements through the spreadsheets available under the **Structure** menu, but it is usually easier to do it directly on the screen using the **Add** command from the **Edit** menu.

To add an element directly on the screen, select its category from the icons of the Elements toolbar (node, support, member, plate or floor). Then, select the "Add" icon of the Cursor toolbar or select the **Add** command from the **Edit** menu.

Note. Right click to cancel any addition in mid action.

A "Continuous system" (Reinforced Concrete and Prestressed Concrete Design) cannot be added with this function because it is already composed of many contiguous members. To copy a continuous system, use the **Copy/Paste** function of **Edit** menu.

Dialog Box Display:

Do not forget that you can stop the displaying of characteristics dialog boxes when adding elements. To do so, un-check the appropriate boxes in the **Preferences** tab of **Project Configuration**.

See also

Code and Alphanumerical Number of Each Element
Adding a Node
Adding a Support
Adding a Member
Adding a Plate
Adding a Floor
Adding Elements to your Model

Procedure:

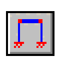
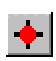
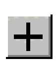
- Activate the Structure mode.
- Select the element type you wish to add.

  The "Continuous System" element is the only one that cannot be added because it is already composed of several elements. See the Reinforced Concrete Design module

- Activate the **Add** mode. To do so, choose **Add Element** from the **Edit** menu or click the "Add" button on the "Cursor" toolbar.
- Click once on nodes that are forming an element.
- To exit the "Add" mode, press the "Cancel" button posted in the **Insertion-Point-Coordinates** dialog box. To cancel a current selection when adding a member or a floor, press the right mouse button.

Remark. To prevent the display of Element Characteristics dialog box when entering new data, deactivate the "Dialog Box Display" option in the **Preferences** tab of **Project Configuration** in (**File** menu).

Adding a Node

- Activate the "Structure" mode by doing one of the following procedures:
  - Click the icon  on Activation toolbar
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the Node element by doing one of the following:
  - Click the icon  on Elements toolbar.
  - Go to **Edit** / **Activate Elements / Node.**
- Activate the **Add** mode by doing one of the following:
  - Click the icon  on Cursor toolbar.
  - Choose **Add Element** from **Edit** menu.
- Determine the coordinates in the **Coordinates of Insertion-Point** dialog box.

If you activate the "Attach-to-Node" option, VisualDesign™ will give you the coordinates of the node closest to your pointer when you press the left mouse button. This option may come in handy when you want to position a new node in relation to an existing node using an x, y or z displacement delta.

- Press OK in the **Coordinates** dialog box to access the **Node Characteristics** dialog box.
- Modify the default node number and indicate what type of node you are dealing with (normal or support). VisualDesign™ will propose, by default, a normal type of node in the "Node Type" field.

  If you want the node parameters to be graphically presented, tick off the appropriate boxes in the **View** tab of **View Options** dialog box.

  You may also modify the X, Y, and Z coordinates.

- Press "OK" to validate the new node parameters or "Cancel" to annul the addition.

  When you exit the **Node Characteristics** dialog box, you will remain in the "Add-at-Cursor" mode to be able to continue to add new nodes.

- To exit the "Add-at-Cursor" mode and return to the previous selection mode (Extended-Selection or Restricted-Selection), press "Cancel".

Restriction when Adding a Node

When you select the "Attach to Node" option from the **Coordinates** dialog box, VisualDesign™ gives you the coordinates of the nearest node to which your pointer is attached when you press down you mouse left button.

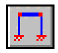
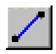
This option may be useful if you wish to position a new node with a displacement delta in *x*, *y* or *z* in relation with an existing node.

However, if you forget to modify one of the coordinates *x*, *y* or *z* in the **Coordinates** dialog box and press down the "OK" button of the characteristics of the node dialog box, you will get an error message.

Adding a Support

- Add a support right in the Nodes spreadsheet or use the following procedure:
  - Follow the same steps as for adding a node.
  - Select a "Support" type of node in the **Node Characteristics** dialog box.

Adding a Member

- Activate the "Structure" mode using one of the following procedure:
    - Click the icon  on Activation toolbar.
    - Select **Activate Mode/Structure** from **Edit** menu.
- Activate the Member element by doing one of the following:
    - Click the icon  on Element toolbar.
    - Select **Activate Elements/Member** from **Edit** menu.
- Activate the "Add" cursor mode using one of the following procedure:
    - Click the icon  on Cursor toolbar.
    - Select **Add Element** from **Edit** menu.
- Using the cursor, plot a member from node i to node j.

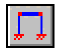
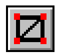
Note that the member point of origin will determine its incidence. This is very important, since the positive direction of the z' axis (local z axis) corresponds to the ij vector of the member (See Angle β Convention).

If you are adding several members, end conditions (Mx, My, Axial and Torsion) will be set from the previous added member.

If you have selected the "Display Dialog Box" option from the **Preferences** tab of the **Project Configuration** dialog box the **Member** dialog box will be displayed. You may choose to fix the parameters at this point, or do so later by setting common characteristics to a group of selected members.

Note. You will remain in the "Add" mode as long as the "Extended" or "Restricted" Window cursor mode is not activated.

Adding a Plate

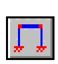
- Activate the "Structure" mode using one of the following procedure:
  - Click the icon  of Activation toolbar.
  - Select **Activate Mode/Structure** from **Edit** menu.
- Activate a Plate element by doing one of the following:
  - Click the icon  on Element toolbar.
  - Select **Activate Elements / Plate** from **Edit** menu
- Activate the **Add** mode by doing one of the following:
  - Click the icon  on Cursor toolbar.
  - Select **Add element** from **Edit** menu.
- Rectangular plate: Plot the rectangular plate by sliding the mouse and clicking on each of the four nodes located in the same plane.
- Triangular plate: Plot the triangular plate by clicking once on each corner (3) and close the surface by clicking the first node. Nodes must be located in the same plane.

Note that the plate's origin point is very important, since the x' axis (local x axis) corresponds to the ij vector of the plate. y' axis is then perpendicular to x' (See Angle β Convention).

If you have activated the "Display Dialog Box" option in **Preferences** tab (**Project Configuration** dialog box), the **Plate Characteristics** dialog box will be displayed. You may choose to fix the parameters right now, or to do so later by setting common characteristics to a group of selected plates.

Note. You will remain in the "Add" mode as long as you do not select the "Extended Window" or "Restricted Window" cursor mode.

Adding a Floor

- Activate the "Structure" mode using one of the following procedure:
  - Click the icon  on Activation toolbar.
  - Select **Activate Mode/Structure** from **Edit** menu.
- Select the type of element you wish to add using one of the following procedure:

- Click the icon on Element toolbar.
- Select **Activate Elements/Floor** from **Edit** menu.

- Activate the **Add** mode using one of the following procedure:
  - Click the icon on Cursor toolbar.
  - Select **Add Element** from **Edit** menu.
- Using the cursor, plot a floor by sliding the mouse and clicking on each of the four nodes located in the same plane.

If you have selected the "Display Dialog Box" option from the **Preferences** tab of the **Project Configuration** dialog box, the **Floor** dialog box will be displayed. You may choose to fix the parameters right now, or to do so later by setting common characteristics to a group of selected floors.

Note. Select the "Extended Window" or "Restricted Window" cursor mode to exit the "Add" mode.

Delete

The "Delete" icon of Edit toolbar

The **Delete** function may be used in "Structure" and "Load Case" activation modes.

In "Structure" mode, this function eliminates the elements selected on the screen. If you cancel nodes, their linked elements are also eliminated.

In "Load Case" mode, the Cancel function eliminates loads applied to selected elements for the chosen load case.

Deleting Elements

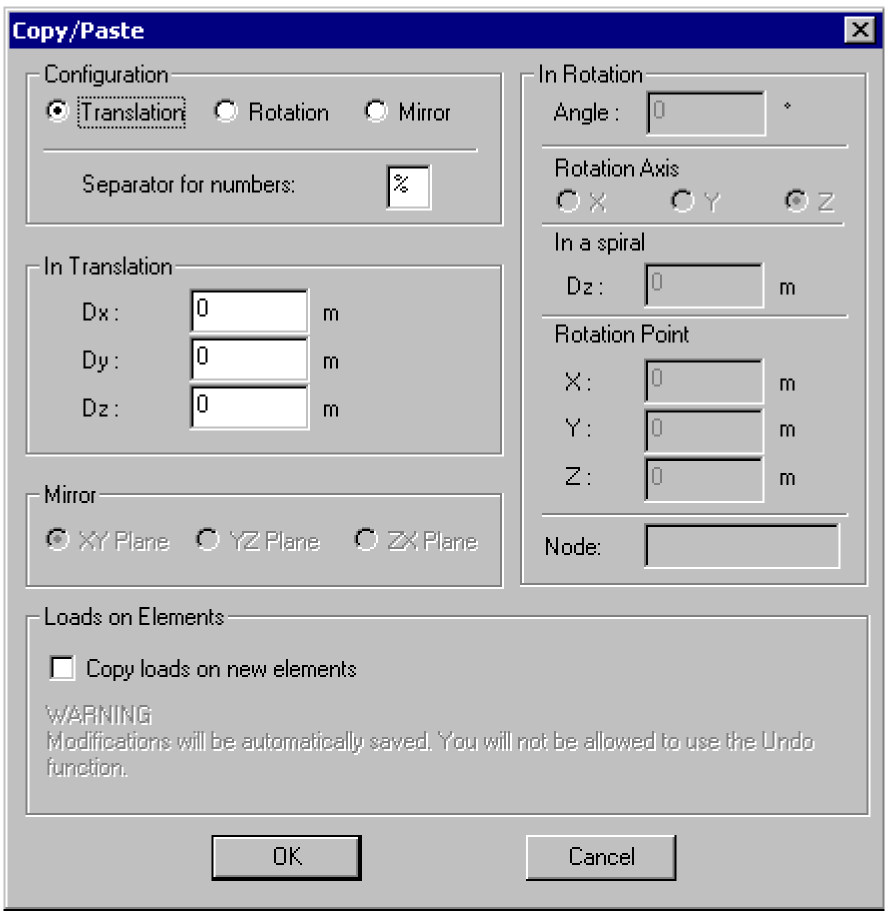
- Activate the "Structure" mode .
- Activate a type of element on Elements toolbar.
- Select the elements that you wish to delete.
- Do one of the following:
  - Click the icon on Edit toolbar.
  - Choose **Delete** from the **Edit** menu.
  - Press the [Delete] key.

Copy/Paste

Configuration of Copy/Paste

The "Copy/Paste" icon of Standard Toolbar

You may copy all kind of elements in different ways: Translation, Mirror, and Rotation. VisualDesign will create missing nodes and assign default numbers based on the separator specified in the **Copy/Paste Configuration** dialog box. This dialog box is called up through the **Copy** or **Copy/Paste Configuration** function of **Edit** menu or through the short-cut keys **[Ctrl] + C** (If elements are selected). Loads applied to elements can also be copied/pasted by activating the appropriate check box in the **Copy/Paste** dialog box.



It is also possible to copy and paste elements from another VisualDesign™ document. This is the purpose of the **Configure Copy/Paste** dialog box. This function enables you to keep a certain pasting configuration (in rotation, for example) for document A, and another one (in translation, for example) for document B.

Note that the translation deltas set in a document **Configure Copy/Paste** dialog box will be applied to the origin coordinates of the pasted elements. If the last pasted elements of the working session have been pasted in another document, different than the one that will receive the new copied elements, the translation deltas will be applied to the origin coordinates of the pasted elements in this other document.

VisualDesign™ enables you to undo and redo all pasting carried out. It keeps in memory the copied selection even after saving or after analysis have been carried out. However, it cannot be cancel if loads have been copied to new elements.

A copied selection will be erased from memory only when another selection will be prepared for pasting.

Note: The **Copy/Paste** function cannot be applied to members composed of a master node and more than one slave node. The members will be pasted but the dependence between nodes will not be preserved.

See also

Copy/Paste in Rotation

Copy/Paste in Translation

Copy/Paste Mirror

Copy/Paste Loads on New Elements

Copying and Pasting Elements

Copy/Paste Elements Between 2 VD Files

Copy/Paste in Rotation

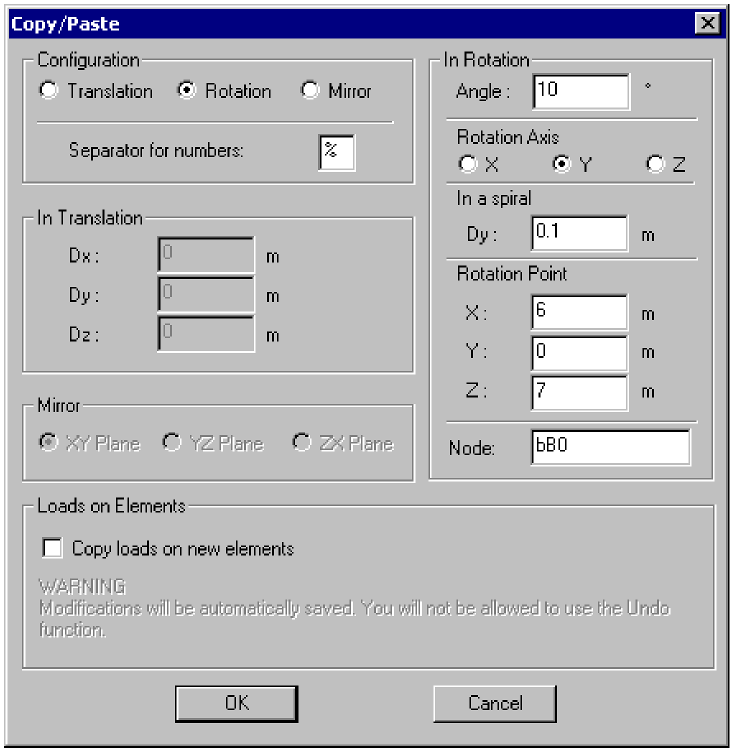
To apply a rotation to pasted elements, activate the "Rotation" radio button of *Configuration* zone in the **Copy/Paste Configuration** dialog box. You can also apply a rotation according to coordinates or even a node (you must enter the node number in the "Node" field).

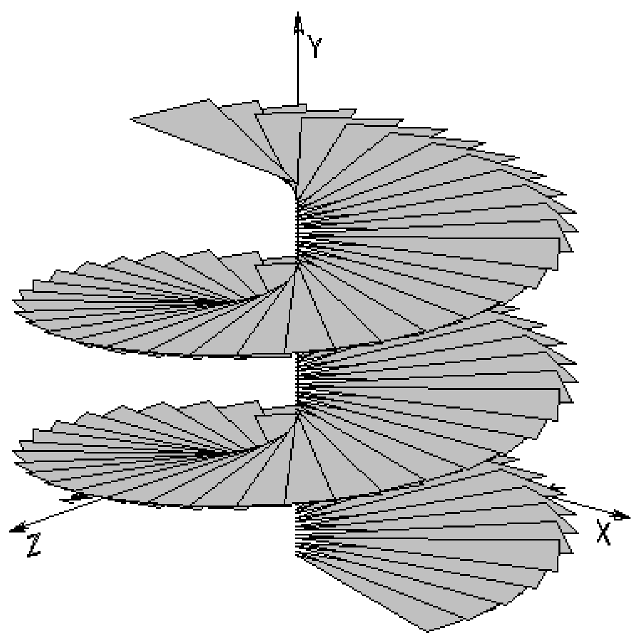
Then, enter the rotation angle and axis.

Copy/Paste in a Spiral:

Activate a rotation axis and enter delta Dx, Dy, or Dz in the zone "In a Spiral". Then, specify a rotation angle and enter the node number corresponding to the point of rotation.

Click OK. Use the short cut keys [**Ctrl**]**+V** the number of times that you want to paste elements using this configuration.





Remark The pasted elements will have the same beta angle as the copied members.

See also

Copying and Pasting Elements

Copy/Paste in Translation

To paste one or many selected elements in translation, activate the "Translation" radio button of the "Configuration" zone of the **Copy/Paste** dialog box. Set the displacement deltas in X, Y and Z in the appropriate boxes.

Click OK. Use the short cut keys [**Ctrl**]**+V** the number of times that you want to paste elements using this configuration.

Remark The pasted elements will have the same beta angle as the copied members.

See also

Copying and Pasting Elements

Copy/Paste Mirror

Copy mirror according to a selected plane by activating the "Mirror" radio button in the "Configuration" zone of the **Copy/Paste** dialog box. Then, specify the plane in which will be pasted the selected elements. Press OK.

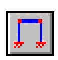
Use the shortcut keys [**Ctrl**]**+V** the number of time that you wish to paste these elements.

Remark The pasted elements will have the same beta angle as the copied members.

See also

Copying and Pasting Elements

Copying & Pasting Elements

- Activate the "Structure" activation mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the appropriate type of element on the Elements toolbar.
- Select elements that you wish to copy and paste by pressing the left mouse button.
- To copy the element, use the shortcut keys [**Ctrl**]**+C** or click . Then, the **Copy/Paste Configuration** dialog box will appear on the screen. Modify parameters if needed and press OK.
- To paste the copied elements, use the short-cut keys [**Ctrl**] **+ V** or click . Press down the shortcut keys [Ctrl] + V every time you wish to paste the copied elements (for the same parameters you defined in the dialog box).

- If you want to modify the configuration of the **Copy/Paste** function, do the following:
  - Select the element that you wish to copy and call up the **Copy/Paste Configuration** dialog box by pressing the [Ctrl] + C shortcut keys. Modify the parameters and click OK to close the dialog box.
  - Press the [Ctrl]+V shortcut keys every time you want to paste the copied element.

OR

  - Select the **Copy/Paste Configuration** function under the **Edit** menu.
  - Modify the parameters in the dialog box and press OK.
  - On your screen, select the element that you wish to copy and press the [Ctrl]+C short-cut keys. The **Copy/Paste** dialog box that appears on your screen includes similar parameters just entered. Press OK.
  - Press the [Ctrl]+V short cut keys every time you want to paste the copied element.

Paste

The "Paste" icon of Standard Toolbar

Use this function, available in **Edit** menu, to paste copied elements or items or use the shortcut keys [Ctrl] + V.

Copy/Paste Loads on Elements

When copying and pasting elements, loads can be copied and pasted too. To do so, activate the appropriate option in the **Copy/Paste** dialog box.

Warning! When the pasting of loaded elements is done, you will not be allowed to use the **Undo** function because VisualDesign will automatically save loads on elements.

See also

Copying and Pasting Elements

Copy/Paste Elements Between two Files

Elements will be pasted according to the copy/paste configuration of the second file. The source file configuration is not considered.

Loads can be copied and pasted in the second file if this option is activated in the **Copy/Paste** dialog box located in the second file.

If you want to copy and paste elements or a partial structure from one VisualDesign™ document into another VisualDesign™ document, some attributes will be lost in the second document.

When copying a **support**, the orientation node will be lost in the second document.

When copying a **member**, continuous systems will be lost in the second document.

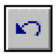
N. B. When using the **Copy/Paste** function between two files, use symbol "$" as a separator to keep the number and member numbers intact.

See also

Configuration of Copy/Paste
Copying and Pasting Elements

Undo and Redo

Undo



The "Undo" icon of Standard toolbar

If you change your mind or make a mistake during an operation, VisualDesign™ lets you undo the last action(s) you performed.

See also

Application of the Undo Function

Redo

Restrictions to the Undo function

Undoing Last Operation(s)

Application of the Undo Function

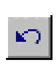
With VisualDesign™, most available functions of the **Edit** menu can be cancelled, such as **Properties**, **Add**, **Enlarge**, **Copy/Paste**, **Move**, **Translation**, **Rotate**, **Delete** and **Split** functions.

Restrictions to the Undo function

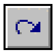
You may use the **Undo** function only in "Structure" activation mode. Furthermore, some actions will empty the modification contents and disable the **Undo** function:

- Saving the file (VisualDesign™ erases from its memory the prior stages to a saving);
- Automatic generation functions and analysis;
- Modifications made inside a spreadsheet if the OK button was pressed down.

Undoing Last Operation(s)

- Do one of the following:
  - Click the icon  on the Standard toolbar as many times as you wish to undo actions.
  - Choose **Undo** from **Edit** menu.
  - Use the [**Ctrl**]**+Z** shortcut.

Redo



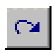
The "Redo" icon of Standard toolbar

If you wish to recover the operation(s) undone, simply click the **Redo** button the number of times you need.

See also

Undo

Redoing Cancelled Operation(s)

- Do one of the following:
  - Click the icon  on the Standard toolbar as many times as you wish to recover cancelled operations.
  - Choose **Redo** from **Edit** menu.
  - Use the [**Ctrl**]**+A** shortcut.

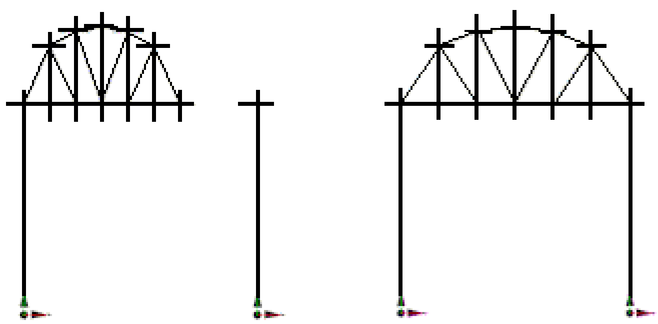
Note. The Redo function has the same limitations as the Undo function.

Move Elements

Enlarge

The **Enlarge** function of **Edit** menu allows moving selected nodes. The multiplication factors (x, y, and z) will be applied to the coordinates of selected node.

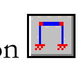
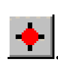
Zero point coordinates can be used as a reference point to enlarge selected nodes. In this case, the distance will be equal to the difference between the coordinates of selected nodes and the coordinates of the reference point.

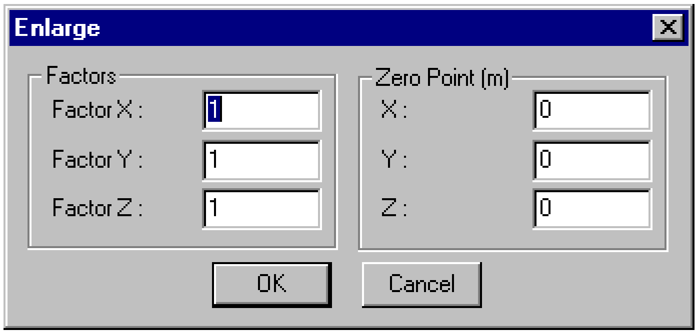


Before After

Trick: To get a mirror image of selected nodes, enter a factor or –1 for the axis that you want nodes to be copied mirror.

Enlarging a Part of the Structure

- Activate the "Structure" mode using one of the following procedures:
    - Select the "Structure" icon  of Activation toolbar.
    - Select **Activate Mode/Structure** from **Edit** menu.
- Activate the Node element .
- Select the nodes to be stretched.
- Select **Enlarge** from **Edit** menu.
- Do one of the following:
    - Enter zero point coordinates;
    - Enter enlargement factors for X, Y or Z-axis.



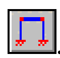
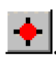
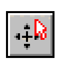
Move

The "Move" icon of Edit toolbar

The only element that you can move around is the node.

When you move a node, everything that is attached to it will be moved, too. You can move more than one node simultaneously. However, you may only move one attribute at a time.

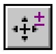
Moving one Node or More

- Activate the "Structure" mode .
- Activate the Node element .
- Select nodes to be moved.
- Do one of these procedures:
  - Click the icon  on Edit toolbar.
  - Choose **Move** from the **Edit** menu.

  The **Coordinates-at-Point-of-Origin** dialog box will appear, displaying the global axes allowing you to specify the point of origin.
- Validate the coordinates at the point of origin by pressing the left mouse button or by choosing "OK" in the **Coordinates-at-Point-of-Origin** dialog box.

  The **Coordinates-at-Insertion-Point** will now replace the **Coordinates-at-Point-of-Origin** dialog box and allow you to determine the insertion point for the selected node(s).
- Use the mouse to move the selected node(s).
- Press the left mouse button to select the new node location, or press OK in the **Coordinates-at-Insertion-Point** dialog box.

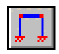
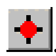
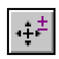
Translation



The "Translation" icon of Edit toolbar

The **Translation** function (**Edit** menu and Edit toolbar) allows translating a node by an amount delta (Δ). This function applies exclusively to nodes.

Translating a Node

- Activate the "Structure" mode by doing one of these procedures:
  - Click the icon  on Activation toolbar
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the Node element .
- Select the node(s) to be translated.
- Then, do one of the following:
  - Click the icon  on Edit toolbar.
  - Select **Translation** from **Edit** menu.
  - Use the shortcut keys [Ctrl]+B.
- Enter the displacement Delta (D) for the X, Y, and Z coordinates in the **Translation** dialog box.
- Press OK to validate your displacement Deltas (D) or Cancel to annul the operation.

Rotate

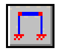
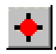
The "Rotate" icon of Edit toolbar

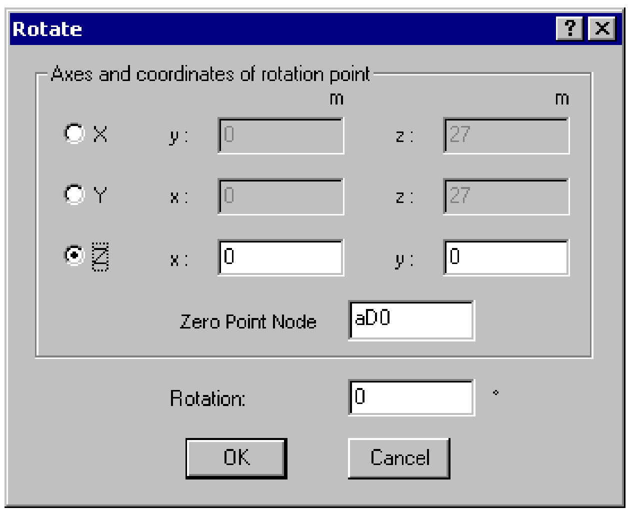
The **Rotate** function of **Edit** menu allows rotating nodes or members. A specific dialog box will either nodes or members are selected.

When nodes are selected, the **Rotate** function can be used to rotate a part of the structure around a selected axis, and from an orientation node and a rotation angle. Elements that are linked to nodes will also rotate.

When members are selected, the **Rotate** function allows modifying the beta angle of selected members according to a rotation angle and with respect to an orientation node, if a node number is specified.

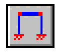
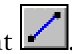
Rotating Nodes

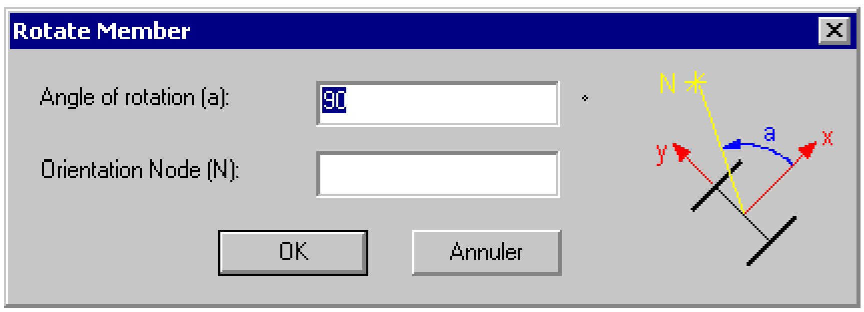
- Activate the "Structure" mode by doing one of these procedures:
    - Click the icon  on Activation toolbar
    - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the Node element .
- Select the nodes that will be rotated.
- Do one of the following:
    - Click the icon  on Edit toolbar.
    - Choose **Rotate** from the **Edit** menu.



- In the **Rotate** dialog box, specify the node coordinates from which the nodes will rotate. This node may be an existing node or another point in space. A node number is already assigned by default as zero point. This node corresponds to the first selected node, in order to speed up the editing if you need such a node.
- Determine the angle of rotation. A positive rotation angle means a counter clockwise rotation.
- Click OK to validate your choices or Cancel to annul the current operation.

Rotating Members

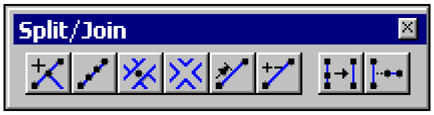
- Activate the "Structure" mode by doing one of these procedures:
  - Click the icon  on Activation toolbar
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the Member element .
- Select members which beta angle needs to be modified.
- Do one of the following:
  - Click the icon  on Edit toolbar.
  - Choose **Rotate** from the **Edit** menu.



- In the **Rotate** dialog box, modify the beta angle of selected member(s), enter the rotation angle (a), and the orientation node (N), if needed.
- Press OK to validate.

Split, Join & Connect

The Split/Join Toolbar



The "Split/Join" toolbar of VisualDesign Main Window

Split functions are available in the "Structure" activation mode to help you split members in different ways and split plates in multiple parts. When using split functions, the initial member end conditions will be copied to the newly created members and plates.

Members can be split at an exact position or according to one node position or more. Pin connections or rigid connections can be generated for crossing members also. The **Split** toolbar also includes a function that joins members or nodes and connects members or nodes.

An option is available for splitting loaded elements along with applied loads.

Restriction:

Functions **Split Multiple**, **Split at Exact Position**, and **Split according to Nodes(s)** cannot be used to split rigid extensions. We recommend not splitting rigid extensions.

Split functions can be used for the following elements:

FOR NODES

Join
Connect

FOR MEMBERS:

Split a Member on screen
Split a Member Evenly (Multiple)
Split Members and Create a Pin Connection
Split Members and Create a Rigid Connection
Split Members at Exact Position
Split Members according to Node(s) Position
Join
Connect

FOR PLATES:

Split Plates into Multiple Parts

See also

Split Functions and Member End Conditions

Split a Loaded Element

Loaded members and plates can be split with respect to the loads applied to original element. Loads will be copied and applied to new elements. However, the **Undo** function cannot be used because VisualDesign will automatically save information after the operation.

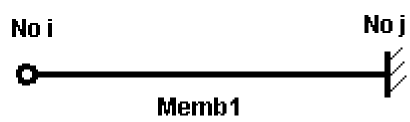
Split Functions and Member End Conditions

Case | **Before the split** | **After the split**

Case 1

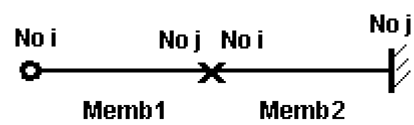
End Conditions

Before the split:



Member 1: pinned - fixed

After the split:

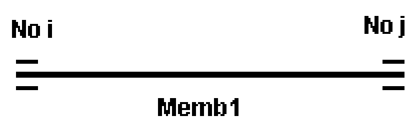


Member 1: pinned – fixed
Member 2: fixed – fixed

Case 2a

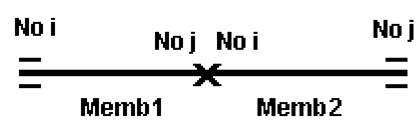
Lateral Support

Before the split:



Member 1: laterally supported at node i and j.

After the split:



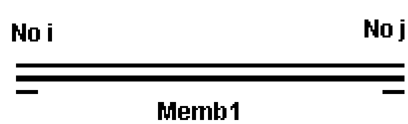
Member 1 is laterally supported at node i only.
Member 2 is laterally supported at node j only.

No lateral support is provided in the middle, at the new node.

Case 2b

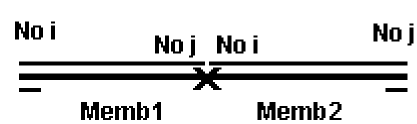
Lateral Support

Before the split:



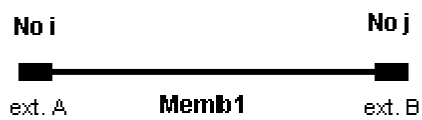
Member 1: A continuous lateral support is provided at superior fibre, inferior fibre is laterally supported at node i and j.

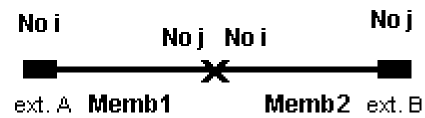
After the split:



Member 1: A continuous lateral support is provided at superior fibre and at node i at inferior fibre.
Member 2: A continuous lateral support is provided at superior fibre and at node j at inferior fibre.

<table>
<tr><th>Case</th><th>Before the split</th><th>After the split</th></tr>
<tr>
<td>Case 3

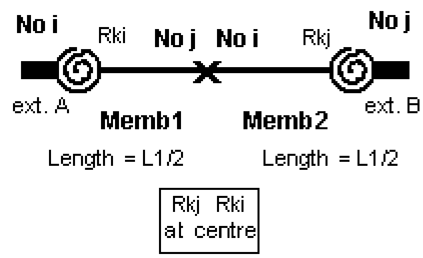
Rigid Extensions</td>
<td>

Member 1 with rigid extensions A and B at nodes i and j.</td>
<td>

Member 1: Rigid extension A at node i only.
Member 2: Rigid extension B at node j only.</td>
</tr>
<tr>
<td>Case 4

Rigid Extensions and semi-rigid connections Rki and Rkj at nodes i and j.</td>
<td>

Member 1 with rigid extensions A and B and semi-rigid connections Rki et Rkj at nodes i and j.
Where ki = Rki * EI/L

Ki is dependent of the member length.</td>
<td>

Member 1: Rki at node i is the same as it was for the original member. The value Rkj at the centre will be equal to half of the original Rkj:

Rkj = (kj*L1/2)/EI

Member 2: Value Rki in the middle will be equal to half of original Rki, and Rkj will be the same as it was for the original member.</td>
</tr>
</table>

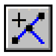
See also

Member Eccentricities and Rigid Extensions

Automatic Generation of Rigid Extensions

Split Members

Split a Member



The "Split a member" icon of Split toolbar

Use this function on the Split toolbar, while in the "Structure" activation mode, to graphically split a member on screen.

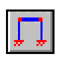
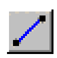
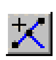
This function will retain all original member features such as end conditions, *beta* angle, lateral support parameters (as a fraction of the length), section type, and material.

See also

Split Function and Member End Conditions

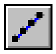
Split a Loaded Element

Splitting a Member on Screen

- Activate the **Structure** mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the *Member* element  on Elements toolbar.
- Then, do one of the following:
  - Click the icon  on Split toolbar.
  - Choose **Member** from **Edit/Split** menu.
- Click the member you want to split. When the mouse button will be released, a split node will appear on the member. Its distance from node i will be displayed perpendicularly to the member.
  - Glide the split node along the member with the mouse and press the left mouse button to freeze it in the desired position.
  - The **New Node** dialog box will appear on the screen so that you can specify its characteristics.
  - The **Split with Loads** dialog box will appear also. If you want to save and apply loads on the split members, activate appropriate box.

Note. The newly created member keeps the characteristics of the original member.

Multiple Split - Members



The "Multiple split" icon of Split Toolbar

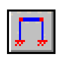
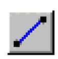
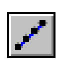
While working in the Structure activation mode, you are allowed to split up one or more members into segments of equal lengths, by selecting function **Multiple Split**. Specify the number of desired subdivisions in the **Multiple Split - Members** dialog box.

If members are guys (Tower Design module), check option **Automatic split of guy members** and VisualDesign will automatically divide guy members into appropriate lengths.

See also

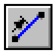
Automatic Split of Guy Members

Splitting Members Evenly in Multiple Parts

- Activate the **Structure** mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the Member icon  on Elements toolbar.
- Select members that you wish to split equally.
- Do one of the following:
  - Click the icon  on Split toolbar.
  - Choose **Multiple** from **Edit/Split** menu.
  - Use the short-cut keys **Ctrl+M**.
- Specify the number of desired subdivisions in the **Multiple Split** dialog box.
- The **Split with Loads** dialog box will appear also. If the original member was loaded and you want the loads to be saved and applied on smaller parts, activate appropriate box.

Note. Characteristics of original element are transferred in newly created members.

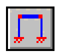
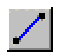
Split a Member at Exact Position

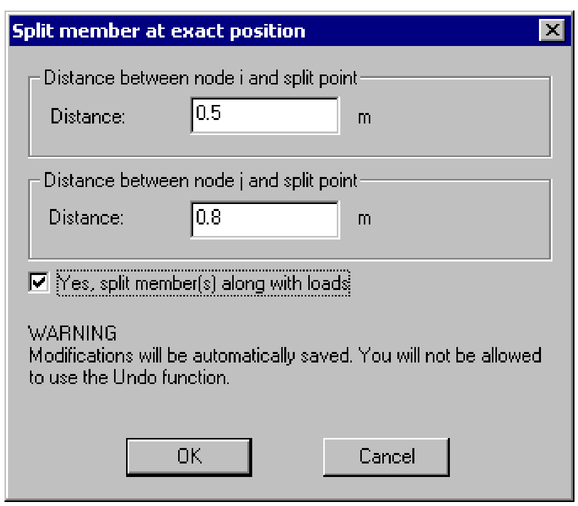


The "Split at exact position" icon of Split toolbar

Use this function to specify the exact position, from node i or node j, where you want to split a member.

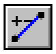
Splitting a Member at Exact Position

- Activate the "Structure" mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the *Member* element  and select the member on your screen.
- Do one of the following procedure:
  - Click the icon  on Edit toolbar.
  - Choose **Split at exact position** from **Edit/Split** menu.
  - Use the short cut keys **Ctrl+E**.
- The following dialog box will appear on the screen. In the example below, a member 1.3 m long will be split at 0.5 m from node i.



Note. Newly created members keep characteristics of the original member.

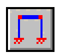
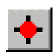
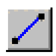
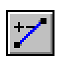
Split According to Node(s)

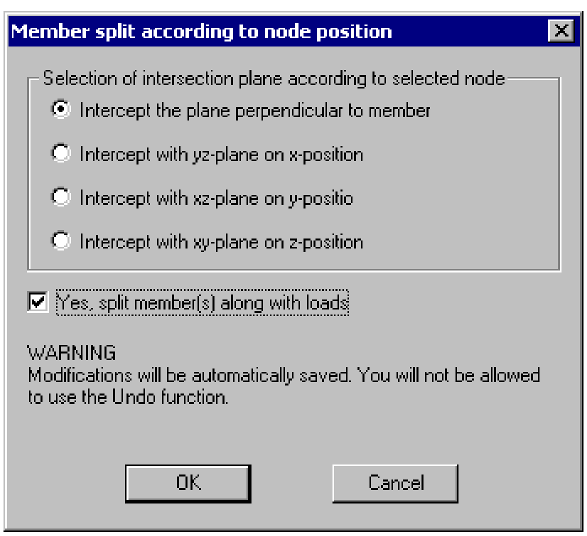


The "Split according to node" icon of Split Toolbar

The command **Split according to node(s)** allows splitting one or more members according to one or many nodes positions. A cutting plane, passing through the selected node(s), must also be selected. By default, a perpendicular plane will be chosen. Other planes are available, namely xy, yz, or zx cutting planes.

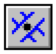
Splitting Member(s) According to Node(s)

- Activate the "Structure" mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the *Node* element  and select one node or more. Activate the Member element  and, while keeping down the [Ctrl] key, select a member or more. Then, do one of the following:
  - Click the icon  on Split toolbar.
  - Select function **Split with node** in **Edit/Split** menu.
  - Press the short cut keys [**Ctrl**]+**Y**.
- The following dialog box will appear. Choose an intersection plane passing through the node coordinates or use the default cutting plane.



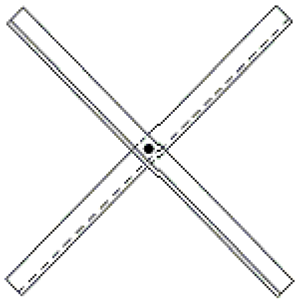
Remark. Newly created members keep characteristics of original member.

Generate Pin Connection

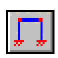
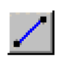
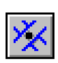


The "Generate pin connection" icon of the Split Toolbar

The command **Generate Pin Connection** allows splitting X-bracings at the point of intersection. A master node and dependant node is created at the intersection. The dependant node is dependant of the master node translation.

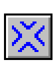


Generating a Pin Connection (Bracings)

- Activate the "Structure" mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the "Member" icon  on Elements toolbar.
- Select two braces while keeping the [Ctrl] key down.
- Click the icon  on Split toolbar.

Note. Newly created members keep characteristics of original member.

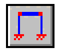
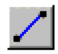
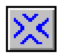
Generate Rigid Connections



The "Generate rigid connections" icon of Split Toolbar

In the Structure activation mode, use the function **Rigid Connections** in menu **Edit/ Split** to automatically generate rigid connections (welded) at the junction of crossing members.

Generating Rigid Connections

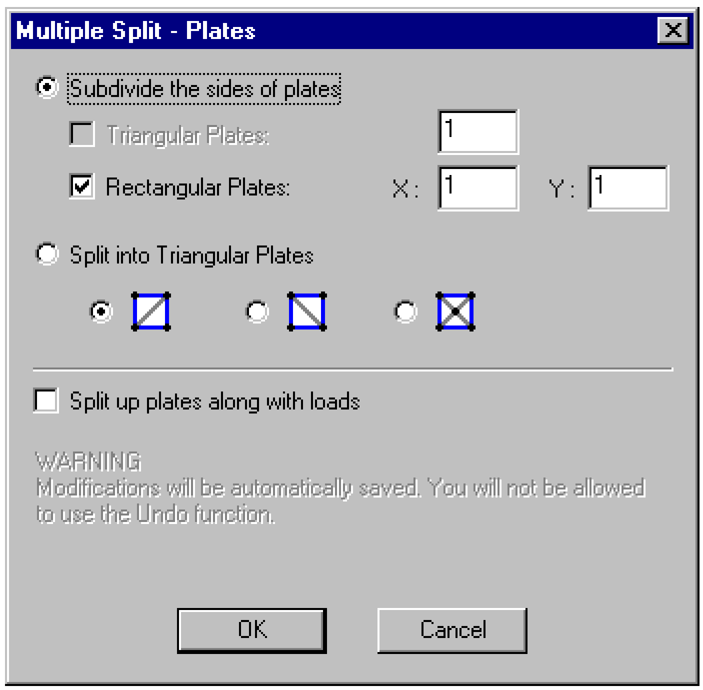
- Activate the "Structure" activation mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate the "Member" element  on Elements toolbar
- On your screen, select two or three crossing members.
- Do one of the following:
  - Click the icon  on Split toolbar.
  - Select **Rigid Connection** from **Edit/Split** menu.
  - Press short cut keys **Ctrl+R**.

Split Plates

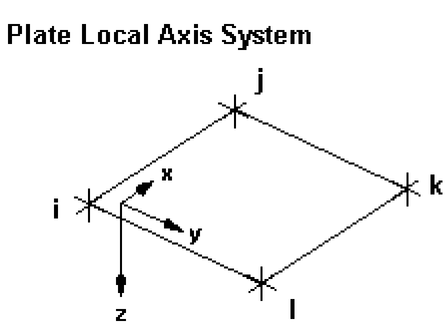
Multiple Split - Plates

The "Multiple split" icon of the Split Toolbar

When you are working in the *Structure* activation mode, you can graphically split rectangular or triangular plates by using the **Multiple** function of **Edit/Split** menu or by using short-cut keys [**Ctrl**]+**M**.



The **Multiple Split – Plates** dialog box allows splitting rectangular plates into rectangular or triangular plates. To split into rectangular plates, enter the number of subdivisions according to the plate local x and y directions, as shown below. Three configurations are available for splitting rectangular plates into triangular plates.

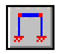
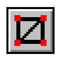
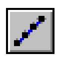


To split a triangular plate, specify the desired number of subdivisions in the **Multiple Split – Plates** dialog box.

SPLITTING LOADED PLATES

If plates are loaded, activate option "Split along with loads" in the **Multiple Split** dialog box.

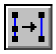
Splitting Plates Evenly in Multiple Parts

- Activate the **Structure** mode by doing one of the following:
  - Click the icon  on Activation toolbar.
  - Choose **Activate Mode/Structure** from **Edit** menu.
- Activate plate elements  on Elements toolbar.
- Select plate(s).
- Do one of the following:
  - Click the icon  on Split toolbar.
  - Go to **Edit / Split / Multiple**.
  - Use the short cut keys [**Ctrl**]+**M**.
- If you selected triangular plate(s), specify the number of side subdivisions in the **Multiple Split – Plates** dialog box.
- If you selected rectangular plate(s), specify the number of side subdivisions in the x and y directions, according to plate local axes system. To split into rectangular plates, select the split configuration.
- Activate the box "Split plates along with loads" if the original plate was loaded and if you want the load to be saved and applied on smaller parts.

Note. Newly created plates keep characteristics of original element.

Join & Connect

Join



The "Join" icon of Split toolbar

The function **Join**, used with the "Structure" mode, allows joining members or nodes.

Nodes

This function can be used to join two selected nodes. The first selected node will be the one that will be moved and merged to the second selected node

Members

Use this function to join co-linear (with a 1 degree tolerance) members into a single member. Intermediate nodes are automatically deleted. As soon as more than one member is selected, the "Join" button and menu option will be available.

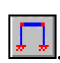
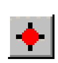
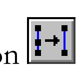
If selected members have different shapes, VisualDesign will use the shape assigned to the first selected member and assign it to all other members.

If an element is attached to an intermediate node located between selected members, VisualDesign will not join members surrounding this node. For example, if a beam is connected to the second node among five selected co-linear members, VisualDesign will join three members only.

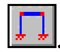
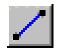
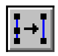
See also

Joining Members
Joining Nodes
Split Functions and Member End Conditions
Split a Loaded Element

Joining Nodes

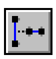
- Activate the "Structure" mode .
- Activate the "Node" icon  on Elements toolbar.
- Select node, while keeping the control key [Ctrl] down. The first selected node will be the one that will be moved and merged
- Follow one of these procedures:
  - Click the icon  on Split toolbar.
  - Choose the function **Join** under **Edit** menu.

Joining Members

- Activate the "Structure" mode .
- Activate the "Member" icon  on Elements toolbar.
- Select members that you want to join: keep the control key [Ctrl] down while you click on each member.
- To join members, follow one of these procedures:
  - Click the icon  on Split toolbar.
  - Choose the function **Join** under **Edit** menu.

Note. The newly created member will have the characteristics of the first selected member.

Connect



The "Connect" icon of Split toolbar

The **Connect** function is available in **Edit** menu if the Structure mode is activated. It allows connecting selected members by stretching some of them and it allows connecting selected nodes through rigid links.

Nodes

The function used with nodes will create rigid link between them. The last selected node will be the master node. All others selected nodes will be slave nodes linked to this master node.

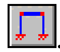
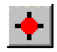
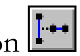
Members

This new function (**Edition** menu) connects two members or more that are separated from a distance. The function will lengthen members and moves their end node if no transverse element (member, plate, or floor) is connected to these nodes. The first selected members will be stretched.

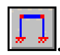
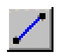
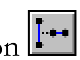
See also

Connecting Nodes (Rigid Links)
Connecting Members

Connecting Nodes

- Activate the "Structure" mode .
- Activate the "Node" icon  on Elements toolbar.
- Select two nodes while keeping the [Ctrl] key down.
- Follow one of these procedures:
  - Click the **Connect** icon  on **Split / Join** toolbar.
  - Select the **Connect** function in **Edit** menu.

Connecting Members

- Activate the "Structure" mode .
- Activate the "Member" icon  on Elements toolbar.
- Select two members while keeping the [Ctrl] key down. The first selected member will be the one that will be stretched.
- Follow one of these procedures:
  - Click the **Connect** icon  on **Split / Join** toolbar.
  - Select the **Connect** function in **Edit** menu.

Search for Elements

Find

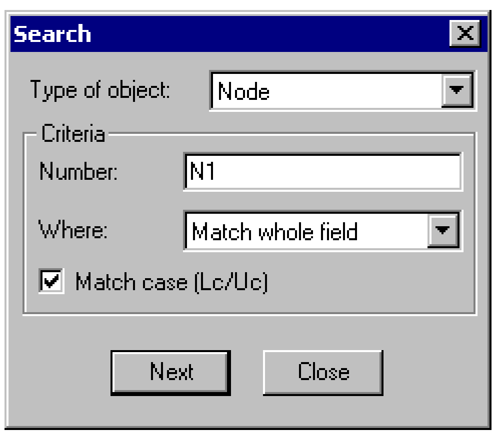
The "Find" icon of Edit toolbar

The **Find** function will circle on-screen the element(s), (e.g. member, node etc.) which correspond(s) to the search criteria. The search criterion is composed of type of element and element alphanumerical numbers. The search operation considers the position of characters (beginning, end, or anywhere in the number) as well as upper- and lower-case, if applicable.

The **Find** function is available in all activation modes (Structure, Load Case, Load Combination, Envelope, Vibration modes, and Design Results).

Finding an Element

- Do one of the following:
  - Click the icon  on Edit toolbar.
  - Choose **Find** from the **Edit** menu.



- In the **Search** dialog box, select a type of element in the drop-down list box.
- Enter the full alphanumerical number of the element that you are looking for, or simply a part of this number.
- Specify whether the entered number matches the whole field, or corresponds to the beginning or end, or anywhere in the field.
- Indicate whether upper- or lower-case characters apply.
- Press the "Next" button to launch the search operation.

The elements that correspond to all the search criteria will be encircled on-screen every time you press "Next" until the search operation is completed and there are no more elements of the requested category. Do not close the dialog box if you want to zoom-in, zoom out or to change the view because the coloured circle will disappear.

- To exit the dialog box, press the "Close" button. The last element found will remain on the screen.

Practical Example: 2D Frame

2D Frame

- Start VisualDesign™. Click on the **New Project** icon . The Structure activation mode is automatically activated.

- Select the **Preferences** tab of **Project Configuration** (**File** menu) and uncheck all the boxes included under the heading "Display Dialog Boxes".

Nodes

- Create nodes through the **Nodes** spreadsheet (**Structure** menu). Specify nodes 1 and 4 as support nodes: Double click in the "Type" column and select option *Support*.

Nodes Spreadsheet

4	Number	Type	Coord. X m	Coord. Y m	Coord. Z m	ID Master No.	Linked DO
1	1	Support	0.00	0.00	0.00	0	n/a
2	2	Normal	0.00	3.00	0.00	0	n/a
3	3	Normal	6.00	3.00	0.00	0	n/a
4	4	Support	6.00	0.00	0.00	0	n/a

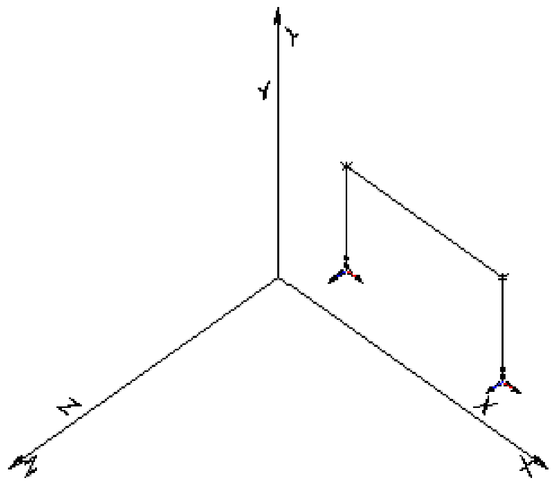
Members

- Select the **Member** icon on the Elements toolbar and press the **Add** icon on the Cursor toolbar. To create a member, click on a node (origin node, i) and on a second one (end node, j). Do the same to create the other members. When elements are created, select the extended selection mode to exit the **Add** mode.

Supports

- To create supports, activate the **Node** element on the Elements toolbar and select all nodes at the bottom of the structure. Press the **Properties** icon . In the **Node Characteristics** dialog box, choose option *Support* in the "Type of node" field. (New supports have restrained displacements and rotations by default values.) To look at support restraints, activate the **Support** icon , double-click on a support node and select the Support tab in the dialog box.

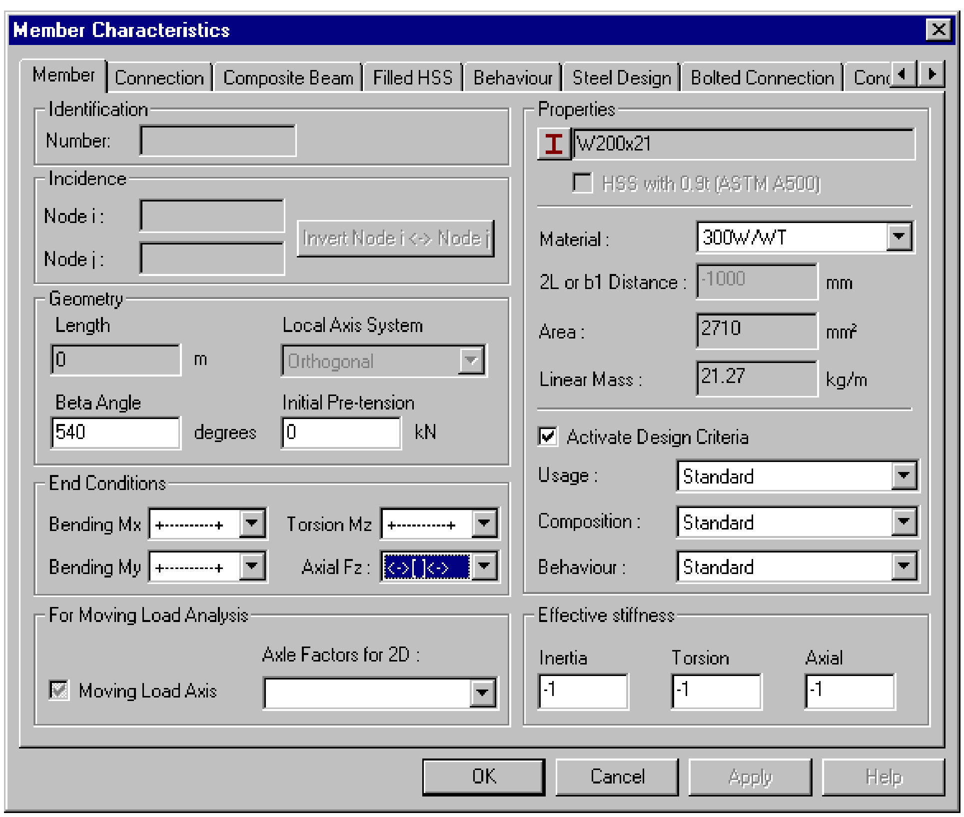
- Press the control key [Pg Up] on your keyboard to get an isometric view of the structure.



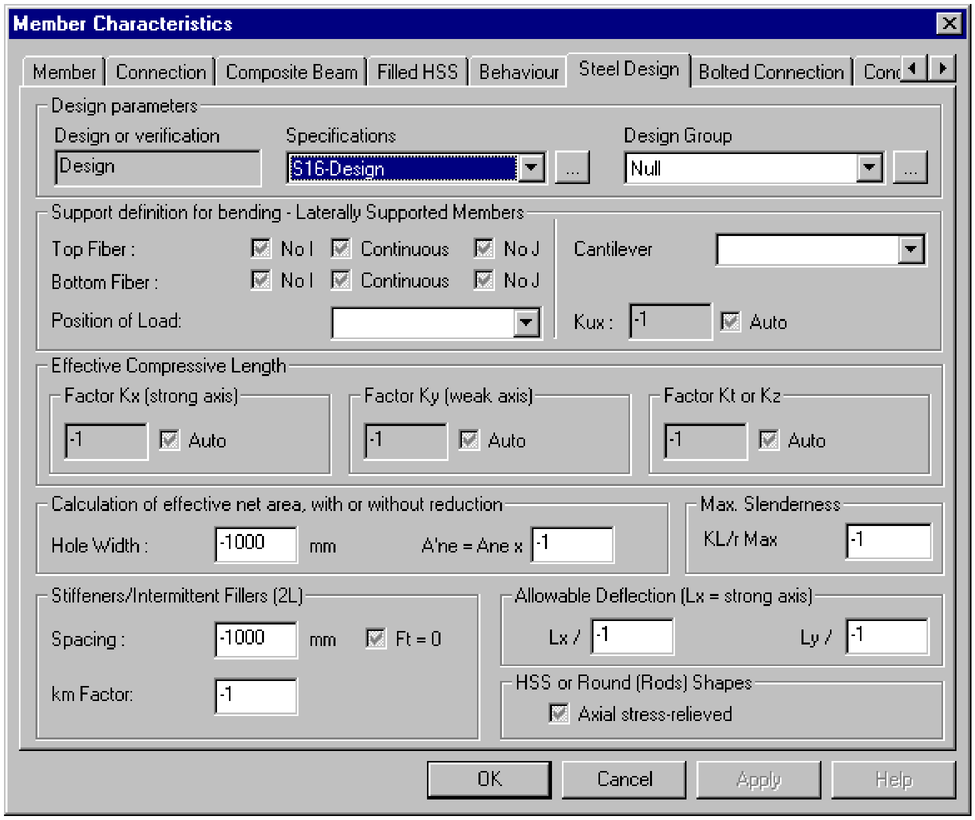
- Save your file. To do so, press the **Save** icon . Give a name to your project and choose a directory.

Definition of Members

- Activate the **Member** icon and select all three members. Press the **Properties** icon . Complete the parameters in the **Member** tab.



- Choose a steel specification in the **Steel Design** tab and press OK.

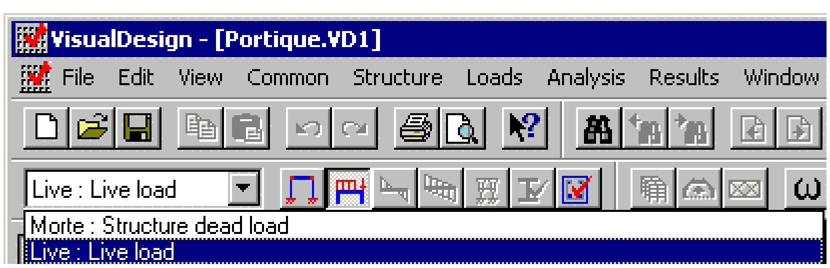


Load Cases

- Give a title to each load case. To do so, go in the **Loads** menu and select the heading **Load Cases/Definition**.

- In the **Loads Definition** spreadsheet, insert a line (To insert a line, select the line number 2 and press down the [Insert] key.) Double-click in the "Type" cell and choose a "Live" type of load. Double-click in the "Number" cell and give it a name. Press OK.

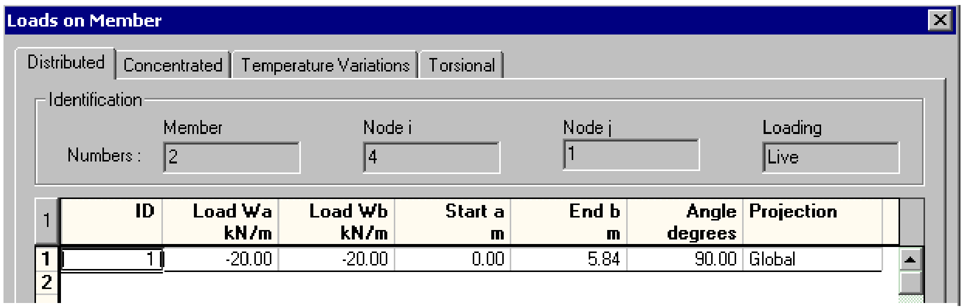
- Activate the Load Case activation mode and choose the Live load case. At the message: "Do you wish to save your project?", answer yes. The name of the load case will be written at the bottom of your screen.



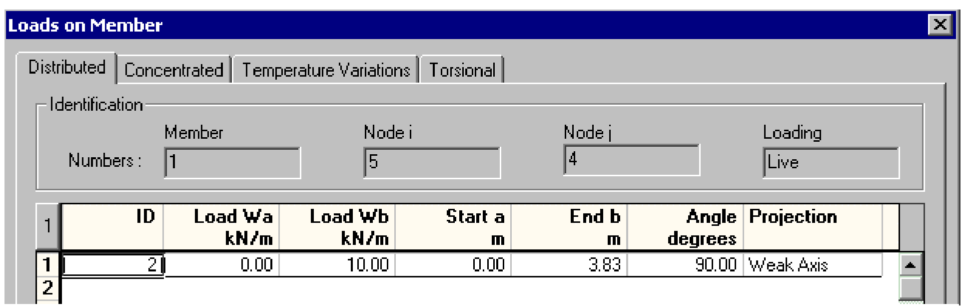
Applying Loads

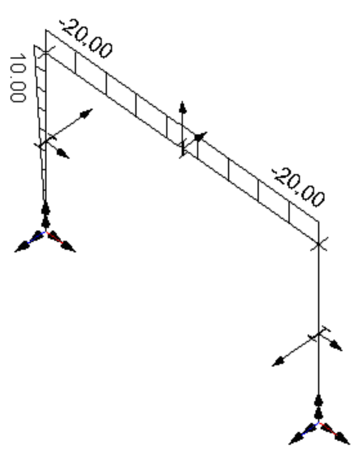
You are now ready to enter loads on the structure. We are going to place a distributed load on the horizontal member and on a vertical member. A punctual load will also be applied to a node.

- Double-click on the horizontal member. In the **Loads on Member** dialog box, click on the number 1, at the first line and insert a line. Double-click in the cell "Load Wa" and enter –20. Double-click in the cell "Load Wb" and enter –20. Press OK. You will see the loads applied to the member.

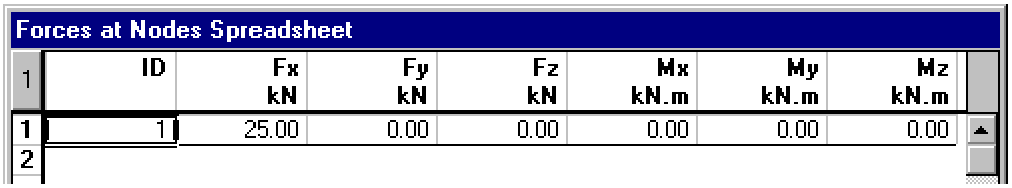


- Double-click on the vertical left member. In this case, the load will be applied on the weak axis, at 90 degrees. Load Wa is applied at node i and Wb, at node j. Loads are positive because their orientation is positive relatively to the member weak axis, as shown below. Display the shape of steel section through the **Attributes** tab of **View Options** dialog box.





- To apply the punctual load on node, activate the **Node** icon and double-click on the node. Enter a positive load of 25 kN in the global x direction. Click on Ok.

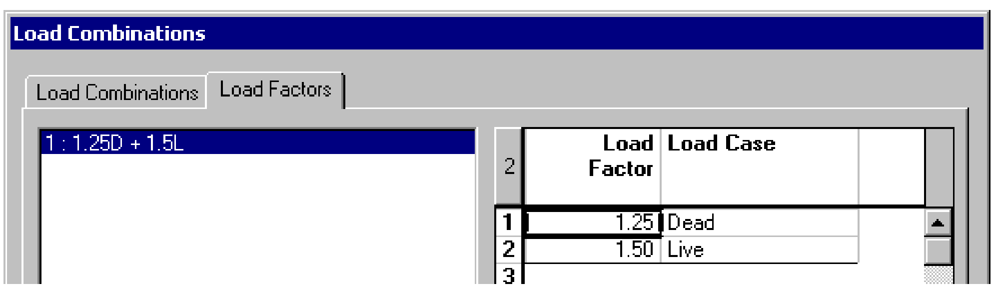


Now, at least one load combination must be defined to run an analysis or a design.

Load Combination

- Go in the **Loads** menu and select the heading **Load Combinations / Definition**. Insert a line in the spreadsheet. Double-click in the "Number" cell, give it a number or name, and do the same for the description.

- Select the **Load Factors** tab. Insert two lines in the right section of the dialog box. In the first line, double click in the "Load" column, choose the Dead load case, and enter 1.25 as load factor. Do the same for the second line: choose the live load case and enter 1.5. Click on OK.

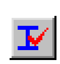


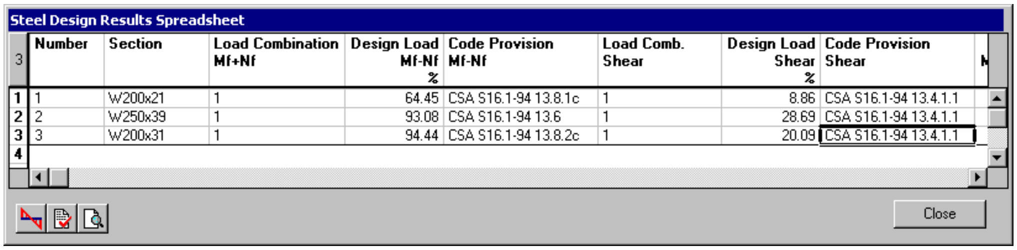
You are now ready to launch the analysis and steel design.

Analysis

- Press the icon **Analysis and Design**  on the Tools toolbar. The **Design** dialog box will be displayed on the screen. It is written that three members will be optimized according to shape area. Press the button "Analyse". When results will be available, close the dialog box.

Results

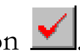
You will notice that the "Design results" icon  is automatically activated once that the design is done. Select all the members and then, select the heading **Structure Design / Steel** in the **Results** menu.



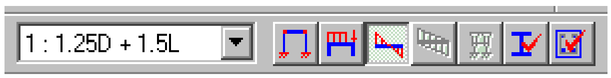
Steel Design Results Spreadsheet

3	Number	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear	Design Load Shear %	Code Provision Shear
1	1	W200x21	1	64.45	CSA S16.1-94 13.8.1c	1	8.86	CSA S16.1-94 13.4.1.1
2	2	W250x39	1	93.08	CSA S16.1-94 13.6	1	28.69	CSA S16.1-94 13.4.1.1
3	3	W200x31	1	94.44	CSA S16.1-94 13.8.2c	1	20.09	CSA S16.1-94 13.4.1.1
4								

Close

- Call the **View Options** dialog box by pressing the icon . Display the members design load by checking the "Design load" box in the **Results** tab.

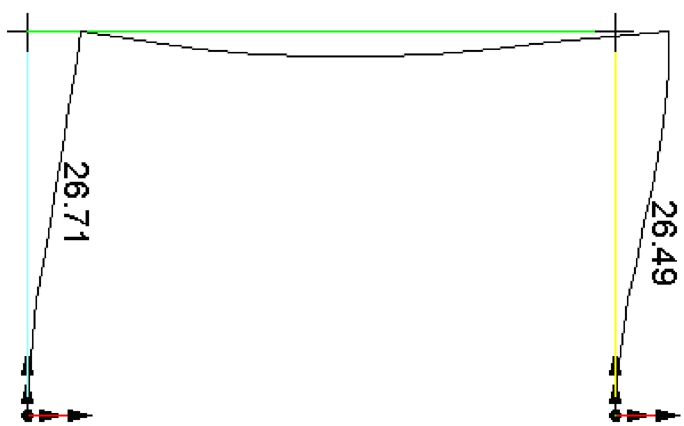
- To look at forces and deflection diagrams, you must activate the Load Combination mode beforehand and choose a load combination title in the drop-down list box of Activation toolbar.



- Then, in the **Results** tab of **View Options**, check the "Deflection" box or another type of results that you want to look at.

To adjust the amplitude of diagram and to display numerical values, use the **Diagrams** toolbar functions.

- You can also display numerical values of forces and deflections diagrams by ticking off the appropriate boxes below the "Numerical" title.



Checking your model

Verification of your Structural Model – Steps

Use this topic as a guide to verify your structural model before and after an analysis if a problem occurs.

First, complete the *General* tab (**Project Configuration**) to distinguish your current model from others (previous versions).

Before an analysis:

1. PROJECT CONFIGURATION

- Project title and description;
- Type of analysis. If non-linear: convergence criterion, number of iterations, etc.;
- Parameters for spectral and time history analysis: Za, Zv, R, U, I, F, Spectrum (code), Total height Hn, Number of stories, etc.
- Units for materials, foundations, structure, loads and results.

2. MATERIALS AND SHAPES

- Make sure that new materials / sections parameters are well defined;
- E, G, Rho (density);
- Ix, Iy, J.

3. GEOMETRY (NODES COORDINATES)

- Location of columns;
- Elevation of floors.

4. STRUCTURAL MODEL

- If a modal analysis is planned, try to model the structure as roughly as you can in order not to create local modes of vibration created by mezzanine or footbridge. Transfer the dead load of such structures to nodes.
- Members: assign shape and material, end conditions, beta angle, pretension, if necessary, release, specifications and design group. For a design, activate design criteria.
- Design: member lateral supports: do not forget to specify a continuous support along a member top fibre if the floor acts this way.

- Important: if cantilevers are present in your structural model, read carefully the On-line Help because the user must specify some cases himself.
- Plates: material, thickness, group of plates.
- Supports: displacements and moment's degrees of freedom, orientation, release. Do not forget that VisualDesign is a 3D software. If you have a 2D model, you must fix Rz and rotation Mx (or My depending on your principal axes). If you do not, a "Null pivot" warning will be posted on your screen (invalid stiffness matrix).
- Floors: One-Way, Two-way, joist, diaphragm action.

5. LOAD CASES

- Dead: member dead load (automatic), floors, plates;
- Extra dead loads: floors or mechanical equipments;
- Live loads: Overload, wind (2 directions), snow on roof, seismic load, temperature, etc.;
- Differential settlements on supports.

6. LOAD COMBINATIONS

- Load Combinations Generation Wizard;
- "Mass" load combination for modal analysis.

7. MODAL ANALYSIS

- Frequencies and vibration modes;
- Seismic directions, Ds (width of lateral resisting system for braced bays or rigid frame);

8. SPECTRAL ANALYSIS

- Percentage of modal (weight) participation for the two main directions (90% according to CNB)
- Information on Levels (stories), check interstory sliding;
- Shear forces and overturning moment.

See also

A Few Tips
FAQ
Modeling Strategy
Structural Modeling
Incomplete Modeling

If there is a problem after the analysis

You got a "Null pivot" message? It means that your structure is not stable. Read this topic: Null pivot in the stiffness matrix

Do not forget that at least one degree of freedom must be active at one node. For example, if there are many beams attached to one column, do not model hinges everywhere because a mechanism will be created in your model.

Make sure that shapes are appropriate (Filled HSS, built-up section properties in the spreadsheet).

Do members rotate around themselves? (Check their local z-axis).

Make sure that continuous members are co-linear. To avoid this problem, model a long member and split it into pieces.

Use functions **Mask** and **Zoom+** to look at certain part of your structure.

Use function **Overlapped Elements** (**Structure** / **Tools**) to detect overlapped elements and delete them.

Use the **View Options** dialog box.

Look carefully at inputs by using the sort function for spreadsheet.

Not sure about results? Check the following:

Check the message about analysis in the **Summary** spreadsheet (**Results** / **Load Combinations**). If some load combination analyses have not converged, you cannot use these results because they are not correct. Change the convergence parameters in **Project Configuration**.

Look at nodes displacements. Some displacements could be excessive. Convergence may be the cause.

For a steel design, check member end conditions and lateral supports. These are very important for a design. Display them through the **View Options** dialog box.

For a concrete design and prestressed concrete design: Did you model rigid extensions? If not, results are not accurate. Go to **Connection** tab (**Member Characteristics** dialog box) and specify minimum rigid extensions in the field "ez".

Are loads too large? VisualDesign will choose the biggest available steel section among the type that you specified in the steel specification (W, WWF, C, L, etc.) if it cannot find any for the loads acting on members. Check the loads, modify the steel specification and launch a new design.

See also

Technical Support

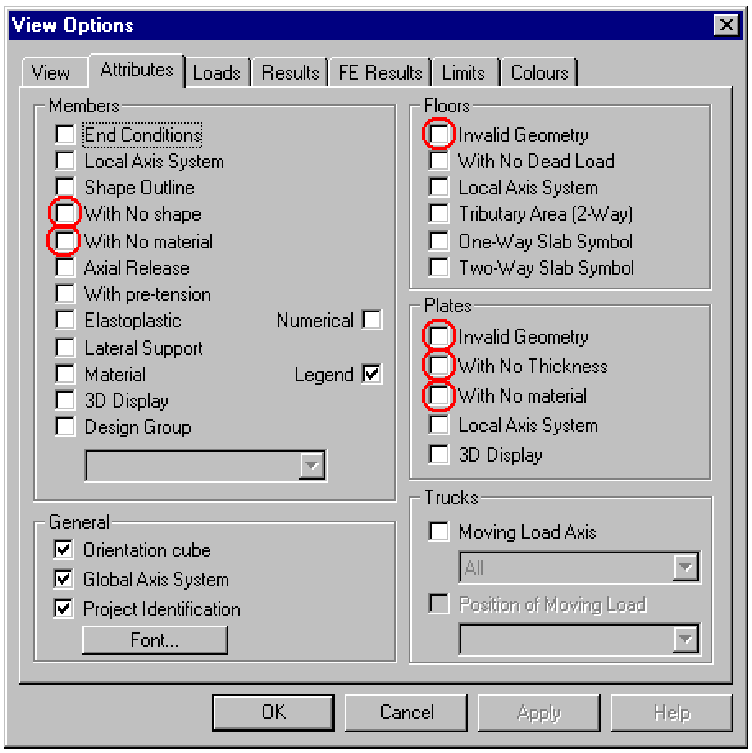
Sorting a spreadsheet

Inactive Nodes
Display Member Characteristics
Display Plate Characteristics
Display Floor Characteristics

Incomplete Modeling

This new tool, which is available in **Edit** menu / **Select Elements**, selects elements with incomplete or inadequate modeling such as members with no assigned shape or material, floors and plates with invalid geometry and plates with no assigned material or thickness.

In fact, this tool activates the following options that are included in the **View Options**' **Attributes** tab:



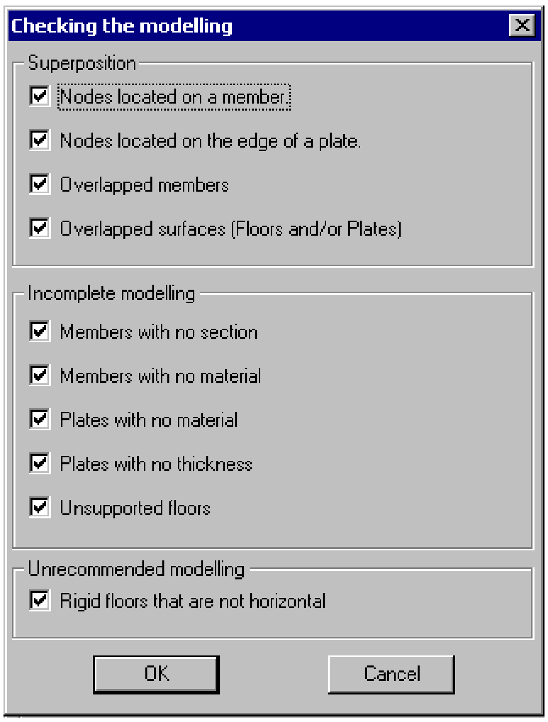
When invalid elements are selected using this function, click on each one in the Structure activation mode or select many of the same type and call up the default spreadsheet by pressing the shortcut keys **[Ctrl]+H**. The appropriate spreadsheet (members, Floors or Plates) will be displayed on screen. Complete the missing information such as materials, shapes or thickness).

Concerning invalid geometry for plates and floors, VisualDesign will notice them only if an analysis is launched. A warning message will appear on screen to inform you that it detected invalid geometries. Then, use the function **Incomplete Modeling** to locate and select them. Correct node coordinates for these surfaces.

Checking the Model

Before launching an analysis, use function **Checking the Model** in the **Structure/Tools** menu to look at missing material and sections. This function also searches for overlapped elements (members, plates or floors) and nodes that are not linked to edges of plates or members but are located on these elements.

Activate options in the dialog box and press OK.



When the search will be over, incomplete modeling and overlapped elements will be selected (highlighted) on screen. Use the **Mask** function to hide other elements and correct your model.

Checking Nodes

Inactive Nodes

An inactive node is not linked to anything. We recommend that you delete inactive nodes that may be present in your structural model.

VisualDesign warns you that there are inactive nodes in your project when you launch a static analysis or a design. It is written in the **Analysis** or **Design** dialog box.

To find and delete these nodes, use the function **Select/Inactive Nodes** in **Edit** menu. Then, select them and press the [Delete] key.

Checking Members

Displaying Members' Shapes

- Select the **Attributes** tab of **View Options** dialog box and activate the "Shape outline" option in the Member section of this tab.

  For a definition of the shape symbol, go to **Member** tab of **Member Characteristics** dialog box and open the *Shape* selection tree, which is accessible through the I-Beam icon posted in the member tab.

- To display the shape on the whole (or fraction) length, activate option *3D Display*. Refer to the *Preferences* tab (**Project Configuration**) to adjust the fraction of the member length that is not displayed to avoid hiding member end conditions.

Displaying Members End Conditions

- Activate the "End Conditions" option available in the **Attributes** tab of **View Options** dialog box. The direction of the swivel for the strong and weak axes will be displayed at member ends.

- To modify the colours for swivel points, go to the **Colours** tab, expand the *Structure* root and select *Member / End conditions*).

Displaying Members with Axial Release

- Go to the **Attributes** tab of **View Options** dialog box and activate the *With axial release* option. Members in **Tension-Only** (<-[]->) or **Compression-Only** (->[]<-) will be highlighted on screen.

- Modify the colour through the **Colours** tab, by clicking the "invalid elements" colour.

Displaying Members with Pre-tension

- Go to the **Attributes** tab of **View Options** dialog box and activate the option *With pre-tension* in the Member section.

Displaying Lateral Supports for Members

- Go to the **Attributes** tab of the **View Options** dialog box and activate the option *Lateral Support* in the Member section.

Displaying Members Having no Material

- Go to the **Attributes** tab of **View Options** dialog box and activate the *No Material* option. Members with no material will be highlighted on screen.

- To assign a material, press the **Properties** function. In the **Member** dialog box (**Member** tab), complete the missing information.

Note. The analysis cannot be performed if one member has no material defined.

Displaying Members Having No Section

- Go to the **Attributes** tab of **View Options** dialog box and activate the *No Section* option. Members with no specified section will be highlighted on screen.
- To assign them a material, click on the **Properties** function. In the **Member** dialog box, go to the **Member** tab and complete the required information.

Note. The analysis cannot be performed if one or more members are without section.

Displaying Overlapped Elements

This function allows users to seek for overlapped elements (members, plates and floors) and nodes that are not linked to members or edges of plates.

PROCEDURE:

Activate the "Structure" mode before selecting this function. Then, select this function located in the **Structure/Tools** menu.

Activate search options in the **Overlapped Elements** dialog box. VisualDesign will post messages about the results of the search.

Link the "free" nodes to appropriate elements and delete overlapped elements.

Checking Plates

Displaying Geometrically Invalid Plates

- Launch a static analysis, which allows VisualDesign to detect invalid geometries (one node or more is out of plane).
- Go to the **Attributes** tab of **View Options** dialog box and activate the option *Invalid Geometry* in the Plates section.

Note. The analysis cannot be performed if one plate is geometrically invalid.

Displaying Plates Having no Material

- Go to the **Attributes** tab of **View Options** dialog box and activate option *With no Material*. The plates having no material will be highlighted on screen.
- To correct this situation, call up the **Plate Characteristics** dialog box by pressing the **Properties** function. Enter a material.

Note. The analysis cannot be performed if a material is missing for one plate.

Displaying Plates with Unspecified Thickness

- Select the **Attributes** tab of **View Options** dialog box and activate the option *With no Thickness*. Invalid plates will be highlighted on screen.

- To correct this situation, call up the **Plate Characteristics** dialog box by pressing the **Properties** function. Enter a plate thickness.

Note. The analysis cannot be performed if one or more plates lack thickness.

Displaying Overlapped Elements

This function allows users to seek for overlapped elements (members, plates and floors) and nodes that are not linked to members or edges of plates.

PROCEDURE:

Activate the "Structure" mode before selecting this function. Then, select this function located in the **Structure/Tools** menu.

Activate search options in the **Overlapped Elements** dialog box. VisualDesign will post messages about the results of the search.

Link the "free" nodes to appropriate elements and delete overlapped elements.

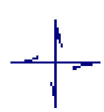
Checking Floors

Displaying Floors Orientation

View the orientation of both one-way slabs and two-way slabs by doing as follows:

- Go to the **Attributes** tab of **View Options** dialog box and activate option *One-Way slab* or *Two way Slab* in the "Floors" section.

The symbols represent the following directions:

Direction	Symbol
One-Way Slab	
Two-Way Slab	

Displaying Two-Way Slab Tributary Surfaces

- Go to the **Attributes** tab of **View Options** dialog box and activate the (Two-way Slab) *Tributary* option in the "Floors" section.

Displaying Geometrically Invalid Floors

- Launch a static analysis to allow VisualDesign detecting invalid geometries.
- To identify invalid floors, which have at least one node out of plane, go to the **Attributes** tab (**View Options**) and activate the *Invalid Geometry* option in the "Floors" section. Invalid floors will be highlighted.
- Click on an invalid floor and open the **Floors** spreadsheet (use shortcut keys [Ctrl] + H). Correct node coordinates.

Note. The analysis cannot be performed if one or more floors are geometrically invalid.

Displaying Overlapped Elements

This function allows users to seek for overlapped elements (members, plates and floors) and nodes that are not linked to members or edges of plates.

PROCEDURE:

Activate the "Structure" mode before selecting this function. Then, select this function located in the **Structure/Tools** menu.

Activate search options in the **Overlapped Elements** dialog box. VisualDesign will post messages about the results of the search.

Link the "free" nodes to appropriate elements and delete overlapped elements.

Chapter

4

LOADS, LOAD COMBINATIONS & ENVELOPES

TABLE OF CONTENTS

Chapter 4 Loads, Load Combinations & Envelopes

General

Loads

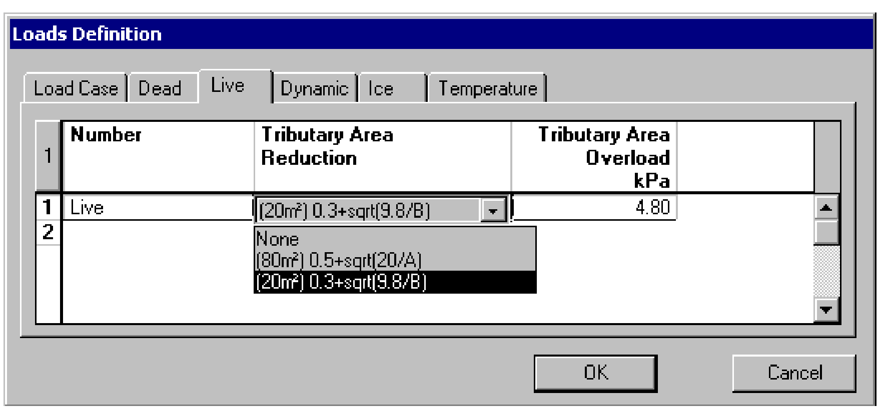
VisualDesign™ allows you to define and apply loads on nodes, members and floors, apply pressure and temperature variation on triangular and rectangular plates and define support settlements. The loads may be applied graphically with the help of dialog boxes or numerically through spreadsheets.

In this chapter, you will find the following headings among others:

Load Definition Spreadsheet
Loads on nodes
Support Settlements
Loads on Members
Loads on Floors
Loads on Plates Dialog Box
Reduction of tributary surfaces
Automatic Generation of Ice Loads
Automatic Generation of Wind Loads
Snow Loads
Load Types
Display convention of Loads on Floors and Members
Defining and Applying Loads on Elements
Modifying Applied Loads
Deleting Loads Applied on Elements

Reduction of Tributary Area

VisualDesign allows considering a reduction of tributary areas for "Live" type of loads, as per CNBC. Reduction may be of type A or B.



VisualDesign™ calculates the "real" tributary area by finding the location where the Vf diagram of live load is equal to zero. The live load case must be defined in the **Live** tab of **Loads Definition** dialog box. The columns included in this tab are optional but must be filled in to reduce the floor overloads.

All the applied loads that are corresponding to this particular "Live" type of load must be of the same type of reduction (A or B) and of the same magnitude (2.4 kPa, 4.8kPa, etc.).

The delimitation of tributary area surrounding an element is at Vf = 0. Unlike the code, VisualDesign™ allows reducing overloads even if the structure has an irregular geometry such as variable spans, elements with different inertias, etc. If you want to check by hand the reduction of overload using the code, you will not obtain the same answer as VisualDesign™ unless tributary areas are calculated in an accurate way as explained above.

IMPORTANT. We recommend modelling floors with respect to the tributary widths of beams that will support the reduced live loads. For example, don't model only two floors along three continuous beams because the reduced overloads will not be valid. Instead, model six floors along the continuous beams (three floors on each side).

Procedure:

- Select the **Loads Definition** spreadsheet and create a "Live" type of load that you will apply to the floors afterwards.
- Select the **Live** tab.
- In the column "Tributary Area - Reduction", specify the type of reduction (A or B).
- Enter the magnitude of overload in the column " Tributary Area – Overload". VisualDesign™ will reduce the reactions according to this value.

Display Convention for Loads on Floors and Members

Loads applied on members and floors are displayed on the screen according to the following conventions, if the projection of load is specified as the global axes system:

- When the load is located above the element, its numerical value is negative and the load points down.
- When the load is located below the element, its numerical value is positive and the load points up.

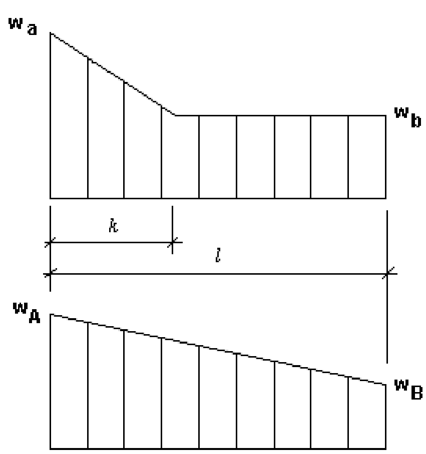
Projection and Type of Load

Type of Load	Type of Projection
Dead	Global
Snow	Global-Horizontal
Wind	Local
Live	Global
Temperature	Strong axis or weak axis
Seismic	N/A

Snow Loads

When snow loads accumulate close to a vertical surface, it is sometimes necessary to split up the floor in two parts or to transform the snow accumulation (trapezoidal + rectangle) in a trapezoidal snow live load.

You may use the following equations to transform the load:



Where:

$$w_A = \frac{2 \cdot k \cdot w_a \cdot l - 2 \cdot k \cdot w_b \cdot l + w_b \cdot l^2 - k^2 \cdot w_a + k^2 \cdot w_b}{l^2}$$

$$w_B = \frac{-k \cdot w_a \cdot l + k \cdot w_b \cdot l + w_b \cdot l^2 + k^2 \cdot w_a - k^2 \cdot w_b}{l^2}$$

Guys' Dead Load

The load factor for guy dead load (1.0) is different from the structure dead load (1.25), according to CAN/CSA-S6-00, AASHTO-LRFD-04 (bridges) and CAN/CSA-S37-01 (towers). So, when using the **Load Combination Generation Wizard**, be sure to specify the appropriate type of dead load for guys, "[D] Guy", in the **Dead load** tab of **Load Definition** dialog box. Then, this dead load must be assigned to guy members in the **Member** tab of **Member Characteristics** dialog box. .

See also

Analysing a Guyed Tower

Guys

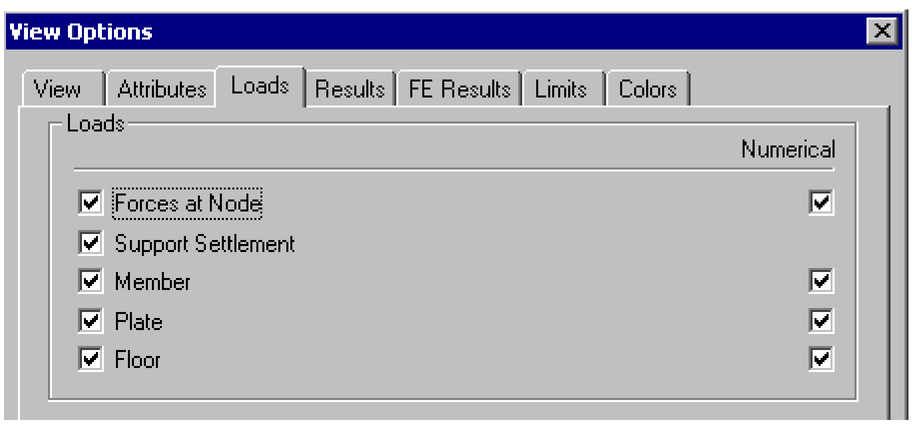
Type of loads - CAN/CSA-S6-00

Type of loads - CAN/CSA- S37-01

Type of loads - AASHTO-LRFD-04

Displaying applied loads

Loads are always displayed on the screen when a load title is selected. If you do not want to see the loads applied on the structure, select the **Loads** tab of **View Options** and uncheck the boxes.



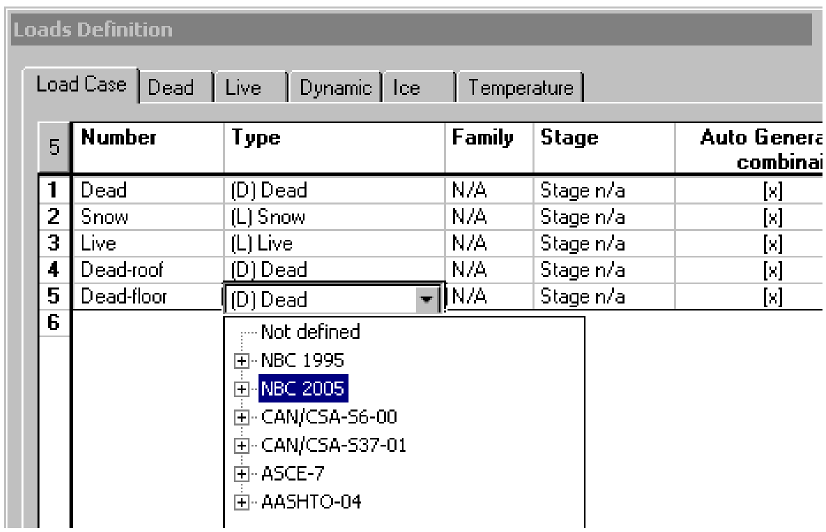
To hide numerical values, uncheck the boxes below the "Numerical" heading.

Load Types and Families

Type of Loads

This topic describes load cases that are listed in the *Type* list box of **Load Cases** tab (**Load Definition** dialog box). A selection tree appears in the list box. It is composed of roots that correspond to building codes and Standards, namely *NBC (95 &05)*, *CAN/CSA-S6-00, CAN/CSA-S37-01, ASCE-7,* and *AASHTO-LRFD-04*. Load cases that are listed under each root belong to the building code or standard. The letter describing each load case (L, S, W, etc) is also supplied.

It is very important to choose the right type of load when using the generator because VisualDesign will combine load cases and put appropriate load factors according to these types.



When a line is inserted, the *Not Defined* type is selected. The user should open the appropriate, and select a load case type if he plans to use the **Load Combination Generation Wizard**.

If load combinations have already been generated, as for an existing file, the appropriate root is open by default.

It is permitted to select load case types that are listed in other roots. However, if they are not part of the building code or standard that will be selected to generate load combinations, these load cases will not be included in the generated load combinations. The user can add them by hand when the generation is completed.

Please refer to following topics to know the list of load case types available in VisualDesign for each building code and standard:

NBC-95 Load Cases

NBC-2005 Load Cases
CAN/CSA-S6-00 Load Cases
CAN/CSA-S37-01 Load Cases
ASCE-7 Load Cases
AASHTO-LRFD-04 Load Cases

See also

Loads Definition Spreadsheet
The Load Cases tab
The Ice tab
The Wind tab
The Dynamic tab
Load Case Families
Automatic Generation of Load Combinations

NBC-95 Load Cases

To select load case types that belong to the Canadian National Building Code (95), double click in the "Type" cell of **Loads Definitions** spreadsheet. Expand the *NBC-95* root and double click on a load case type.

This table lists the NBC-95 load case types that are available in VisualDesign:

Type of Load	Description	Family (1)	Note
(D) Dead	Permanent Loads	N/a	
(E) Seismic	Overload due to seismic forces.	N/a	Spectral Envelope E
(L) Live	Overload due to usage.	4	
(L) Snow	Overload due to snow.	1	The snow load is always combined with the live load (L). A Snow load and an Auto-Ice load (or Auto-snow in this case) can be part of the same family.
(L) Auto Ice	Overload due to ice coating on members.	1	Generation of Ice Loads
(L) Dynamic	Overload due to dynamic forces other than those created by an earthquake.	4	General Dynamic Analysis (transient, harmonic, etc.)
(T) Temperature	Overload due to temperature variations.	2	Temperature loads are always combined with creep and shrinkage effects if this option is activated in the **Load Combination Generation**

			Wizard.
(T) Deformation	Overload due to deformations other than those created by temperature loads and settlements.	2	
(T) Interaction	Overload due to resulting forces induced by differential settlement under the structure foundation.	2	Refer to Soil-Structure Interaction
(W) Wind	Overload due to wind forces acting on the structure.	3	
(W) Auto Wind	Overload due to wind forces acting on the structure.	3	Generation of Wind Load
(P) Prestressing (Virtual load)	Overload due to creep and shrinkage of materials.	N/a	This load case is automatically created by VisualDesign if this option is activated in the Load Combination Generation Wizard. The *Prestressing* load case is always combined with temperature and deformation loads.

Note 1: In this table, the number representing the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

NBC-2005 Load Cases

To select load case types that belong to the Canadian National Building Code (2005), double click in the "Type" cell of **Loads Definitions** spreadsheet. Expand the *NBC-2005* root and double click on a load case type.

This table lists the NBC-2005 load case types that are available in VisualDesign:

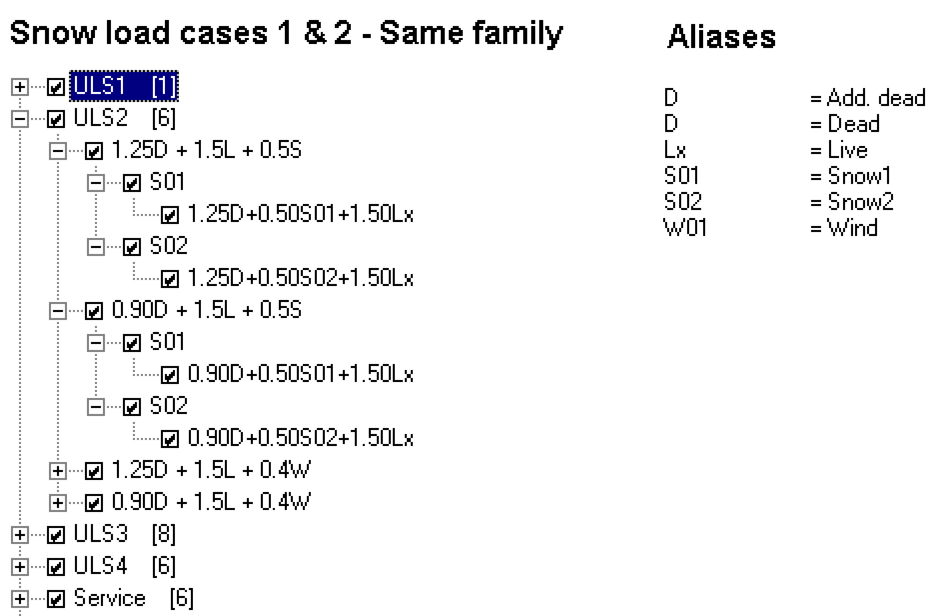
Type of Load	Description	Family (1)	Note
(D) Dead	Permanent Loads	N/a	
(H) Lateral earth pressure	Permanent Loads	N/a	This load case is always combined with the dead load case "(D) Dead"

Type of Load	Description	Family (1)	Note
(E) Seismic	Overload due to seismic forces.	N/a	Spectral Envelope E
(L) Live	Overload due to usage.	4	
(L) Dynamic	Overload due to dynamic forces other than those created by an earthquake.	4	General Dynamic Analysis (transient, harmonic, etc.)
(S) Snow (2)	Overload due to snow.	1	A Snow load and an Auto-Ice load (or Auto-snow in this case) can be part of the same family.
(S) Auto Ice	Overload due to ice coating on members.	1	Generation of Ice Loads
(T) Temperature	Permanent load Overload due to temperature variations.	Note 3	Temperature loads are always combined with creep and shrinkage effects if this option is activated in the **Automatic Generation of Load Combinations**.
(T) Deformation	Permanent load Overload due to deformations other than those created by temperature loads and settlements.	N/a	
(T) Interaction	Permanent load Overload due to resulting forces induced by differential settlement under the structure foundation.	N/a	Refer to Soil-Structure Interaction
(W) Wind	Overload due to wind forces acting on the structure.	3	
(W) Auto Wind	Overload due to wind forces acting on the structure.	3	Generation of Wind Load

Type of Load	Description	Family (1)	Note
(P) Prestressing (Virtual load)	Overload due to creep and shrinkage of materials.	Note 3	Activate option "prestressing and shrinkage/creep" in the **Specific Options** page of Automatic Generation of Load Combination Generator The *Prestressing* load case is always combined with temperature and deformation loads.

Note 1: In this table, the number that represents the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

Note 2 The CNBC-2005 uses different load factors when combining snow load cases (art. 4.1.6.3.2). Load factors for combined and reduced loads are (1.5, 0.75) and (0.5, 0.25). VisualDesign generates the following aliases when load combinations are automatically created and when snow load cases are belonging to the same family.



Note 3: The *Prestressing* load case is always combined with temperature and deformation loads if this special load case was activated in the in the **Specific Options** page of the Load Combination Generator for a prestressed concrete project.

With a steel design, temperature loads are always combined with shrinkage/creep effects applied to the slab. Refer to Thermal gradient.

CAN/CSA-S6-00 Load Cases

To select load case types that belong to this Canadian Standard, double click in the "Type" cell of **Loads Definitions** spreadsheet. Expand the *CAN/CSA-S6-00* root and double click on a load case type.

Dead Loads and Load Factors

The nomenclature used in VisualDesign to describe dead loads and load factors is based on CAN/CSA-S6-00 standard, table 3.5.1 b) *Permanent Loads: Maximum and Minimum Values of Load Factors for ULS*.

VisualDesign DL	Dead Load	αD Max	αD Min
(D1) Prefab Elements	Prefab elements, excepted timber.	1.10	0.95
(D2) Cast Concrete	Concrete cast-in-place, wood (timber), and all non-structural elements.	1.20	0.90
(D3) Wearing Surface	Wearing surface from nominal or specified thickness.	1.50	0.65
(D4) Backfill	Backfill, negative friction on piles.	1.25	0.80

VisualDesign DL	Earth Pressure	αE Max	αE Min
(E23) Pressure Active + At rest	Pressure at rest and active pressure	1.25	0.80
(E4) Backfill Pressure	Backfill Pressure	1.25	0.80
(E5) Hydrostatic Pressure	Hydrostatic Pressure	1.10	0.90

This table describes load types included in VisualDesign and to be used with CAN/CSA/S6-00 standard:

Type of Load	Description	Family (1)	Note
(D1) Prefab Elements	Dead load of prefab elements	N/a	
(D2) Cast Concrete	Dead load of cast concrete	N/a	
(D3) Wearing Surface	Dead load of wearing surfaces.	N/a	

Type of Load	Description	Family (1)	Note
(D4) Backfill	Dead load of backfill.	N/a	
(D6) Guys	Guys dead load.	N/a	Refer to Dead load of guys
(E23) Active + At rest Pressure	Loads due to earth pressure: Active pressure and pressure at rest.	N/a	
(E4) Backfill Pressure	Loads due to earth pressure: Backfill pressure.	1	
(E5) Hydrostatic Pressure	Loads due to earth pressure: Hydrostatic pressure.	N/a	
(L) Live	Overload due to usage.	4	
(L) Dynamic	Overload due to dynamic forces other than those created by an earthquake.	4	General Dynamic Analysis (transient, harmonic, etc.)
(K) Temperature	Overload due to temperature variations.	2	Temperature loads are always combined with creep and shrinkage effects.
(K) Deformation	Overload due to deformations other than those created by temperature loads and settlements.	2	
(S) Interaction	Overload due to resulting forces induced by differential settlement under the structure foundation.	2	Refer to Soil-Structure Interaction
(W) Wind on Structure	Overload due to wind forces acting on the structure.	3	
(W) Auto Wind	Overload due to wind forces acting on the structure.	3	Generation of Wind Load
(V) Wind on Traffic	Overload due to wind forces acting on live load.	3	

Type of Load	Description	Family (1)	Note
(EQ) Seismic	Overload due to seismic forces.	N/a	
(F) Hydrodynamic Pressure	Overload due to water loads, stream pressure, and floating debris.	N/a	
(A) Ice Accretion	Overload due to ice accretion	N/a	
(H) Collision	Overload due to collision forces (vehicular or vessel).	N/a	
(P) Prestressing (VisualDesign virtual load)	Overload due to creep and shrinkage of materials.	N/a	VisualDesign automatically creates this load case if it had been asked for in the Load Combination Generation Wizard. The *Prestressing* load case is always combined with temperature and deformation loads.

Note 1: In this table, the number representing the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

CAN/CSA-S37-01 Load Cases

To select load cases that belong to the Canadian CAN/CSA-S37-01 standard, double click in the "Type" cell of **Loads Definitions** spreadsheet. Expand the *CAN/CSA-S37-01* root and double click on a load case type.

This table lists load cases per CAN/CSA-S37-01 standard and available in VisualDesign:

Type of Load	Description	Family (1)	Note
(D) Dead	Permanent Loads	N/a	
(D) Guys	Permanent loads of guys.	N/a	Refer to Dead load of guys
(I) Ice	Overload due to ice coating on members.	1	
(I) Auto Ice	Overload due to ice coating on members.	1	Generation of Ice Load
(W) Wind	Overload due to wind forces acting on the tower.	2	
(W) Auto Wind	Overload due to wind forces acting on the tower.	2	Generation of Wind Load
(T) Temperature	Overload due to temperature variations.	3	
(T) Deformation	Overload due to deformations other than those created by temperature loads and settlements.	3	
(T) Interaction	Overload due to resulting forces induced by differential settlement under the structure foundation.	3	Refer to Soil-Structure Interaction
(E) Seismic	Overload due to seismic forces.	N/a	Spectral Envelope E

Note 1: In this table, the number representing the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

ASCE-7 Load Cases

To select load cases that belong to the American Standard ASCE-7, double click in the "Type" cell of **Loads Definitions** spreadsheet. Expand the *ASCE-7* root in the selection tree and double click on a load case type.

This table lists the ASCE-7 load case types that are available in VisualDesign:

Type of Load	Description	Family (1)	Note
(D) Dead	Permanent Loads	N/a	
(L) Live	Overload due to usage.	6	
(L) Dynamic	Overload due to dynamic forces other than those created by an earthquake.	6	General Dynamic Analysis (transient, harmonic, etc.)
(Lr) Live load on roof	Overload on roof	1	
(W) Wind	Overload due to wind forces acting on the structure.	2	
(W) Auto Wind	Overload due to wind forces acting on the structure.	2	Generation of Wind Load
(S) Snow	Overload due to snow.	3	The snow load is always combined with the live load (L). A Snow load and an Auto-Ice load (or Auto-snow in this case) can be part of the same family.
(S) Auto Ice (or Auto-snow)	Overload due to ice coating (or snow) on members.	3	Generation of Ice Load
(E) Seismic	Overload due to seismic forces.	N/a	Spectral Envelope E
(R) Rain	Nominal overload due to initial rainwater or ice exclusive of the ponding contribution.	4	
(T) Temperature	Overload due to temperature variations.	5	Temperature loads are always combined with creep and shrinkage effects, if asked for.

Type of Load	Description	Family (1)	Note
(T) Deformation	Overload due to deformations other than those created by temperature loads and settlements.	5	
(T) Interaction	Overload due to resulting forces induced by differential settlement under the structure foundation.	5	Refer to Soil-Structure Interaction
(F) Fluids	Overload due to fluids		
(H)	Overload due to lateral earth pressure, water & bulk		
(Fa) Flood	Overload due to flood		
(Di) Ice	Overload due to ice	3	
(Wi) Wind-on-ice	Overload due to wind-on-ice	2	
(P) Prestressing (VisualDesign virtual load)	Overload due to creep and shrinkage of materials.	N/a	VisualDesign automatically creates this load case, if asked for The *Prestressing* load case is always combined with temperature and deformation loads.

Note 1: In this table, the number representing the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

AASHTO-LRFD-04 Load Cases

Permanent Loads

The nomenclature used in VisualDesign to describe dead loads and load factors is based on AASHTO-LRFD-04 standard, Table 3.4.1-2 *Load Factors for Permanent Loads, γp*.

Type of Load	γp max	γp min
DC: Component and Attachments	1.25	0.90
DD: Downdrag	1.80	0.45
DW: Wearing Surfaces and Utilities	1.50	0.65
EH1: Horizontal Active Earth Pressure	1.50	0.90
EH2: Horizontal Earth Pressure At Rest	1.35	0.90
EV: Vertical Earth Pressure	1.35	0.90
ES: Earth Surcharge	1.50	0.75

This table describes load case types included in VisualDesign per AASHTO-LRFD-04 standard:

Type of Load	Usage	Family (1)	Note
(DC) Dead	Dead loads of structural components and non-structural attachments	N/a	
(DC) Guys	Dead loads of guys.	N/a	Refer to Dead load of guys
(DD) Downdrag	Downdrag.	N/a	
(DW3) Wearing Surface	Dead loads of wearing surfaces and utilities.	N/a	
(EH1) Active	Horizontal active earth pressure	1	
(EH2) At rest	Horizontal earth pressure at rest	1	
(EV) Earth Fill	Vertical pressure from dead load of earth fill.	N/a	

Type of Load	Usage	Family (1)	Note
(ES) Earth Surcharge	Earth surcharge load.	N/a	
(CE) Centrifugal Force	Vehicular centrifugal force.	2	
(L) Live	Vehicular live load, breaking force, pedestrian, etc.	5	In VisualDesign: (LL) = Lm + (IM)
(L) Dynamic	Dynamic forces other than those created by an earthquake.	5	General Dynamic Analysis (transient, harmonic, etc.)
(WA) Water Load	Water load and stream pressure.	N/a	
(WS) Wind on Structure	Wind forces acting on the structure.	3	
(WS) Auto Wind	Wind forces acting on the structure.	3	Generation of Wind Load
(WL) Wind on Traffic	Wind forces acting on live load.	3	
(FR) Friction	Friction	N/a	
(T) Temperature	Overload due to temperature variations.	4	Temperature loads are always combined with creep and shrinkage effects, if asked for.
(T) Deformation	Overload due to deformations other than those created by temperature loads and settlements.	4	
(TG) Temperature Gradient	Temperature Gradient	4	
(SE) Interaction	Overload due to resulting forces induced by differential settlement under the structure foundation.	4	Refer to Soil-Structure Interaction
(EQ) Seismic	Seismic forces.	N/a	

Type of Load	Usage	Family (1)	Note
(IC) Ice Accretion	Overload due to ice accretion	N/a	
(CV/CT) Collision	Overload due to collision forces (vehicular or vessel).	N/a	
(P) Prestressing (VisualDesign virtual load)	Overload due to creep and shrinkage of materials.	N/a	VisualDesign automatically creates this load case, if asked for. It is always combined with temperature and deformation loads.

Note 1: In this table, the number representing the family means that these load cases can be part of the same family. It does not represent the number that has to be entered in the "Family" cell of **Load Definitions** spreadsheet.

Load Case Families

A family is useful when using the **Load Combination Generation Wizard** when particular load cases need to be combined together. The family is automatically created for some load case types only and a family will always include load cases that belong to the same type such as *Wind*, *Autowind*, *Wind on traffic* or *Ice*, *Auto-Ice*, etc. These load cases will always be combined together when generating load combinations. Family numbers are editable by double clicking in the cell.

Select option *N/a* in the drop-down list box if you do not want to use families.

The concept was created for backfill pressure loads used with CAN/CSA-S6-00 standard. Backfill pressure loads are combined together but they have different load factors. VisualDesign combines backfill earth pressures and *Alpha E* load factors as per table 3.5.1b) of this standard.

Up to 25 families can be created in the **Loads Definition** spreadsheet. It is permitted to have a family number 1 for wind loads and another family number 1 for temperature loads. However, we recommend using different numbers to avoid mistakes and confusion.

Families are allowed for the following Codes and load cases:

NBC-1995:

- Live and Dynamic;
- Wind and Autowind;
- Temperature and Interaction;
- Snow and Auto-ice (In this case, Auto-ice can be considered as Auto-snow).

NBC-2005:

- Wind and Autowind;
- Live and Dynamic;
- Snow and Auto-ice (In this case, Auto-ice can be considered as Auto-snow).

ASCE-7:

- Live and Dynamic;
- Wind and Autowind;
- Temperature and Interaction;
- Snow and Auto-ice (In this case, Auto-ice can be considered as Auto-snow).
- Live load on Roof;
- Rain.

CAN/CSA-S6-00 and AASHTO-LRFD-04:

- Live and Dynamic;
- Wind, Autowind, and wind on traffic;
- Temperature and Interaction;
- Backfill pressure.

CAN/CSA-S37-01:

- Wind, and Autowind;
- Ice, and Auto-ice.

Example

We have six load cases of the *Wind* type (W1, W2, ... W6). Usually, when using the Wizard, the load combination D + W will generate six load combinations, each including the dead load and one of a wind load.

If W1 and W2 are part of the same family and W3 and W4, another family, VisualDesign will generate four load combinations instead of six:

LC1 = D + (W1+W2);
LC2 = D + (W3+W4);
LC3 = D + W5;
LC4 = D + W6.

In the **Selections** page of **Load Combination Wizard**, the wind load cases and corresponding aliases will be as follows:

W1-1: Wind load W1 included in family #1;
W1-2: Wind load W2 included in family #1;

W2-1: Wind load W3 included in family #2;
W2-2: Wind load W4 included in family #2;

W01: single wind load W5

W02: single wind load W6

Restrictions: Wind and Ice Loads

VisualDesign calculates wind on iced members. (The field "Ice thickness" is part of the **Wind** tab.) To avoid problems when using families with wind loads, make sure that:

All the wind loads that are belonging to the same family have an ice thickness of zero;

Or

All the wind loads that are belonging to the same family have an ice thickness greater than zero.

See also

The Load Cases tab
The Wind tab
The ice tab

Loads Definition Spreadsheet

Loads Definition Spreadsheet

Before applying loads on your structure, all load cases title and type must be defined. This spreadsheet is located in **Loads / Load Cases / Definition**. It is divided into four tabs: **Load Cases**, **Dead**, **Live**, **Dynamic**, **Wind, Ice** and **Temperature.**

The **Load Cases** tab is the main one. All load case titles and types must be defined in this tab. Specific parameters are entered in other tabs. Load case families are supplied in the **Load Cases** tab in order to combine appropriate load cases when using the **Load Combination Generation Wizard**. Families are created for backfill pressure, wind, and ice loads. To learn more about families, refer to topic Load Case Families.

The **Dead** tab is useful to differentiate dead load cases, such as self-weight, additional dead load, and others.

The **Live** tab must be filled in if a reduction of tributary areas for loaded floors is required. A "Live" type of load must be selected in the **Load Cases** tab to activate the **Live** tab.

The **Dynamic** tab is activated if a *Dynamic* type of load has been entered in the **Load Cases** tab. A dynamic type of load is used to define general dynamic load cases. You must own the ***Dynamic Analysis*** module.

The **Wind** and **Ice** tabs are used for applying wind and ice loads on open structures such as towers.

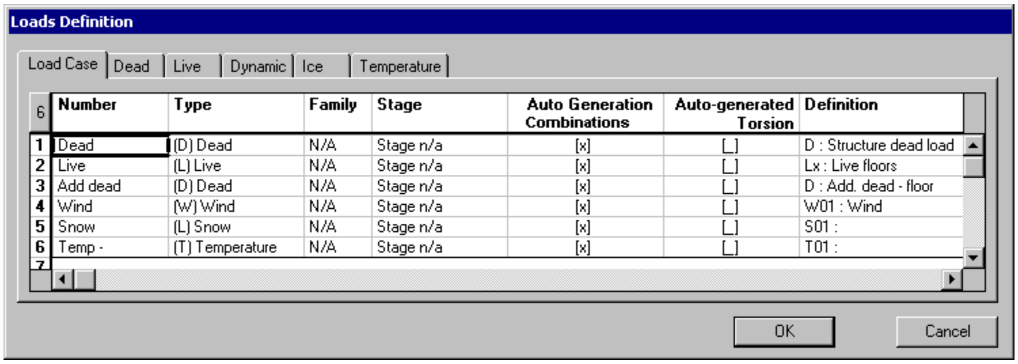
The **Wind** tab will be activated if a *Wind* or *Autowind* type of load is entered in the **Load Cases** tab and if calculation method (CAN/CSA-S37-01 or Environment Canada) for wind loads has been chosen in the **Steel** tab of **Project Configuration**.

The **Temperature** tab is useful to indicate if this load can be combined with ice and snow loads, when using the **Load Combination Generation Wizard**.

See also

Type of Loads
Load Cases tab
The Dead tab
The Live tab
Dynamic tab
Wind tab
Ice tab
The Temperature tab

The Load Case tab (Master)



6	Number	Type	Family	Stage	Auto Generation Combinations	Auto-generated Torsion	Definition
1	Dead	(D) Dead	N/A	Stage n/a	[x]	[ ]	D : Structure dead load
2	Live	(L) Live	N/A	Stage n/a	[x]	[ ]	Lx : Live floors
3	Add dead	(D) Dead	N/A	Stage n/a	[x]	[ ]	D : Add. dead - floor
4	Wind	(W) Wind	N/A	Stage n/a	[x]	[ ]	W01 : Wind
5	Snow	(L) Snow	N/A	Stage n/a	[x]	[ ]	S01 :
6	Temp -	(T) Temperature	N/A	Stage n/a	[x]	[ ]	T01 :

Group: Load case data

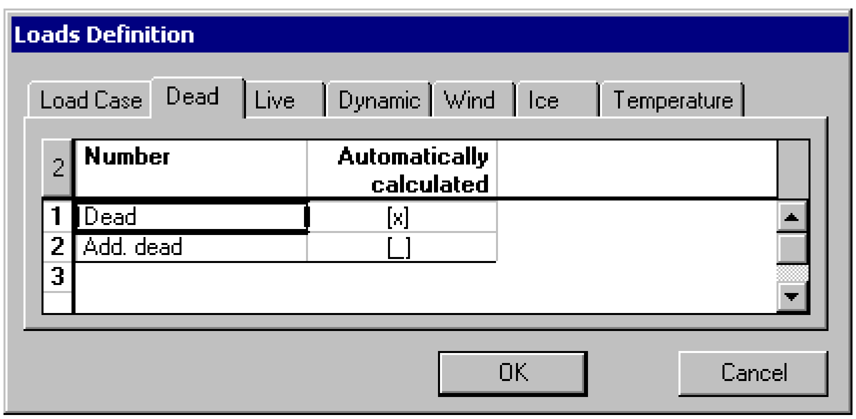
Column	Description	Editing
ID	Calculated automatically	No
Number	Number/name of this load case (12 alphanumeric characters)	Single click
Type	Double click to open the Load Case selection tree. Open the appropriate root and highlight a type of load case. Please refer to Type of Loads for more details.	Double-click
Family	Family number to which belongs this load case (backfill pressure, wind, ice, etc.), if it needs to be combined with specific load cases. Choose option *N/a* if you do not want to use families. Please refer to Load Case Families	Double-click
Stage	Steel design module or Prestressed concrete design module: When the analysis is completed, this shaded field informs the user the construction stage that corresponds to this load case.	No
Auto Generation Combinations	Disable this option ([ ]) to remove this load case from the automatic generation of load combinations. By default, all load cases are activated. See topic Generator of Load Combinations	Double-click or Space bar
Auto generated Torsion	Dynamic Analysis module: This option ([x]) includes accidental torsion effects from the spectral analysis. Equivalent static loads will be generated by the software and automatically integrated into static analysis or design.	No
Definition	Comment.	Single click

Note If you want to include accidental torsion effects into a static or time history analysis, you must perform a spectral analysis beforehand. The spectral analysis generates equivalent static loads to take into account accidental torsion effects.

The Dead tab

All dead loads are grouped in this tab. The subdivision of dead load cases allows differentiating the self-weight of elements from other cases that could correspond to construction stage loads (composite beams with the steel design module or prestressed concrete module).

The option "Automatically calculated" must be activated for the dead load cases that represent the elements' self-weight, if they are not part of any construction stages. In fact, the user must specify a dead load case in each **Member**, **Plate** and **Floor Characteristics** dialog box to inform VisualDesign about the calculation of self-weights. We recommend not modifying this option if other dead loads are already applied to the structure. You could loose dead load cases.

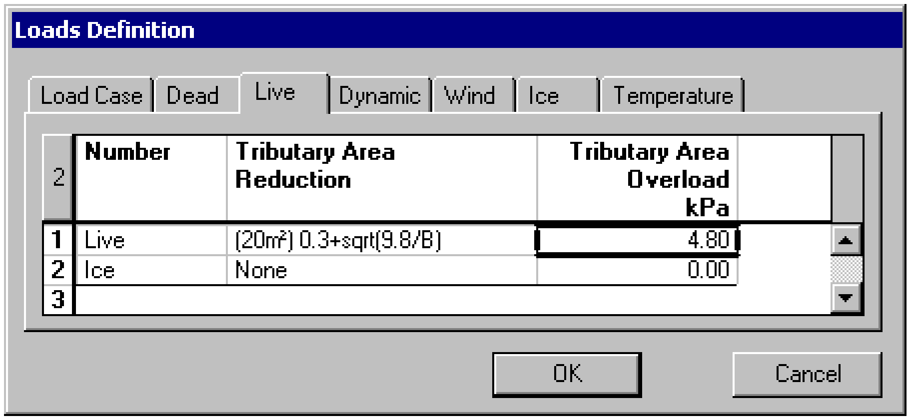


Group: Load Case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Number/name of this load case (12 alphanumeric characters)	No
Automatically calculated	Activate this option ([x]) if you want VisualDesign to consider this load case as the structure self-weight, which is automatically calculated.	Double click or spacebar

The Live tab

All live load cases are grouped in this tab. Complete the parameters to reduce the floor tributary areas.



Group: Load Case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Number/name of this load case (12 alphanumeric characters)	No
Tributary Area - Reduction	Select the reduction formula that applies to the floor tributary area.	Double-click
Tributary Area - Overload	Enter the magnitude of overload that will be considered in reducing the forces transferred to columns.	Single click

See also

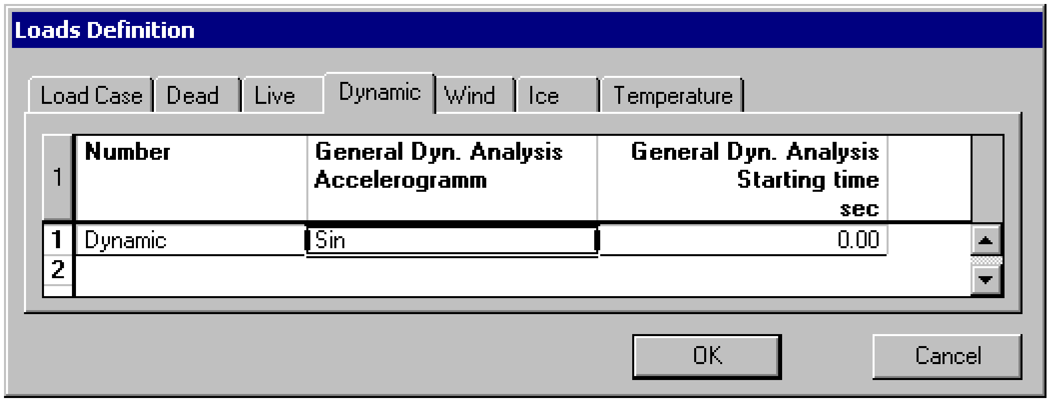
Reduction of tributary area

The Load Cases tab

The Dynamic tab

This tab applies to general dynamic loadings. Therefore, you must own the Dynamic Analysis module.

A "Dynamic" type of load must be selected in the **Load Cases** tab to activate this tab. It allows selecting a pre-defined dynamic loading (Loads menu) and specifying a starting time for this accelerogramm.



Group: Load case data

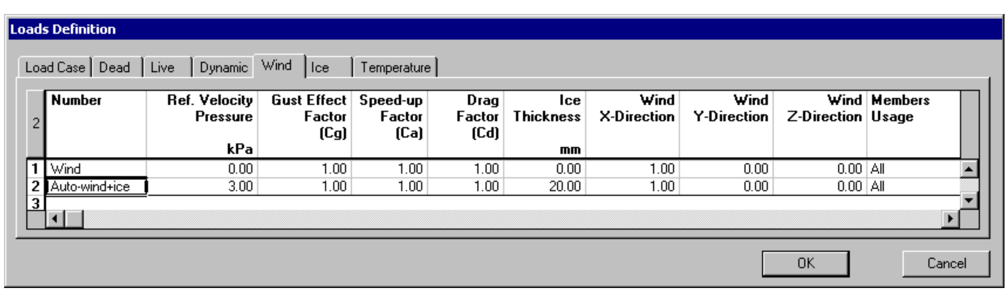
Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
General Dynamic Analysis-Accelerogram	Choose an accelerogram that will be applied to a specific node of the structure.	Double-click
General Dynamic Analysis – Starting time	Specify a starting time for the accelerogram.	Single click

See also

General Dynamic Analysis

The Wind tab

A *Wind* or *Auto wind* type of load case must be selected in the **Load Cases** tab of **Loads Definition** spreadsheet in order to activate this tab. A calculation method must also be selected in the **Steel** tab of **Project Configuration** dialog box. Fields that will appear in this spreadsheet are those relative to the chosen calculation method.



A "Wind" type of load can be used for a wind load that is applied to a selected tower panel. (Refer to Wind on Panels). Ice coating can be specified for members composing the selected panel.

An "Auto-wind" type of load is useful to generate ice loads for an open structure such as towers. If sections changed during the design process, ice coatings are automatically calculated and applied by VisualDesign. Ice coatings are considered in the calculation of the projected area under wind loads.

Wind Loads According to CAN/CSA-S37-01 Calculation Method

Group: Load case data

Column	Description	Editing
ID	Calculated automatically	No
Load case number	12 alphanumeric characters	Single click
q	Reference Pressure	Single click
Gust Effect Factor (Cg)	Gust Effect Factor (Cg)	Single click
Speed-up Factor (Ca)	Speed-up Factor (Ca)	Single click
Drag Factor (Cd)	Drag Factor (Cd)	Single click
Ice Thickness	Enter the ice thickness on members that will be subjected to wind loads.	Single click
Wind in the x-direction	A value of 1.0 represents 100% of wind applied in this direction.	Single click
Wind in the y-direction	A value of 1.0 represents 100% of wind applied in this direction	Single click
Wind in the z-direction	A value of 1.0 represents 100% of wind applied in this direction	Single click
Member Usage	Select the member usage for which this load will be apply.	Double click

Wind Loads According to Environment Canada Calculation Method

Column	Description	Editing
ID	Calculated automatically	No
Load case number	12 alphanumeric characters	Single click
Gust Effect Factor (Cg)	Gust Effect Factor (Cg)	Single click

Column	Description	Editing
Speed-up Factor (Ca)	Speed-up Factor (Ca)	Single click
Drag Factor (Cd)	Drag Factor (Cd)	Single click
Ice Thickness	Enter the ice thickness on members that will be subjected to wind loads.	Single click
Wind in the x-direction	A value of 1.0 represents 100% of wind applied in this direction.	Single click
Wind in the y-direction	A value of 1.0 represents 100% of wind applied in this direction	Single click
Wind in the z-direction	A value of 1.0 represents 100% of wind applied in this direction	Single click
Member Usage	Select the member usage for which this load will be apply.	Double click
a1	Site coefficient given by Environment Canada	Single click
a2	Site coefficient given by Environment Canada	Single click
a3	Site coefficient given by Environment Canada	Single click
Zh	Site coefficient given by Environment Canada	Single click
Zo1	Site coefficient given by Environment Canada	Single click
Vo1	Wind velocity per Environment Canada (mph).	Single click

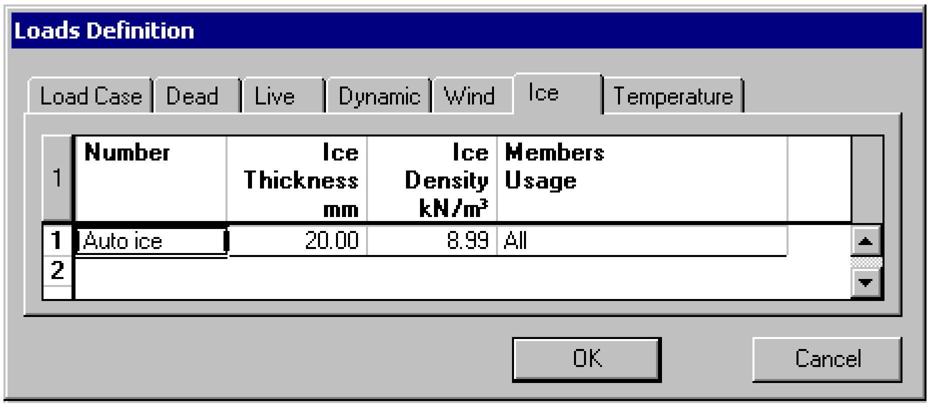
See also

Tower Design
Automatic Generation of Wind Loads
Generating Wind Loads
Automatic Generation of Ice Loads
Wind on Panels

The Ice tab

An *Auto ice* type of load must be selected in the **Load Cases** tab of **Loads Definition** spreadsheet in order to activate this tab.

If sections are modified during the design process, VisualDesign will automatically calculate and apply new ice coatings over members.



Group: Load case data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Ice thickness	Enter the ice thickness on members	Single click
Density	Enter ice density.	Single click
Member Usage	Select the member usage for which this load will be apply.	Double-click

See also

Tower Design

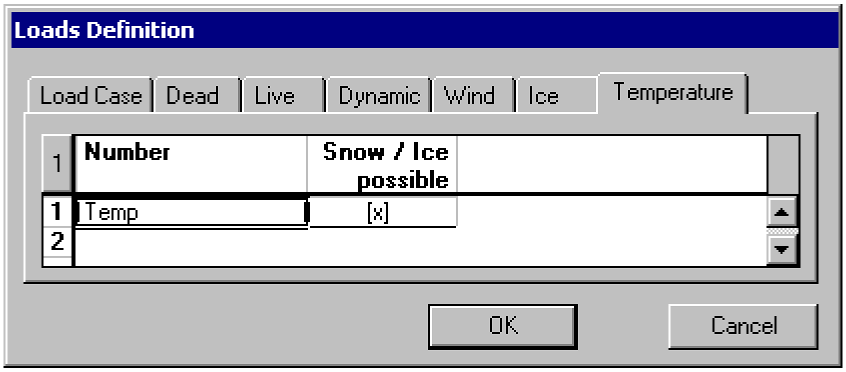
Automatic Generation of Ice Loads

Generating Ice Loads

The Temperature tab

A *Temperature* type of load must be specified in the **Load Cases** tab of **Loads Definition** spreadsheet in order to activate this tab.

If this temperature load can be combined with ice or snow loads, according to selected code, activate the option ([x]) in the column "Snow/Ice possible" if you plan to use the Load Combination Generator.



Group: Load case data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	No
Snow/Ice possible	Activate this option ([x]) if this temperature load can be combined with snow or ice loads.	Double click or Spacebar

Automatic Generation of Loads

Automatic Generation of Ice Loads

The **Automatic Generation of Ice Loads** function (under **Loads** menu /**Load Cases**) allows you to generate ice loads on open structure (such as towers) during all cycles of design.

N.B. You must define an *Auto ice* type of load in the **Load Cases** tab of **Loads Definition** spreadsheet. Then, in the **Ice** tab, complete the required parameters.

Tower Design Module:

When shapes change, VisualDesign™ calculates new ice loads according to new shapes. VisualDesign™ will apply the ice according to the configuration that you chose before launching the design. The program will keep it during all its design cycles.

Steel Design Module:

When shapes change, VisualDesign™ calculates new ice loads according to new shapes and applies ice according to the configuration that you chose before launching the design. The program will keep it during all its design cycles. This means that if you deleted some ice loads on members, the program will not create ice loads on those members even if the shapes have changed during the design.

See also

Generating Ice Loads

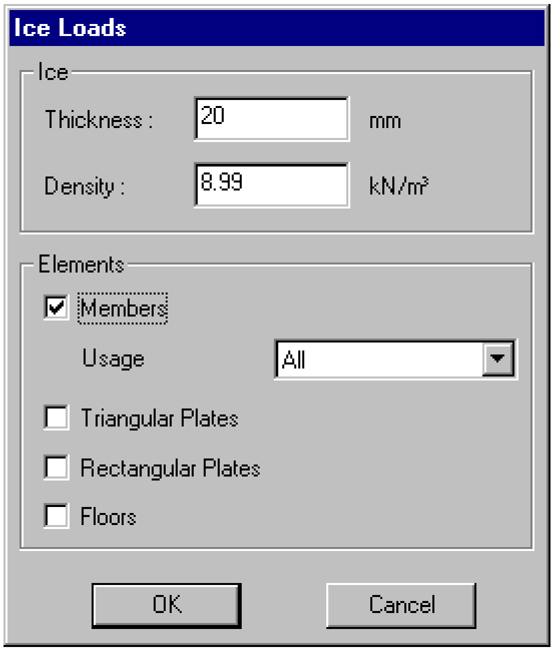
Loads Definition spreadsheet

Tower Design Module

Calculation of ice coating

Generating Ice Loads

- Select an *Auto-Ice* type of load in the **Load Cases** tab of **Loads Definition** spreadsheet.
- Select the **Ice** tab and complete required parameters.
- Then, activate the "Load Case" mode on Activation toolbar and select the *Auto-Ice* load case in the drop-down list box.
- Go to **Loads** / **Load Cases** / **Automatic Generation** / **Ice Loads.** You can modify parameters in the **Ice Load** dialog box.



- If you wish to delete some ice loads, select elements and press [Delete]. VisualDesign will keep this configuration during all design cycles except for tower design, where ice coating cannot be deleted. They will be automatically added.

Calculation of Ice Coating

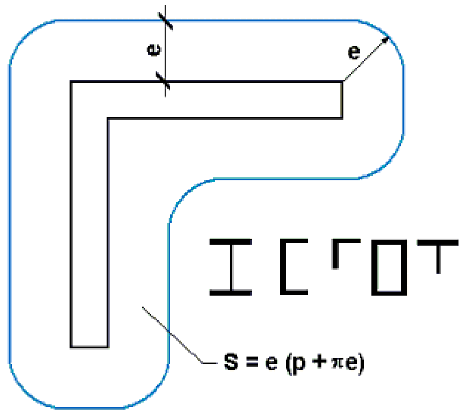
Ice coatings are linear loads oriented according to global axis system. Ice loads applied on floors and plates are surface loads oriented according to global axis system.

The weight of the ice coating on members is calculated as follows:

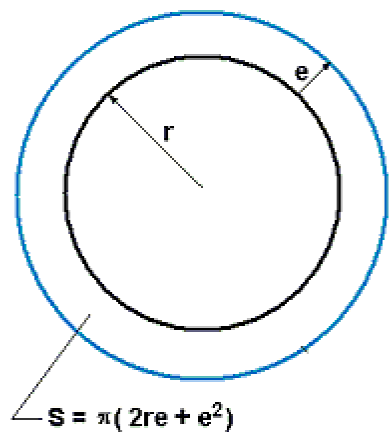
W = Density * Thickness * S

"S" is the area of ice surrounding the shape and is calculated with the equations presented below:

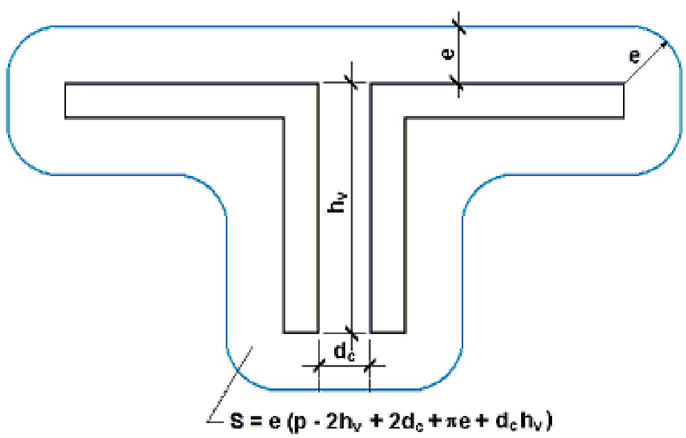
General



Round shapes



2L shapes



Automatic Generation of Wind Loads

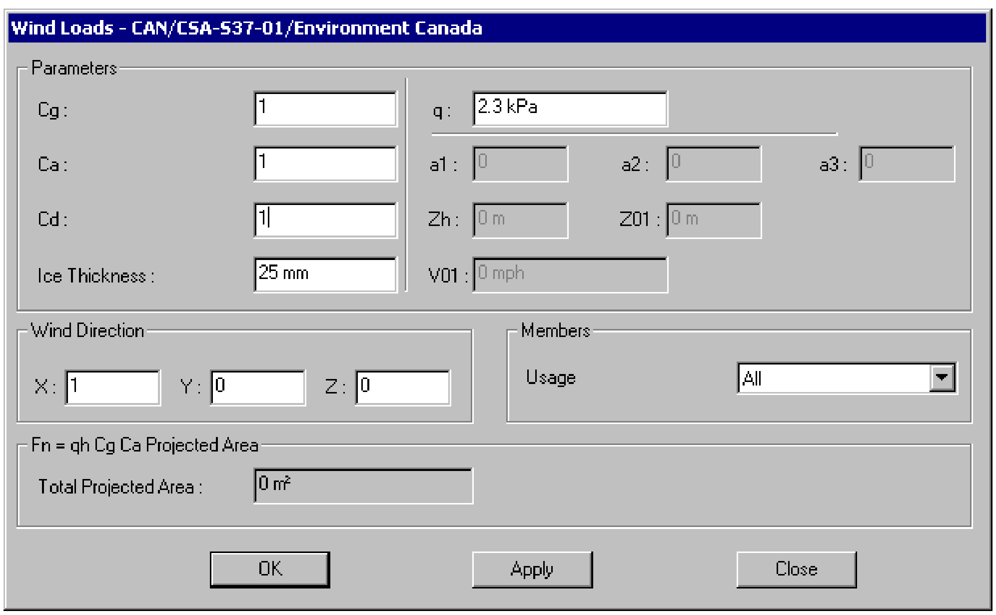
Selecting a method for wind calculation

The calculation method for wind loads must be selected in the **Steel** tab of **Project Configuration** dialog box before generating wind loads. Available methods are based on *CAN/CSA-S37-01* Standard and *Environment Canada*. The **Wind Load** dialog box will include specific fields for the chosen method.

Then, an *Auto wind* type of load must be defined in the **Load Cases** tab of **Loads Definition** spreadsheet beforehand. The wind parameters can be specified in the **Wind** tab of **Loads Definition** spreadsheet.

Generating Wind Loads

Then, from the **Loads / Load Cases** menu, select **Automatic Generation/Wind Loads** to automatically generate wind pressure on an open structure according to a given direction of wind (x, y and z components). Furthermore, the total projected area acting on the structure is automatically calculated. Only members can be loaded with wind. Then, VisualDesign™ will transfer wind loads as forces at end nodes.



Note The *Auto-wind* type of load is used for the calculation of wind on guys only, and a Cd of 1.2 must be specified. The wind applied to members must be a *Wind* type of load, and a Cd of 1.0 is required.

During a Cyclic Design (Steel design):

If VisualDesign™ modifies shapes during its cyclic steel design, new wind loads will be calculated according to new shapes and a new analysis will be carried on with new pressure loads. The program will do so as long as it reaches convergence. VisualDesign™ will apply wind loads according to the wind configuration that you chose before launching the design. The program will keep it during all its design cycles.

According to chosen method used to calculate wind pressure (*Steel* tab of **Project Configuration**), the dialog box will include the following parameters:

Wind Pressure according to CAN/CSA-S37-01:

Fields	Description
Parameters	
q	Reference pressure
Cg	Gust Effect Factor
Ca	Speed-up Factor
Cd	Drag Factor
Ice Thickness	Ice coating on members
Wind Direction	
X	X component of wind direction
Y:	Y component of wind direction
Z:	Z component of wind direction
Members	Specify member usage that will be loaded automatically at design.
Fn = qh Cg Ca Projected Area	
(Af) Total Projected Area	Automatic calculation of total projected area.

Wind Pressure according to Environment Canada:

Topic	Description
Parameters	
Cg	Gust Effect Factor
Ca	Speed-up Factor
Cd	Drag Factor
a1	Site coefficient given by Environment Canada
a2	Site coefficient given by Environment Canada
a3	Site coefficient given by Environment Canada
Zh	Site coefficient given by Environment Canada
Zo1	Site coefficient given by Environment Canada
Vo1	Wind velocity as per Environment Canada (mph)
Ice Thickness	Ice coating on members
Wind Direction	
X	X component of wind direction
Y:	Y component of wind direction
Z:	Z component of wind direction
Members	Specify member usage that will be loaded automatically at design.
Fn = qh Cg Ca Projected Area	
(Af) Total Projected Area	Automatic calculation of total projected area.

See also

Generating Wind Loads
Tower Design module
Loads Definition spreadsheet
Steel tab of Project Configuration
Wind on Panels

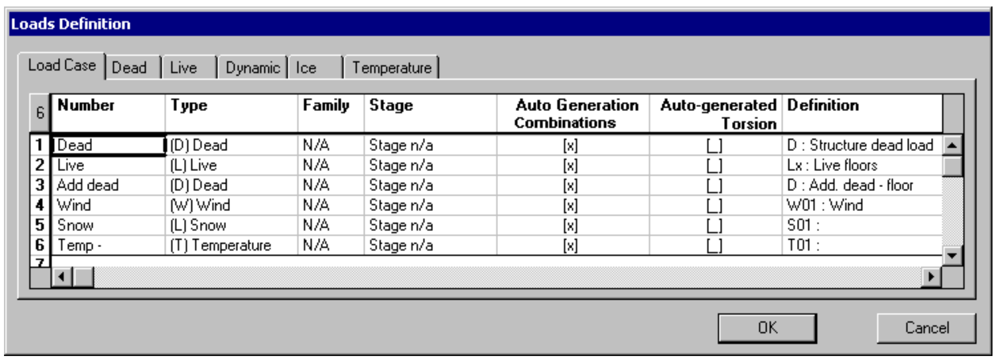
Generating Wind Loads

- Select **Project Configuration** in **File** menu. In the **Steel** tab, select a method for the calculation of wind pressure.
- Select the **Loads Definition** spreadsheet. Define an "Auto-wind" type of load in the **Load Cases** tab. Then, select the **Wind** tab and complete required parameters. Close the spreadsheet.
- Activate the "Load Case" mode on Activation toolbar and select the *Auto-Wind* load in the drop-down list box.
- Go to **Loads** menu and select **Automatic Generation / Wind Loads**. Fill in parameters in the **Wind Load** dialog box.

Editing Loads

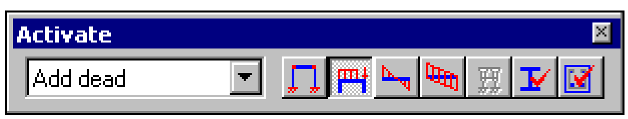
Defining Loads

- Define load titles and types in the **Loads Definition** dialog box. To do so, go to **Loads/Load Cases** / **Definition**. Insert lines to enter load cases. Give a name to each and double-click in the "Type" column to select the right type of load case according to code.



- Activate the Load Case mode. To do so, choose **Activate mode/Load Case** from the **Edit** menu or click "Load Case" on the "Activation" toolbar.

- Select a load case title from the scroll list on the Activation toolbar,



You are now ready to graphically apply loads to elements.

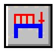
Applying Loads

- Double-click on an element to open the **Loads** dialog box or click once and press the **Properties** icon. Enter the load magnitude according to local or global axis system. Loads can also be applied through spreadsheets, which are listed in the **Loads** menu.

Or

- To apply common loads on many elements of the same type, keep the [Ctrl] key down while you select (click once) each element and press the **Properties** icon. Enter loads in the dialog box. Loads will be applied to all selected elements.

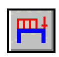
Modifying Applied Loads

- Activate the "Load Case" activation mode by pressing the "Load Case" icon  of Activation toolbar.
- Select the load case title from the pull-down menu of the Activation toolbar.
- Select the appropriate spreadsheet in the **Loads** menu. The spreadsheet will include all elements of the same type.

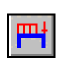
OR

- Activate a type of element on Activation toolbar, select elements and open the appropriate spreadsheet in the **Loads** menu. The spreadsheet will include loads applied to selected elements only.
- You may now modify loads using the spreadsheet's contextual menu (Replace, Modify (+, -, x or /), etc.) or by typing in new values.

Applying Multiple Modifications to Loads

- Activate the "Load Case" mode using one of the following procedures:
  - Click the "Load Case" icon  of Activation toolbar.
  - Go to **Edit/Activate Mode/Load Case**.
- Select a group of elements of the same category.
- Open the **Loads** dialog box using one of the following procedures:
  - Click on the "Properties" icon  of Edit toolbar.
  - Go to **Edit/Properties**.
- Types in common loading values then press OK.

Deleting Applied Loads to Elements

- Activate the "Load Case" mode by doing one of the following:
  - Click the "Load Case" icon  of Activation toolbar.
  - Choose **Activate Mode/Load Case** from the **Edit** menu.
- Select the appropriate load case on **Activation** toolbar.
- Activate the appropriate element on Elements toolbar and select the loaded elements of this type.

- Do one of the following:
    - Click the "Delete" icon on the Edit toolbar.
    - Choose **Delete** from the **Edit** menu.
    - Press the [Delete] key.
- The applied loads will be deleted on selected elements.

Loads Applied to Nodes

Loaded Nodes

To apply loads on nodes, first activate the "Load Case" mode from Activation toolbar. Then, select a load case title from the list box.

Double-click on a node to open the **Forces on Nodes** spreadsheet or select many and activate the **Properties** function from **Edit** menu.

Specify the intensity of concentrated loads or bending moments in the global x, y, and z-axis.

See also
Forces on Nodes Spreadsheet
Support Settlements Spreadsheet
Colour Display of Loads on Nodes
Applying Loads on Nodes

Forces on Nodes Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Node Number	12 alphanumeric characters	Single click
Fx	Concentrated load towards global x-axis.	Single click
Fy	Concentrated load towards global y-axis.	Single click
Fz	Concentrated load towards global z-axis.	Single click
Mx	Bending moment around x-axis (global system)	Single click
My	Bending moment around y-axis (global system)	Single click
Mz	Bending moment around z-axis (global system)	Single click

Applying Loads to Nodes

- Go to **Loads/Load Cases /Definition**.
- Create a load case title and select the appropriate type of load in the spreadsheet.
- Activate the "Load Case" activation mode by pressing the "Load Case" icon of the Activation toolbar.
- Select the load case title from the pull-down menu of the Activation toolbar.
- Do one of the following:
  - Click twice on a node to reach the **Forces on Nodes** spreadsheet or select many nodes and click the **Properties** icon from Edit toolbar.
  - Access the **Forces on Nodes** spreadsheet through the **Loads/Load Cases** menu.

Coloured Display of Loads applied on Nodes

Forces and bending moments are always displayed by default. To disable this option, open the **View Options** dialog box and uncheck option "Loads on Nodes" in the **Loads** tab.

Arrows represent forces and bending moments on nodes. The direction of arrows follows the rule of the right hand.

You can modify their graphic attributes through the **Colour** tab of **View Options** dialog box.

See also
View Options

Loads Applied to Supports

Loaded Supports

Two loads can be applied to supports: Displacement (x, y, and z) and rotations (theta x, y, and z). These loads are always applied according to global axes system, except if the support is oriented.

Note: To assign a settlement in one of the specified directions, the support must have been blocked beforehand in that direction under the "Structure" activation mode, or have a preset rigidity (spring support).

The following table indicates which settlement can be applied to a support in the "Load Case" mode, as well as results that will be supplied, depending on the degree of freedom allowed for in the "Structure" mode. (To enter or view data, activate the "Structure", "Load Case", or "Load Combination" mode, and select a support or many. Then call the **Properties** function.)

Support's degrees of freedom	**Loads (Settlement) Displacement or Rotation**	**Load combination Results (Displacement or Rotation)**	
		Displacement	**Response**
Restrained	Applicable	Settlement Value	Variable
Unrestrained	Not applicable	Variable	Null
Spring	Applicable	Variable	Variable
Secant modulus K	Applicable	Variable	Variable

See also

Colour Display of Loads Applied to Supports
Support's DOF
Applying Settlements to Supports

Applying Settlements to Supports

To apply forces or settlements to supports, you must first activate the "Load Case" mode on the Activation toolbar and select the load case title. Then, activate the Support icon on Elements toolbar and select one support or more. Select **Properties** from **Edit** menu.

- Create a load case title. To do so, select the **Definition** heading under **Loads / Load Cases** menu.
- Activate the "Load Case" activation mode on Activation toolbar and select the load case title from the pull-down menu of the Activation toolbar.

- Activate the Support icon on Elements toolbar and select one support or more.
- Select the **Properties** function of **Edit** menu.
- Enter the direction of displacement or rotation (x, y, or z-axis) in the **Support Settlements** spreadsheet.
- To add a load directly to a support, use the **Forces on Nodes** spreadsheet.

Support Settlement Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Support Number	12 alphanumeric characters	Single click
Displ. x	Imposed settlement in the x-direction (global axis system)	Single click
Displ. y.	Imposed settlement in the y-direction (global axis system)	Single click
Displ. Z	Imposed settlement in the z-direction (global axis system)	Single click
θx	Imposed rotation – x-direction (global axis system)	Single click
θy	Imposed rotation – y-direction (global axis system)	Single click
θz	Imposed rotation – z-direction (global axis system)	Single click

Coloured Display of Support Displacements

Only displacements can be viewed graphically on the screen. Rotations are ignored.

To view displacements, check the "Support Settlements" option of **Loads** tab from the **View Option** dialog box.

The graphic attributes of this option can be modified through the **Colours** tab from the same dialog box.

Loads Applied to Members

General

Application of Loads on Members

There is no limit regarding the quantity of loads that can be applied to one member for the same load case title (concentrated, distributed, torsion, temperature, and shrinkage).

Users have the choice to apply loads on member(s) through a spreadsheet or a dialog box.

Through the dialog box:

- Select one or many members and select the **Properties** function from **Edit** menu. The **Loads on Member** dialog box is then displayed on the screen. It is composed of five tabs: Distributed, Concentrated, Temperature Variations, Torsion, and Shrinkage. Enter or modify loads.

Through the spreadsheet:

- To enter, modify, or sort data, select all members or a few of them and call up the **Load on Members** spreadsheet under **Loads/Load Cases/Members** menu. Modify values in the spreadsheet using functions included in spreadsheet's contextual menu (right click). Refer to Spreadsheet's contextual menu (Contextual menu).

Temperature Load: Behaviour and Assumptions

- VisualDesign™ applies the temperature variation in a linear way from the top to the bottom of the section, according to the selected projection axis.
- Position of top and bottom depends essentially on the member incidence.
- If a concrete beam is subjected to a thermal gradient that makes it curve upwards, the top of the section will not be cracked because this behaviour is caused by the expansion of material. If ends of member are restrained, the top of the section will be compressed and the bottom, tensioned.

To look at the constant bending moment caused by a thermal gradient applied between two fixed supports: Pinned the supports and apply the same bending moments (in opposite direction) on each support. You will obtain the same effect as thermal gradient.

Remark It is not the temperature, but solely the *temperature differential* that must be indicated in boxes "Top of Section" and "Bottom of Section"

Example:

We want to evaluate the internal forces in a steel column where the bottom fibre is exposed to a 20ºC while the temperature at the top fibre is -40ºC.

If the steel section has been factory manufactured at 20°C, the temperature differential that should appear in the box "Top of Section" will be **–60°C** and the "Bottom of Section", will be **0°C**.

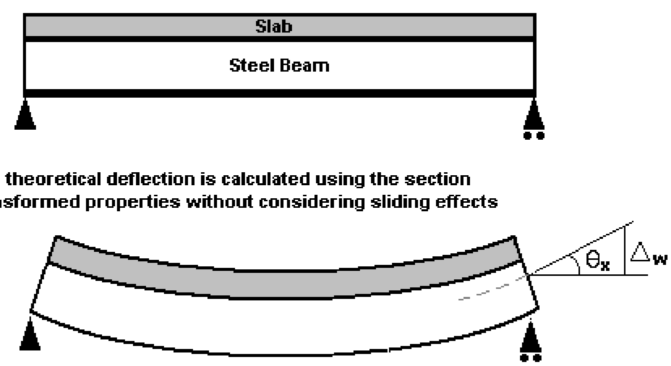
Thermal Gradient and Shrinkage Effects

A thermal gradient can be applied to the slab of a steel/concrete composite beam. It will be applied to the top and bottom of the slab. Enter the temperature variation at the top and bottom of the concrete slab and select option *Thermal Gradient* in column "Type of Application".

VisualDesign's approach

Step 1:

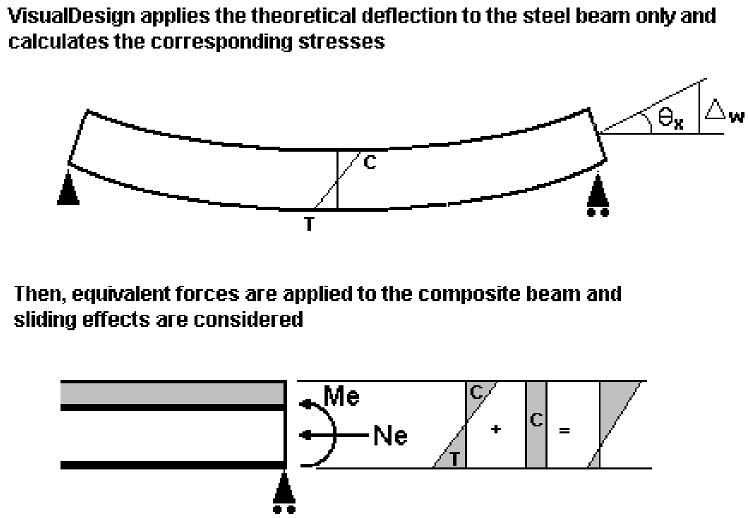
Deflections σw and θx are calculated with the transformed properties of the composite beam without considering any sliding effects (theoretical composite beam).



Step 2:

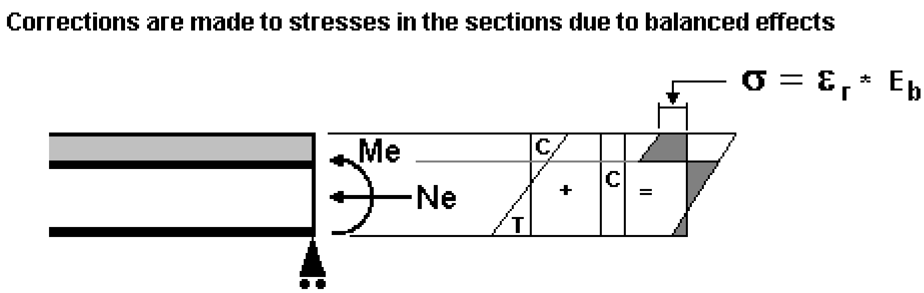
The deflection found at step one is applied to the steel member and the real area and inertia are used.

- If the section is composite, VisualDesign will consider the transformed area and inertia and will include sliding effects.
- If the section is not composite, only the steel section is considered. The real area and inertia of the steel section will be used in the calculation unless the user activated the option that considers the reinforcement in the slab for negative bending moments. In this case, the calculation of inertia will be done considering tensioned reinforcement in the slab.



Balanced Effects

The correction of stresses in the section is due to balanced effects (shrinkage and thermal gradient). It is done with the section-transformed properties with no sliding effect and a ratio of 1n is used (or the one specified by the user in the **Composite Beam** tab of **Project Configuration**).



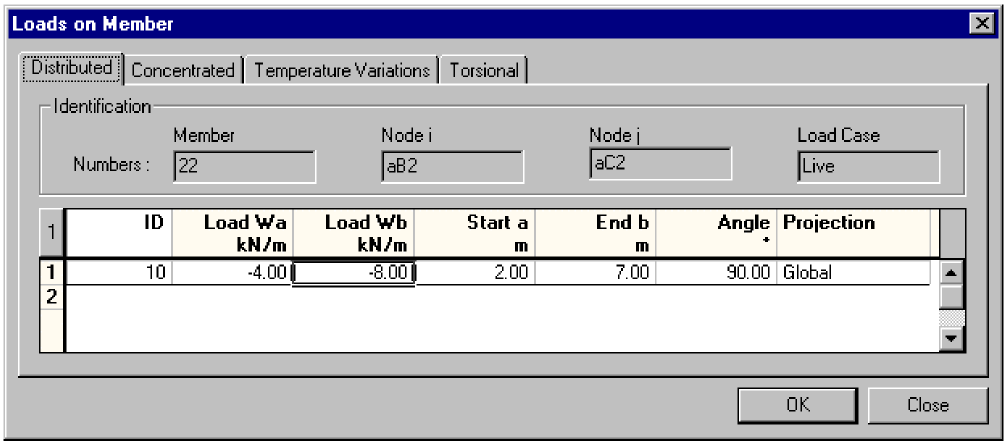
Applying Loads to Members

- Go to **Loads / Load Cases / Definition** menu. Insert lines and create load case titles and types.
- Activate the "Load Case" activation mode and select the load case title from the pull-down menu of Activation toolbar.
- Select one member or more.
- Call the **Properties** function under **Edit** menu. Enter loads in the **Loads on Member** dialog box. It is composed of five tabs: Distributed, Concentrated, Temperature Variations, Torsion, and Shrinkage.

- You can also define loads on members through spreadsheets (**Loads /Load Cases / Members**.

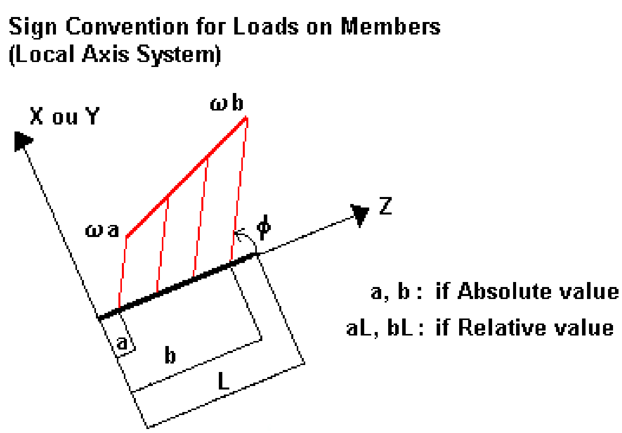
Loads on Member Dialog Box

This dialog box is composed of the following tabs: **Distributed**, **Concentrated**, **Temperature Variations**, **Torsion**, and **Shrinkage** (composite member only)



Distributed Loads

You can apply an unlimited number of distributed loads on any members. They can be uniform, trapezoidal or triangular, depending on the selected projection.



Select one member or more and click the **Properties** function from **Edit** menu to open the **Loads on Member** dialog box.

The table below describes the headings included in the **Distributed** tab of this dialog box.

Heading	Description
Loads Wa and Wb	Magnitude of distributed load located at point A and B along the member length.
Beginning (a) and End (b)	Beginning and ending positions of the distributed load along the member. By default, load is fully distributed along the member length. Positions can be specified as fraction of the member length, depending on selected units. (To modify units, click on column title "Position" and press down right mouse button. Select "Change Units").
Angle	The angle of inclination of distributed load on the member strong or weak axis must be specified if the projection is "Strong Axis" or "Weak Axis". If the selected projection is "Global Horizontal" or "Global", the numerical value of angle will be ignored.
Projection	**Strong Axis**: The distributed load is applied according to the member positive local y-axis. **Weak Axis**: The distributed load is applied according to the member positive local x-axis. **Global**: The distributed load is applied according to positive global y-axis (dead weight). **Global Horizontal**: The distributed load is applied according to positive global y-axis, along the horizontal projection of member. Mostly used for snow loads.

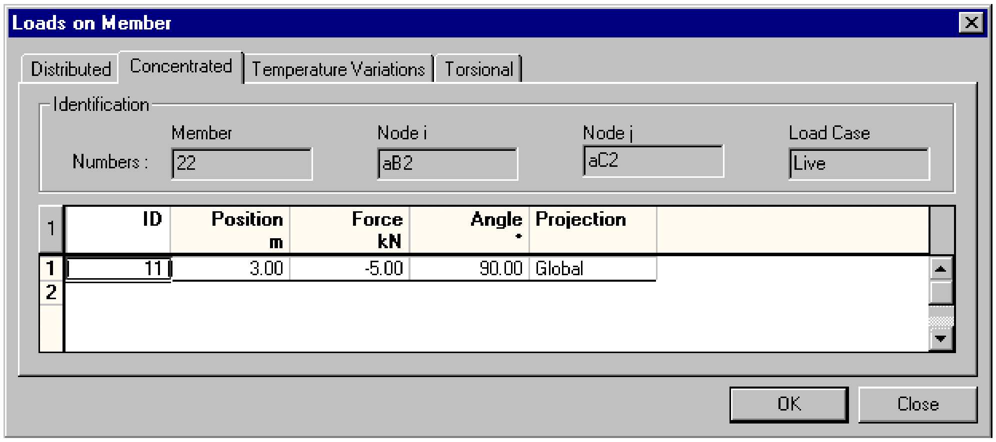
See also

Distributed Loads on Members spreadsheet

Concentrated Loads

A concentrated load is applied to one point along the member length. An unlimited number of concentrated loads can be applied to a member.

Select one member or more and click the **Properties** function from **Edit** menu to open the **Loads on Member** dialog box.



The table below gives a description of headings included in the **Concentrated** tab of this dialog box.

Column	Description
Position	Position of the concentrated load along the member length. The position can be expressed as a fraction of the member length, depending on selected units. (To modify units, click on the column title "Position" and right click. Select "Change Units")
Force	Magnitude of concentrated load.
Angle	Angle of inclination of concentrated load according to the member strong or weak axis. The angle must be defined if the selected projection is "Strong Axis" or "Weak Axis". (If the selected projection is "Global", the angle will be ignored even if a value is specified)
Projection	**Strong Axis**: the concentrated load is applied according to the member positive local y-axis. **Weak Axis**: the concentrated load is applied according to the member positive local x-axis. **Global**: the concentrated load is parallel to gravity axis (ex.: a dead load would have a negative value).

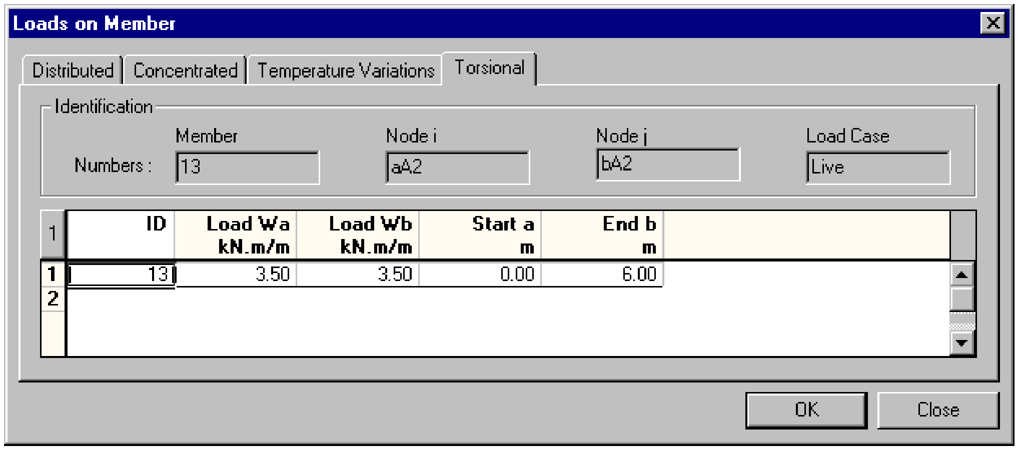
See also

Concentrated Loads on Members spreadsheet

Torsional Loads

It is possible to apply an unlimited number of torsion loads on a member. Torsion loads are applied at an angle of 90°. Projection of load is not required.

Select one member or more and click the **Properties** function from **Edit** menu to open the **Loads on Member** dialog box. Select the **Torsional** tab.



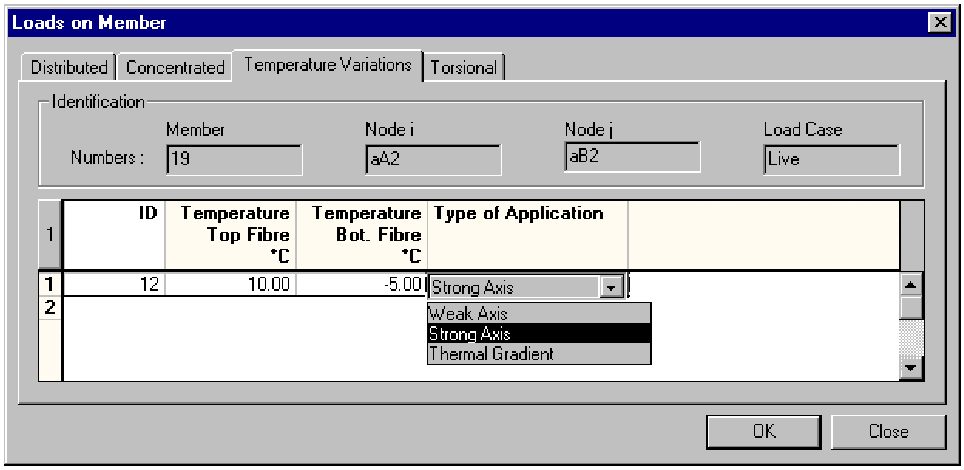
Column	Description
Load Wa	Magnitude of torsion load at point A (node I).
Load Wb	Magnitude of torsion load at point B (node j).
Start a	Start position of torsion load along the member length. By default, load is fully distributed along the member length. Position can be entered as a fraction of member length, depending on selected units. (To modify units, click on column title "Position", right click, and select "Change Units".
End b	End position of torsion load along the member length. By default, load is fully distributed along the member length.

See also

Torsional Loads on Members Spreadsheet

Temperature Variations

Select one member or more and click the **Properties** function from **Edit** menu to open the **Loads on Member** dialog box. Select the **Temperature Variations** tab and specify the temperature differential at top and bottom of the section.



The table below describes the headings included in this tab.

Column	Description
Top of Section	Temperature differential at the top fibre.
Bottom of Section	Temperature differential at the bottom fibre.
Type of Application	The selection of strong or weak axis will influence the linear application of the temperature variation on the section. The option *Thermal Gradient* is specific to steel/concrete composite beam. See Composite Beam

See also

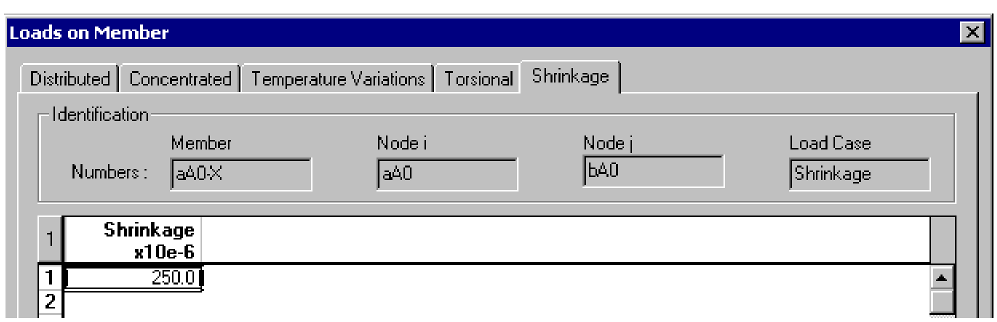
The Temperature Variations on Members spreadsheet

Composite Beams

Shrinkage

Select this tab and enter deformation due to shrinkage that will be applied to the concrete slab of a composite (steel/concrete) section.

N.B. Calculation of shrinkage effects is done automatically for a prestressed concrete composite element.



The table below describes the headings included in this tab.

Heading	Description
ID	Automatically calculated
Shrinkage x10e-6	Deformation due to shrinkage. It will be applied to the concrete slab of a composite section.

Loads on Member Spreadsheet

These spreadsheets are accessible in **Loads / Load Cases / Members**.

Concentrated Loads on Members Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Member Number	12 alphanumeric characters	Single click
Position	Position of concentrated load on member (x=0 at node i)	Single click
Force	Magnitude of concentrated load applied to the member.	Single click
Angle	Angle of applied load.	Single click
Projection	Type of load projection: according to weak, strong or global axis.	Double-click

Distributed Loads on Members Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Member Number	12 alphanumeric characters.	Single click
Load Wa	Magnitude of distributed load at point a.	Single click
Load Wb	Magnitude of distributed load at point b.	Single click
Start a	Position of load Wa along member length.	Single click
End b	Position of load Wb along member length.	Single click
Angle	Angle of applied load.	Single click

Column	Description	Editing
Projection	Type of load projection: according to weak, strong, global, or horizontal global axes.	Double-click

See also

The Distributed tab (Load on Member Dialog Box)

Temperature Variations on Members Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Member Number	12 alphanumeric characters	Single click
Top. Temp.	Temperature differential at the top of the section.	Single click
Bot. Temp.	Temperature differential at the bottom of the section.	Single click
Type of Application	Select the weak or strong axis. Option *Thermal Gradient* is specific to composite beams (steel/concrete). See Composite Beams	Double-click

See also

Composite Beams

Torsional Loads on Members Spreadsheet

Group: Load case: (title)

Column	Description	Editing
ID	Automatically calculated	No
Wa Load	Magnitude of torsional load at point *a*	Single click
Wb Load	Magnitude of torsional load at point *b*	Single click
Start a	Position (origin) of the torsion load along the member length. By default, the load will be applied on the entire member length.	Single click
End b	End position of the torsion load along the member length.	Single click

See also

The Torsional tab (Load on Members Dialog Box)

Loads due to Shrinkage

Group: Load case: (title)

Column	Description	Editing
ID	Automatically calculated	No
ε shrinkage	Shrinkage factor that will be applied to the concrete slab of a composite section.	Single click

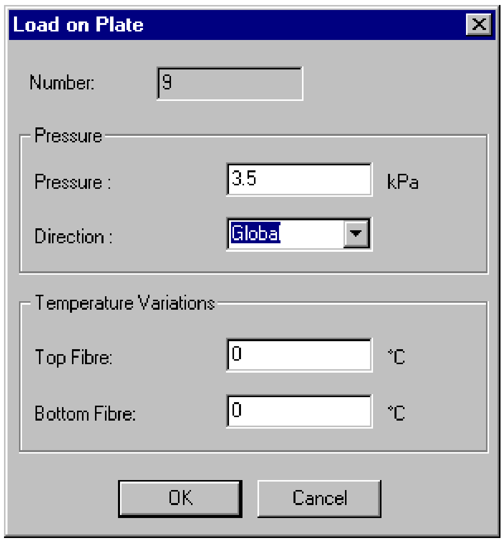
See also

Steel-concrete Composite Beam

Loads applied to Plates

Load on Plate Dialog Box

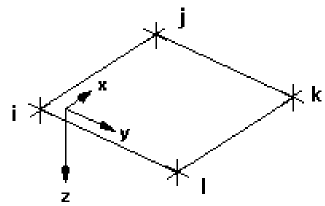
The **Load on Plate** dialog box allows applying or modifying pressure loads and temperature variations.



Description of this dialog box:

Topic	Description
Number	Plate number (if you selected only one).
Pressure	
Pressure	Uniform pressure applied to the plate(s)
Direction	**Local**: the pressure load is applied perpendicular to the plate, in the positive direction of local z-axis. **Global**: the pressure load is applied according to the positive direction of global y-axis. **Global Horizontal**: the pressure load is applied along the horizontal projection of plate, according to the positive direction of global y-axis.
Temperature Variations	
Top	Temperature differential at the top of the plate.
Bottom	Temperature differential at the bottom of the plate.

Plate Local Axis System



See also

Pressure on Plates spreadsheet

Temperature Variations on Plates spreadsheet

Applying Loads on Plates

Applying Loads to Plates

- Activate the Load Case mode and select the load case title from the pull-down menu of Activation toolbar.
- Click twice on a plate to reach the **Loads on Plate** dialog box or select many plates and press the **Properties** icon from the Edit toolbar.
- Enter pressure and temperature variation on plates.

OR

- Access one of the loads spreadsheets from **Loads** menu.

Loads on Plates Spreadsheets

Pressure on Plates Spreadsheet

Group: Load case: (title)

Column	**Description**	**Editing**
ID	Calculated automatically	No
Plate Number	12 alphanumeric characters	Single click
Geometry	Geometry of this plate: Rectangular or Triangular	No
Pressure	Magnitude of applied pressure.	Single click
Projection	Type of load projection: local, global, or horizontal global.	Double-click

Temperature Variation on Plates

Group: Load case: (title)

Column	Description	Editing
ID	Automatically calculated	No
Plate number	12 alphanumerical characters describing the plate.	Single click
Geometry	Geometry of this plate: Rectangular or Triangular.	No
Temperature Top Fibre	Enter the temperature variation at the top of the plate.	Single click
Temperature Bottom Fibre	Enter the temperature variation at the bottom of the plate.	Single click

See also

Shear wall and effects of temperature loads

Loads applied to Floors

General

Application of Loads

Distributed loads and concentrated loads can be applied to floors. (The floor dead load can be integrated into the structure dead load through the **Floor Characteristics** dialog box.) There is no limit regarding the quantity of loads that can be applied to one floor.

Beams are required along the floor outline (4 sides). Loads will be distributed on adjacent beams at an angle of 45 degrees for a two-way floor and on the end beams, for a one-way floor. Use a material density of zero to create a "fictive" beam if there is no beam on one side of a two-way floor.

To apply loads on floors, select the "Load Case" activation mode on Activation toolbar. Then, select the appropriate load case title. Loads can be applied through the spreadsheet or dialog box.

Dialog box:

- Select one or many floors and select the **Properties** function from **Edit** menu. The **Loads on Floor** dialog box is then displayed on the screen. It is composed of two tabs: *Distributed* and *Concentrated*. Enter loads.

Spreadsheet:

- To enter, modify, or sort data, select all floors or a few of them and select the **Load on Floors** spreadsheet through **Loads/Load Cases /Floors** menu.

See also

Floor Characteristics dialog box
Distributed Loads on the Floor
Concentrated Loads on the Floor
Floor Dead Load
The Display Convention for Loads on Floors and Members
Displaying Floors having no Dead Load

Floor Dead Load

The software uses the floor material and thickness to automatically calculate the dead weight. However, the user can also specify it through the "Floor Dead Load" field available in the **Floor Characteristics** dialog box.

This floor dead load is distributed on adjacent beams, as other loads on floors, depending on the type of floor (one-way or two-way). So, the floor dead load will be added to adjacent members' dead load (automatically calculated by VisualDesign™).

To view the distribution of the floor dead load, select the "Load Case" activation mode and from the pull down list of load case titles, select "Dead". Select the floor and call the **Properties** function. The **Loads on Floor** dialog box allows viewing (but not modifying) the floor dead load.

See also

Floor Characteristics

Displaying Floors with Unspecified Dead Load

- Go to the **Attributes** tab of **View Options** dialog box and activate the *No Dead Load* option in the "Floors" section. Floors with no specified dead load will be highlighted on screen.

- To assign a dead load to them (optional), select the **Properties** function to open the **Floor Characteristics** dialog box, and enter a dead load. This dead load will be considered in the structure dead load case (it is automatically calculated in the Loads Definition spreadsheet).

Note. The analysis can be performed even if there is no dead load specified for floors.

See also

Floor Characteristics

The Attributes Tab

Load on Floor Dialog Box

To apply or modify concentrated loads on floors, select the "Load Case" activation mode on Activation toolbar. Then, select the appropriate load case title.

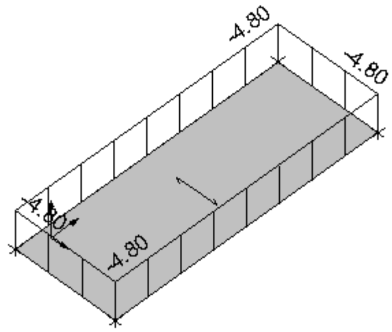
Select one or several floors and click the **Properties** icon to open the **Load on Floor** dialog box. It is composed of two tabs: the **Distributed** tab and **Concentrated** tab.

Distributed Loads

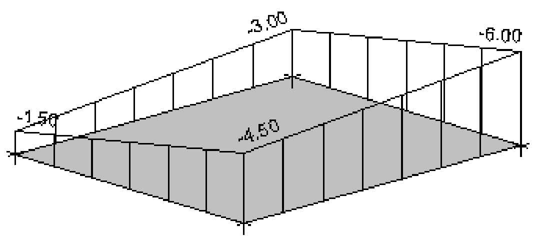
A distributed load on a floor is corresponding to a pressure load. Beams (real or fictitious) are required on the outline for 2-way floors. Bearing beams are needed on the two sides of a one-way floor. You can apply an unlimited number of distributed loads to any floor.

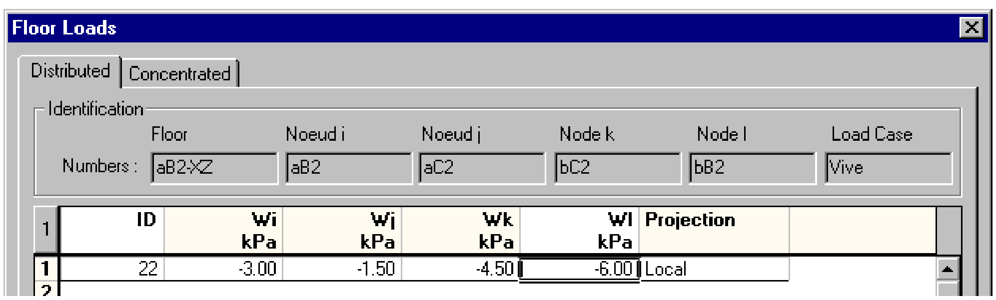
Distributed loads can be uniform or trapezoidal.

Distributed Load on Floor



Trapezoidal Load on Floor





The table below describes the headings included in the **Distributed** tab of this dialog box.

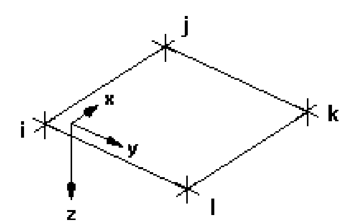
Column	Description
Identification	Shaded fields that inform the user about the selected floor number, incidence nodes and load case title.
ID	VisualDesign's ID number for this distributed load.
Loads Wi, Wj, and Wk	These loads (pressure) must be set by the user and will be applied to the floor nodes i, j, and k.
Load Wl	Load Wl is automatically calculated to form a plane with loads Wi, Wj, and Wk.

Column	Description
Projection	**Local**: the distributed load goes along the positive direction of the local z-axis system of the floor. **Global**: the distributed load goes along the positive direction of the global y-axis system. **Global Horizontal**: the distributed load goes along the positive direction of the global y-axis system. The load is distributed on the horizontal projection of the floor.

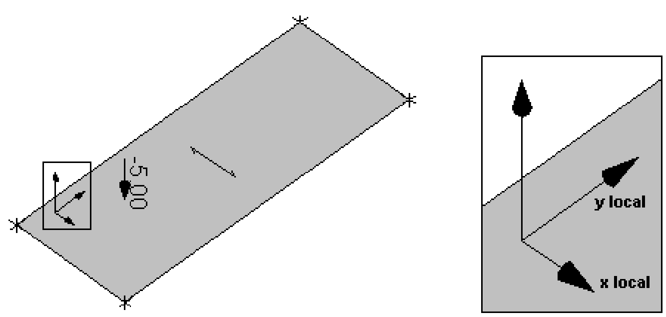
Concentrated Loads

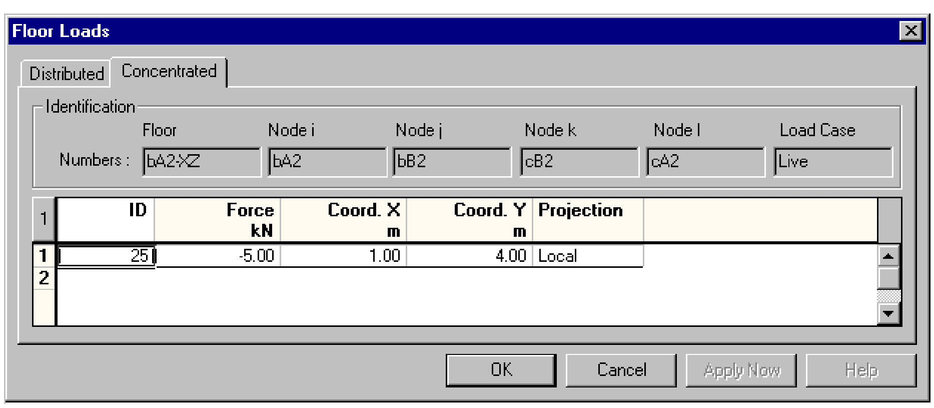
A concentrated load applied to the floor is a point load and is positioned (x, y) according to the floor local axis system.

Floor Local Axes System



Concentrated Load on Floor





The table below describes the headings included in the **Concentrated** tab:

Column	Description
Identification	Shaded fields inform the user about the selected floor number, incidence nodes and load case title.
ID	VisualDesign's concentrated load ID number.
Force	Magnitude of concentrated load.
X-, Y-Coordinates	Position of concentrated load: Coordinates X and Y are corresponding to the floor local axes system.
Projection	**Local:** the concentrated load is applied perpendicular to the floor. **Global:** the concentrated load is oriented in the same direction as gravity axis.

See also

Concentrated Loads on Floors Spreadsheet

Load on Floor Spreadsheets

Concentrated Loads on Floors Spreadsheet

This spreadsheet is accessible through the menu **Loads / Load Cases / Floor / Concentrated.**

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Floor Number	12 alphanumeric characters	Single click
Force	Force to be applied	Single click
Coord. X	Local x-coordinate	Single click
Coord. Y	Local y-coordinate	Single click
Projection	Type of load projection: local or global.	Double-click

See also

The Concentrated tab (Load on Floor Dialog Box)

Distributed Loads on Floors Spreadsheet

This spreadsheet is accessible through the menu **Loads / Load Cases / Floor / Distributed.**

Group: Load case: (title)

Column	Description	Editing
ID	Calculated automatically	No
Floor Number	12 alphanumeric characters	Single click
Wi	Pressure of distributed load at corner i of floor.	Single click
Wj	Pressure of distributed load at corner j of floor.	Single click
Wk	Pressure of distributed load at corner k of floor.	Single click
Wl	Pressure of distributed load at corner l of floor. This value depends on the three previous values.	No
Projection	Type of load projection: local, global, or horizontal global.	Double-click

Load Combinations

Definition of Load Combinations

Load combinations are automatically generated according to a selected Standard or Code when using the **Load Combination Generation Wizard**. This function is available under **Load Combinations /Automatic Generation** in **Loads** menu.

However, for those wanting to define load combinations "by hand", they can do it through the **Combinations** dialog box (**Loads / Load Combinations /Definition)**. It is composed of two tabs: **Load Combinations** and **Load Factors**. Each one of these tabs includes a spreadsheet.

See also

The Load Combinations Tab

The Load Factors Tab

Load Combination Generator

Copying a Load Combination along with Load Factors

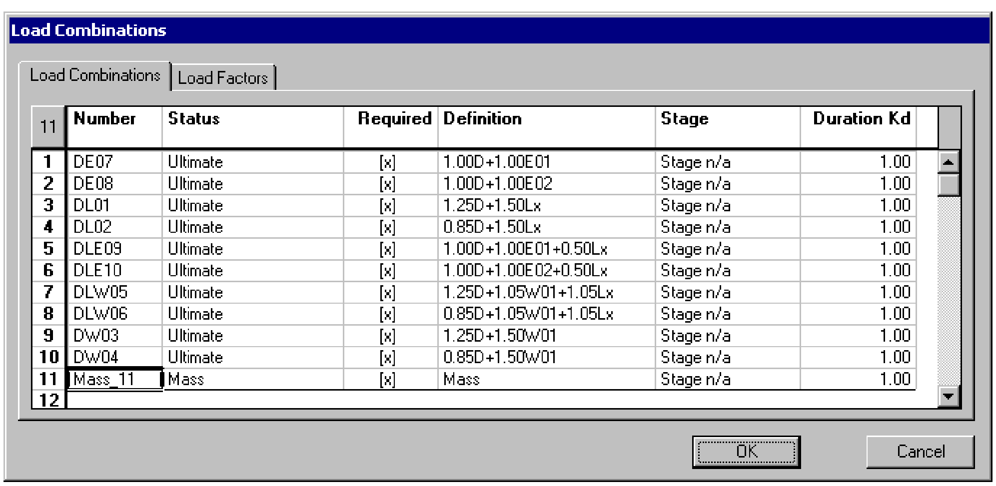
Load Combinations Dialog Box

This dialog box is completed if you used the **Load Combination Generator.**

To open this dialog box, go to **Loads / Load Combinations / Definition**.

The Load Combinations Tab

If you want to create load combinations by hand, insert lines in the spreadsheet and complete the columns. Look below at the definition of each cell that is included in the spreadsheet.



11	Number	Status	Required	Definition	Stage	Duration Kd
1	DE07	Ultimate	[x]	1.00D+1.00E01	Stage n/a	1.00
2	DE08	Ultimate	[x]	1.00D+1.00E02	Stage n/a	1.00
3	DL01	Ultimate	[x]	1.25D+1.50Lx	Stage n/a	1.00
4	DL02	Ultimate	[x]	0.85D+1.50Lx	Stage n/a	1.00
5	DLE09	Ultimate	[x]	1.00D+1.00E01+0.50Lx	Stage n/a	1.00
6	DLE10	Ultimate	[x]	1.00D+1.00E02+0.50Lx	Stage n/a	1.00
7	DLW05	Ultimate	[x]	1.25D+1.05W01+1.05Lx	Stage n/a	1.00
8	DLW06	Ultimate	[x]	0.85D+1.05W01+1.05Lx	Stage n/a	1.00
9	DW03	Ultimate	[x]	1.25D+1.50W01	Stage n/a	1.00
10	DW04	Ultimate	[x]	0.85D+1.50W01	Stage n/a	1.00
11	Mass_11	Mass	[x]	Mass	Stage n/a	1.00
12						

Group: Load Combination data

Column	Description	Editing
ID	Calculated automatically	No
Number	Load combination name or number (12 alphanumeric characters).	Single click
Status	Specify the load combination status, according to a code, through the selection tree. If a code is not required, select option *Analysis only*. To learn more, refer to topic Load Combination Status	Double-click
Required	This option [x] integrates this load combination at analysis or withdraws it [ ]. By default, all generated load combinations will be analysed.	Double-click or Spacebar
Definition	Example: 1,25+(1,50Lx)	Single click
Stage (1)	Steel Design module or Prestressed Concrete Design module: If construction stages were defined, indicate the stage number where this load combination will be applied to the structure.	Single click
Duration Kd	Timber Design only: This duration factor will be constant for this load combination if it differs from load cases.	Single click

Note 1: Composite Beams (steel/concrete and concrete/concrete)

Loads specified at a given construction stage are automatically cumulated at each following construction stages. Example: stage 3 loads will be added to stage 4 loads.

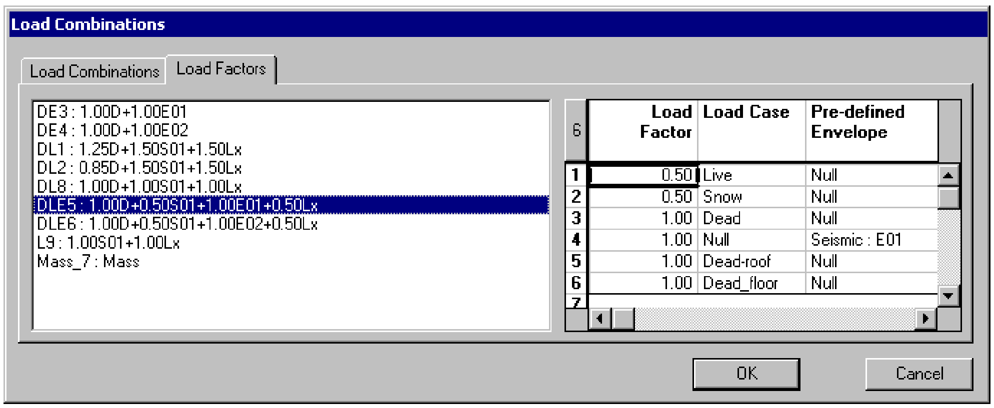
See also

The Load Factors Tab
Load Combination Generator
Prestressing tab (Project Configuration)
Composite Beam tab (Project Configuration)
Copying a Load Combination along with Load Factors
The Timber Design module

The Load Factors Tab

This tab is part of the **Load Combinations** dialog box, which is accessible through the **Loads** menu / **Load Combinations**. The **Load Factor** tab is completed if the Load Combination Generator was used.

The left part of the dialog box is a list box that contains the name of the generated load combinations. You can move to another load combination by clicking on its name. The right part of the dialog box is a spreadsheet where load factors are entered and assigned to each load case and pre-defined envelopes (E, Et, Etnl et Lm), if any.



If no lines are present in the spreadsheet, insert the number of required lines (one line per load case that is part of a load combination).

Group: Load Combination: (title)

Column	Description	Editing
ID	Calculated automatically	No
Load factor	Highlight a load combination in the left part of the dialog box and enter each load factor that must be assigned to each load case and each pre-defined envelope that is composing this load combination.	Single click
Load case	Highlight a load combination in the left part of the dialog box and select the load case that is part of this load combination.	Double-click
Pre-defined envelope	Highlight a load combination in the left part of the dialog box and select the pre-defined envelope that is part of this load combination.	Double-click

See also

Load Combination Generator

Copying a Load Combination along with Load Factors

Defining Load Combinations

- Use the Load Combination Generator (**Loads / Load Combinations / Automatic Generation**).

Or

- Go to **Loads / Load Combinations / Definition.** Insert lines in the first tab. Give a name to each load combination and specify the status.
- Select the **Load Factors** tab.
  - Click once on a load combination in the left part of the dialog box and insert lines in the right part. The number of lines must correspond to the number of load cases that is composing this load combination.
  - Double-click in the "Load Case" column and select a load case. If a pre-defined envelope must be included, do the same in the "Pre-defined Envelope" column.
  - Enter load factors for each load case and pre-defined envelope.
  - Click OK to exit the dialog box.

Copying a Load Combination & Load Factors

To simultaneously copy a load combination along with load factors, use function **Duplicate** available in the spreadsheet's contextual menu.

Procedure:

- Open the **Load Combinations** spreadsheet.
- Select the line that corresponds to the load combination you want to copy.
- Right click and choose function **Duplicate** in contextual menu.

Load Combination Statuses (General)

Load combination statuses must be specified in the **Load Combinations Definition** spreadsheet, through a selection tree when double clicking in the "Status" cell. The roots that you will find in this selection tree correspond to a code or standard and they are composed of a list off available statuses. Available Codes and Standards are: CNBC-95-ULS, CNBC-95-WSD, CNBC-05-ULS, CAN/CSA-S6-00, CAN/CSA-S37-01, AASHTO-LRFD-04, ASCE-02-SD, and ASCE-02-ASD.

Statuses are automatically initialized when the **Load Combination Generator** is used to generate required load combinations per selected code.

If a design is not required, use the general statuses that are listed under the **No code** root, namely *Not required*, *Analysis only*, *Ultimate*, *Service*, *Fatigue*, *Instantaneous deflection*, *Long-term deflection*, and *Mass*.

The table below describes the statuses that are common for all codes and standards:

Status	Description
Construction Stage (1)	This status is specific to construction stage load combinations.
Analysis only	Management of analyses. This load combination will be analysed and included in envelopes but will not be considered for a design.
Ultimate	This load combination status means that it is required for a design.
Service	This load combination status applies to serviceability load combinations.
Fatigue	This load combination status applies to fatigue load combinations. They are not generated by VisualDesign but can be specified by users.
Instantaneous Deflection	Instantaneous deflection usually includes live loads only.
Long-Term Deflection	Long-term deflection usually includes permanent loads only.
Mass	This load combination is required for modal analysis and usually includes the structure dead loads and some percentage of snow loads, among others.

Note 1: Construction stages are available for the design of composite beams. The Steel Design module is required for steel/concrete composite beams and the Prestressed Concrete module is required for verifying prestressed composite beams. The Reinforced Concrete Design module is required to design concrete/concrete composite beams.

Specific Statuses

Load Combination Statuses - CAN/CSA-S6-00

The description of load combinations is taken from Table 3.5.1 (a)

Status	Description
ULS no 1	Ultimate Limits States No. 1 $\alpha_D D + \alpha_E E + \alpha_P P + 1.70L$
ULS no 2	Ultimate Limits States No. 2 $\alpha_D D + \alpha_E E + \alpha_P P + 1.60L + 1.15K$
ULS no 3	Ultimate Limits States No. 3 $\alpha_D D + \alpha_E E + \alpha_P P + 1.40L + 1.0K + 0.50W + 0.50V$
ULS no 4	Ultimate Limits States No. 4 $\alpha_D D + \alpha_E E + \alpha_P P + 1.25K + 1.64W$
ULS no 5	Ultimate Limits States No. 5 $\alpha_D D + \alpha_E E + \alpha_P P + 1.0EQ$
ULS no 6	Ultimate Limits States No. 6 $\alpha_D D + \alpha_E E + \alpha_P P + 1.30F$
ULS no 7	Ultimate Limits States No. 7 $\alpha_D D + \alpha_E E + \alpha_P P + 0.9W + 1.30A$
ULS no 8	Ultimate Limits States No. 8 $\alpha_D D + \alpha_E E + \alpha_P P + 1.0H$
ULS no 9	Ultimate Limits States No. 9 $1.35D + \alpha_E E + \alpha_P P$
FLS no 1	Fatigue Limits States 1.0D + 1.0E+ 1.0P + 1.0L
SLS no 1	Serviceability Limits States No. 1 1.0D + 1.0E+ 1.0P + 0.9L + 0.8K + 1.0S
SLS no 2	Serviceability Limits States No. 2 0.9L

Load Cases

A: ice accretion load;
D: dead load;
E: loads due to earth pressure and hydrostatic pressure including surcharges other than dead load;
F: loads due to stream pressure and ice forces, or debris torrents;
H: collision load arising from highway vehicles or vessels;
K: all strains, deformations, displacements, and their effects, including the effects of the restraint and those of friction or stiffness in bearings. Strains and deformations include those due to temperature change and temperature differential, concrete shrinkage and creep; but not elastic strains;
L: live load, including dynamic load allowance when applicable, based on CL-625 Truck or Lane;
P: secondary prestress effects;
EQ: earthquake load;
S: load due to differential settlement and/or movement of the foundation;
V: wind load on traffic;
W: wind load on structure;

Load Combination Statuses - AASHTO-LRFD-04

The description of load combinations is taken from Table 3.4.1-1 *Load Combinations and Load Factors.*

N. B. To lighten the table, some load cases have been grouped together according to AASHTO Standard Table 3.4.1-1, unless otherwise noted. The following groups are used in this table (the definition of load case codes is given below):

D: DC, DD, DW, EH, EV, ES, EL

L: LL, IM, CE, BR, PL, LS

T: TU, CR, SH

N. B.: Refer to Table 3.4.1-2 for the definition of load factors for permanent loads, γp.

Status	Description
STRENGTH I	γPD + 1.75L + 1.0WA + 1.0FR + (0.50 and 1.20)T + γTGTG + γSESE
STRENGTH II	γPD + 1.35L + 1.0WA + 1.0FR + (0.50 and 1.20)T + γTGTG + γSESE
STRENGTH III	γPD + 1.0WA + 1.4WS + 1.0FR + (0.50 and 1.20)T + γTGTG + γSESE

Status	Description
STRENGTH IV	γP (EH+EV+ES+DW+DC) + 1.0WA + 1.0FR + (0.50 and 1.20)T and 1.5(EH+EV+ES+DW+DC) + 1.0WA + 1.0FR + (0.50 and 1.20)T
STRENGTH V	γPD + 1.35L + 1.0WA + 0.4WS + 1.0WL + 1.0FR + (0.50 and 1.20)T + γTGTG + γSESE
EXTREME EVENT I	γPD + γEQL + 1.0WA + 1.0FR +1.0EQ
EXTREME EVENT II	γPD + 0.50L + 1.0WA + 1.0FR + (1.0IC or 1.0CT or 1.0CV)
SERVICE I	1.0D + 1.0L + 1.0WA + 0.3WS + 1.0WL + 1.0FR + (1.0 and 1.20)T + γTGTG + γSESE
SERVICE II	1.0D + 1.3L + 1.0WA + 1.0FR + (1.0 and 1.20)T
SERVICE III	1.0D + 0.8L + 1.0WA + 1.0FR + (1.0 and 1.20)T + γTGTG + γSESE
SERVICE IV	1.0D + 1.0WA + 0.7WS + 1.0FR + (1.0 and 1.20)T + 1.0SE
FATIGUE	0.75 (LL, IM & CE)

Load Cases

PERMANENT LOADS

DD: downdrag;
DC: dead load of structural components and non-structural attachments;
DW: dead load of wearing surfaces and utilities;
EH: horizontal earth pressure load;
EL: accumulated locked-in force effects resulting from the construction process, including the secondary forces from post-tensioning;
ES: earth surcharge load;
EV: vertical pressure from dead load of earth fill.

TRANSIENT LOADS

BR: vehicular braking force;
CE: vehicular centrifugal force;
CR: creep;
CT: vehicular collision force;
CV: vessel collision force;
EQ: earthquake;
FR: friction;
IC: ice load;
IM: vehicular dynamic load allowance;
LL: vehicular live load;
LS: live load surcharge;
PL: pedestrian live load;
SE: settlement;
SH: shrinkage;
TG: temperature gradient;
TU: uniform temperature;
WA: water load and stream pressure;
WL: wind on live load;
WS: wind load on structure.

Load Combination Statuses - ASCE-7-02 SD

The description of load combinations is taken from ASCE-7-02, Section 2.3 "Combining Factored Loads Using Strength Design".

Status	Description
Ultimate 1	1.4 (D + F)
Ultimate 2	1.2 (D+F+T) + 1.6 (L+H) + 0.5 (Lr or S or R)
Ultimate 3 (1)	1.2D + 1.6 (Lr or S or R) + (L or 0.8W)
Ultimate 4 (1)	1.2D + 1.6W + L + 0.5 (Lr or S or R)
Ultimate 5 (1)	1.2D + 1.0E + L + 0.2S
Ultimate 6 (2)	0.9D + 1.6W + 1.6H
Ultimate 7 (2)	0.9D + 1.0E + 1.6H
Service 1*	D + F
Service 2*	D + H + F + L + T
Service 3*	D + H + F + (Lr or S or R)
Service 4*	D + H + F + 0.75 (L + T) + 0.75 (Lr or S or R)

Status	Description
Service 5*	D + H + F + W
Service 6*	D + H + F + 0.75 (L + W) + 0.75 (Lr or S or R)
Service 7*	0.6D + W + H
Fatigue	This status is available but VisualDesign does not automatically generate Fatigue load combinations for this code. Users must define them.

Note 1: The load factor on L in combinations (3), (4), and (5) is permitted to equal 0.5 for all occupancies in which L in Table 4.1 is less than or equal to 100 psf, with the exception of garages or areas occupied as places of public assembly.

Note 2: The load factor on H shall be set equal to zero in combinations (6) and (7) if the structural action due to H counteracts that due to W or E. Where lateral earth pressure provides resistance to structural actions from other forces, it shall not be included in H but shall be included in the design resistance.

*: These load combinations can be generated by VisualDesign, based on the ASD method, even if they are not required per this code, because the software needs them to calculate forces and stresses under serviceability limits states, for some particular applications.

Load Cases

D: dead load;
Di: weight of ice;
E: earthquake load;
F: load due to fluids with well-defined pressures and maximum heights;
Fa: flood load;
H: load due to lateral earth pressure, ground water pressure, or pressure of bulk materials;
L: live load;
Lr: roof live load;
R: rain load;
S: snow load;
T: self-straining force;
W: wind load;
Wi: wind-on-ice determined in accordance with Section 10.

2.3.3 Flood Load:

When a structure is located in a flood zone, the following load combinations shall be considered:

In V-Zones or Coastal A-Zones, 1.6*W* in combinations **Ultimate 4** and **Ultimate 6** shall be replaced by 1.6*W* + 2.0*Fa*.

In noncoastal A-Zones, 1.6*W* in combinations **Ultimate 4** and **Ultimate 6** shall be replaced by 0.8*W* + 1.0*Fa*.

2.4.3 Atmospheric Ice:

When a structure is subjected to atmospheric ice and wind-on-ice loads, the following load combinations shall be considered:

0.5 (Lr or S or R) in combination **Ultimate 2** shall be replaced by 0.2Di + 0.5S.

1.6W + 0.5 (Lr or S or R) in combination **Ultimate 4** shall be replaced by Di + Wi + 0.5S.

1.6W in combination **Ultimate 6** shall be replaced by Di + Wi.

Load Combination Statuses - ASCE-7-02 ASD

The description of load combinations is taken from ASCE-7-02, Section 2.4 "Combining Nominal Loads Using Allowable Stress Design".

Status	Description
Service 1	D + F
Service 2	D + H + F + L + T
Service 3	D + H + F + (Lr or S or R)
Service 4	D + H + F + 0.75 (L + T) + 0.75 (Lr or S or R)
Service 5	D + H + F + (W or 0.7E)
Service 6	D + H + F + 0.75 (W or 0.7E) + 0.75L + 0.75 (Lr or S or R)
Service 7	0.6D + W + H
Service 8	0.6D + 0.7E + H
Fatigue	This status is available but VisualDesign does not automatically generate Fatigue load combinations for this code. Users must define them.

Load Cases

D: dead load
Di: weight of ice
E: earthquake load
F: load due to fluids with well-defined pressures and maximum heights
Fa: flood load
H: load due to lateral earth pressure, ground water pressure, or pressure of bulk materials
L: live load
Lr: roof live load
R: rain load
S: snow load
T: self-straining force
W: wind load
Wi: wind-on-ice determined in accordance with Section 10.

2.4.2 Flood Load:

When a structure is located in a flood zone, the following load combinations shall be considered:

In V-Zones or Coastal A-Zones, 1.5*Fa* shall be added to other loads in combinations **Service 5**, **Service 6**, and **Service 7** and *E* shall be equal to zero in combinations **Service 5** and **Service 6**.

In noncoastal A-Zones, 0.75*Fa* shall be added to other loads in combinations **Service 5**, **Service 6**, and **Service 7** and *E* shall be equal to zero in combinations **Service 5** and **Service 6**.

2.4.3 Atmospheric Ice:

When a structure is subjected to atmospheric ice and wind-on-ice loads, the following load combinations shall be considered:

0.7Di shall be added to combination **Service 2**.

(Lr or S or R) in combination **Service 3** shall be replaced by 0.7Di + 0.7Wi + S.

W in combination **Service 7** shall be replaced by 0.7Di + 0.7Wi.

Load Combination Generator

Load Combinations Generator

This wizard allows generating load combinations according to a specific construction code and with the user-defined load cases (live, dead, wind, seismic, etc.). Access this tool by selecting **Load Combinations/ Automatic Generation** under **Loads** menu.

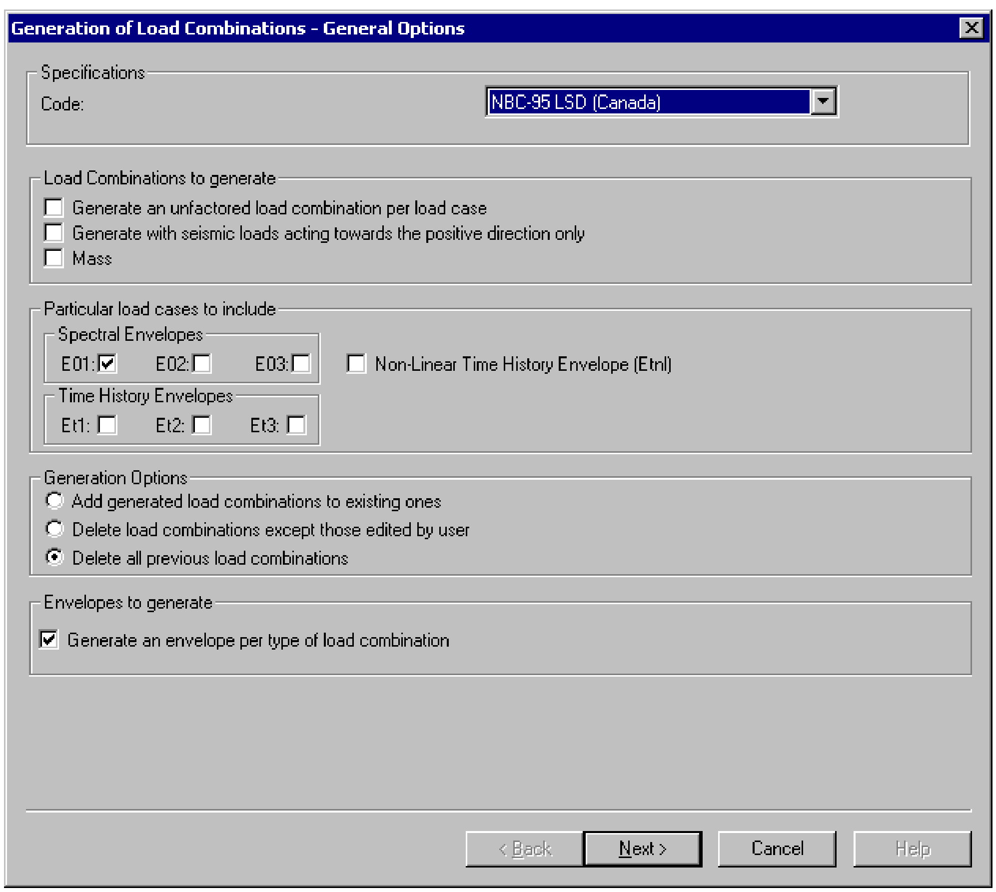
The dialog box is composed of three pages: **General Options**, **Specific Options** and **Selections**.

Bridge Evaluation module only (S6-00):

An additional page is included in the generator for the calculation of load factors:

General Options Page

The **General Options** page concerns general options, as the building code that will be used to generate appropriate load combinations. It also includes generation options that apply to all building codes and standards.



Here is the description of fields included in this page:

Heading	Description
Specifications	**Code**: Specify the construction code to be used for the generation of load combinations. The following codes and standards are available: NBC 1995 (Canada) – LSD (Limit States Design); NBC 1995 (Canada) – WSD (Working Stress Design); CAN/CSA-S6-00: Highway and Bridge Design Code; CAN/CSA-S37-01: Tower Design; ASCE-7-02 SD: American standard for Strength Design; ASCE-7-02 ASD: American standard for Allowable Stress Design; AASHTO-LRFD-04 (USA): Highway and Bridge Design Code; CNB-2005 (Canada)
Load Combinations to be Generated	Activate options by checking boxes. Available options are: Generate an unfactored load combination for each load case; Generate with seismic loads acting towards the positive direction only (1); Mass.
Particular Load Cases to be Included	
Spectral Envelopes	Specify spectral envelopes to be included in generated load combinations: E01, E02, and/or E03. Refer to *Linear Seismic Directions* for specifying envelopes that correspond to seismic directions.
Time History Envelopes	Specify time history envelopes to be included in generated load combinations: Et1, Et2, and/or Et3.
Non-linear Time History Envelope	Specify non-linear time history envelope Etnl to be included in generated load combinations
Generation Options	**Add generated load combinations to existing ones**: The generated load combinations are simply added to the previously defined ones. **Delete load combinations except those edited by user**: Only load combinations that have been previously generated with the wizard will be destroyed. **Delete all previous load combinations**: All the existing load combinations included in the project are destroyed before creating new load combinations.
Envelopes to be generated	Generate an envelope per type of load combination (Strength, Ultimate, ULS, Service, Fatigue, etc.)

Note 1:

Generation of Seismic Loads:

For Generation Modules – Culverts and Piers, Abutments: VisualDesign™ will generate positive AND negative seismic load cases if this option is not activated. This is true for ULS, LRFD, and ASD codes.

Press the "Next" button to reach the second page of the Wizard, **Specific Options**, which includes generation options that apply to the selected building code or standard.

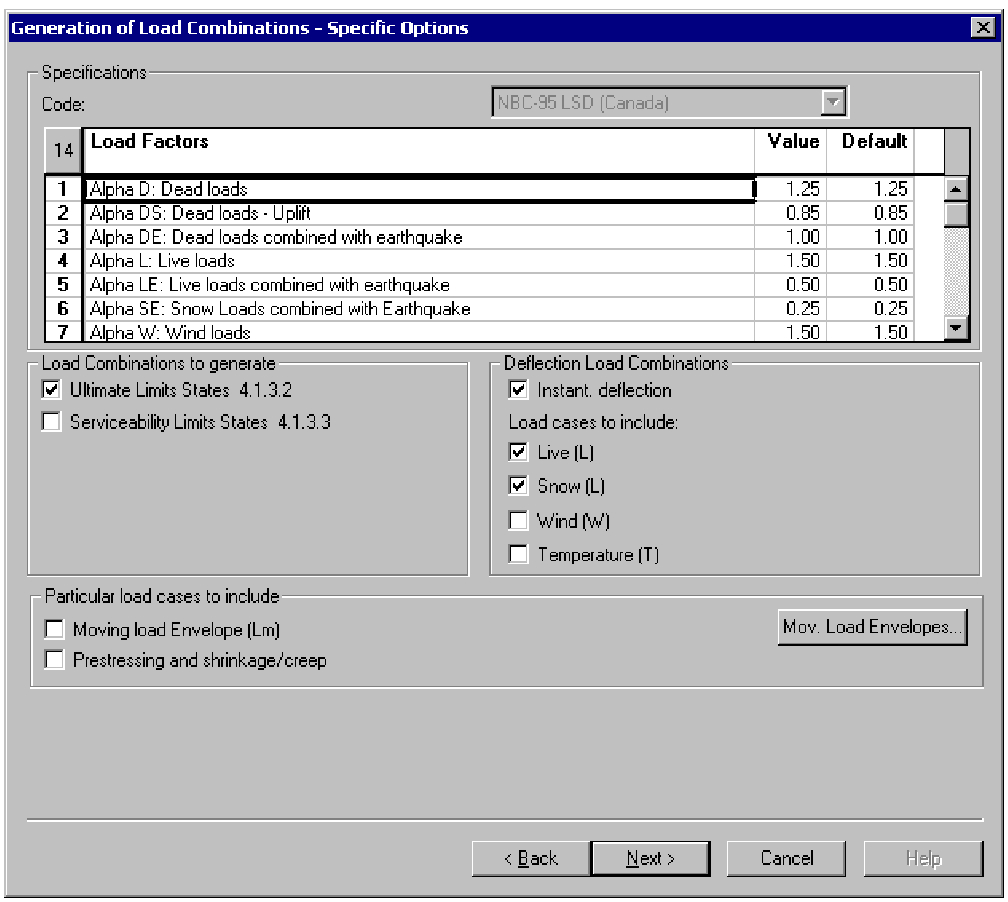
See also

Linear Seismic Directions (Creating envelopes E01, E02, E03)

Specific Options Page

The **Specific Options** page will appear on screen as soon at you click the "Next" button posted in the bottom of the first page. This page is composed of specific generation options that apply to the selected building code.

Use the "Back" button to go back to previous page.



Here is the description of fields included in this page:

Topic	Description
Specifications	**Code**: Shaded field that shows the selected building code or standard.
Load Factors spreadsheet (1)	Use load factor default values (alpha, gamma and psy) listed in the "Default" column or enter your own load factors in the "Value" column. See note 1 below, to know more about the gamma factor of NBC building code and ASCE standard.
Load Combinations to be Generated	By default, all load combinations are activated and will be generated. To disable some, uncheck boxes. Refer to: Load Combinations - General Load Combinations - CAN/CSA- S6-00 Load Combinations - AASHTO-04 Load Combinations - ASCE-7-02 SD Load Combinations - ASCE-7-02 ASD
Deflection Load Combinations	If available for chosen code, this option generates required load combinations fro checking the maximum *Instantaneous Deflection* deflections. Please activate the live loads to be considered in these load combinations.
Particular Load Cases to be Included (2)	If available for selected building code, specify particular load cases to be included in generated load combinations: Moving Load envelope (Lm) Prestressing load and shrinkage/creep loads Combination of seismic envelopes: 100%E1 + 30%E2.
The "Moving Load Envelopes" button	This button calls up the **Definition of Moving Load Envelopes** spreadsheet. This spreadsheet allows getting the default load combinations that shall be included in each moving load envelope, as required per selected code. To know more, please go to Definition of Moving Load Envelopes.
The "Bridge Evaluation" button	For CAN/CSA-S6-00 Standard Only: This button, located in the lower part of the page, gives access to a dialog box. Parameters shall be specified for computing load factors required for a bridge evaluation. Refer to topic Load Combinations –Bridge Evaluation.

Press the "Next" button to go the last page of the Wizard, which is the **Selections** page.

Use the "Back" button to go back to previous page.

Note 1: Importance Factor (gamma)

This factor affects some factored loads other than dead loads, in order to consider the consequences of collapse as related to the use and occupancy of the building.

NBC 1995 (Canada): The factor is normally 1.0 or 0.8.

ASCE-7 (US): the factor is equal to 0.5, except for places of public assembly, live loads in excess of 100 psf, and for garage live load where the factor is 1.0.

Note 2: Combination of Seismic Envelopes

Two spectral envelopes will have to be specified in the **Linear Seismic Directions** spreadsheet. Each envelope will be assigned to a seismic direction. If this option is activated in the **Specific Options** page of the **Wizard**, VisualDesign will generate the load combinations that include 100% E01 + 30% E02, and vice-versa, instead of including E01 only.

See also

Load Combinations Generator

Definition of Moving Load Envelopes

The Selections tab

Load Combinations for Bridge Evaluation

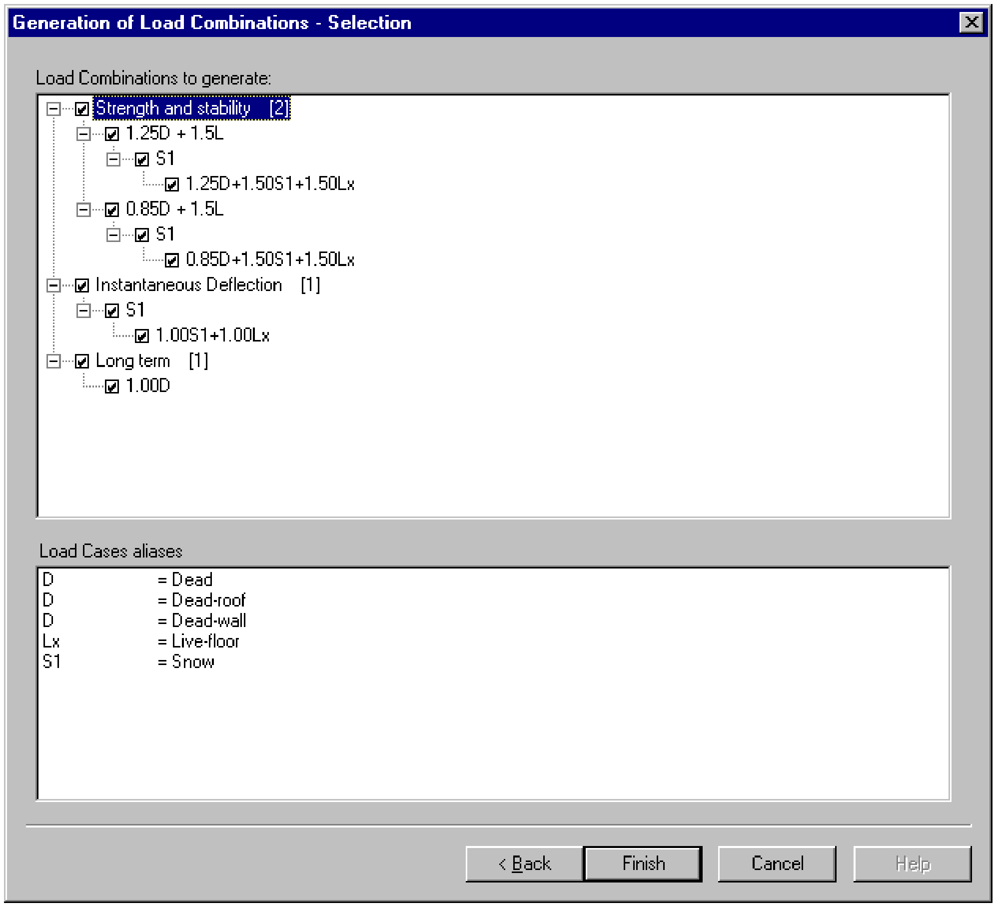
Linear Seismic Directions spreadsheet

Selections Page

In this dialog box, a selection tree is displayed which includes all the load combinations that will be generated. To simplify the writing of load combination names and descriptions, aliases (short names) are generated and described in this page.

These aliases will be automatically added to your load cases description in the **Loads Definition** spreadsheet so that you can identify them easily. Dead loads are consecutively named D1, D2, …,Dn, if they are not part of the same family, and so are wind loads (W1, W2…) and temperature loads (T1, T2…). Since live loads are combined, they are represented like binary numbers: Lxoxx means L1+L3+L4.

Topic	Description
Load Combinations to be Generated	When you press the "Finish" button, look at the expanded tree including all load combinations to be generated. You are allowed to select and deselect the tree's branches and leaves in order to select the load combinations that you wish to generate.
Load case aliases	This list shows load case names' aliases used for load combinations nomenclature.



At the push of the "Finish" button, selected load combinations are automatically generated and the **Load Combinations Definition** spreadsheet is opened. You can modify them as you wish or exit the spreadsheet.

See also

The Options tab

Load Combinations Generator

Load Case Families

Moving Load Envelopes

The spreadsheet **Definition of Moving Load Envelopes**, which is available in the **Loads / Moving Loads** menu, allows activating and defining moving load envelopes that will be included when generating load combinations with the Wizard and when the Moving Load Analysis will be launched. Furthermore, it allows specifying the type of 2D axle factors that must be used for each envelope.

This spreadsheet can also be open with the button "Mov. Load Envelopes", posted in the **Moving Load Analysis** dialog box and in the **Specific Options** page of the **Load Combination Wizard**.

Here is the description of columns included in this spreadsheet:

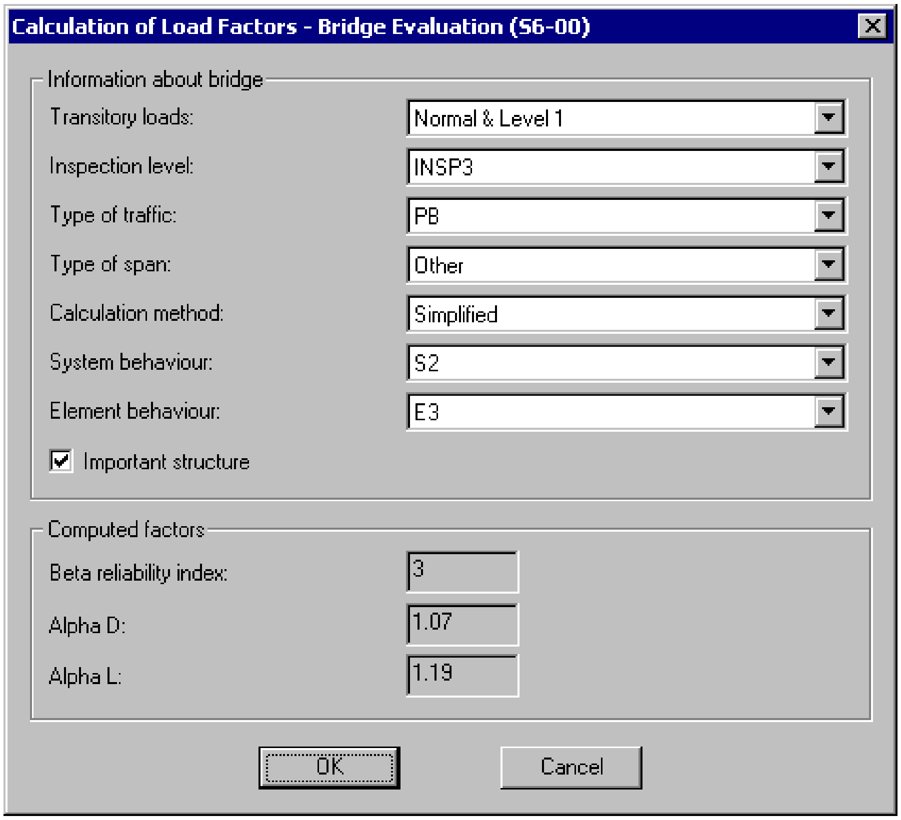
Column	Description	Editing
ID	Automatically calculated	No
Number	Number of moving load envelope (Lm01 to Lm10).	Single click
To be analysed	Activate this box [x] to include this moving load envelope in the moving load analysis and/or in generated load combinations when using the Wizard.	Double click or Space bar
2D Axle Factors to be used	This column represents the type of 2D axle factor that applies to this envelope, for a 2D project, according to the chosen code. 2D axle factors are defined in the 2D Axle Factors spreadsheet, for a single loaded lane and for many loaded lanes.	Double click
Type of Load Combinations	According to selected code, activate [x] or disable the box [x] to include or not the envelope when generating this type of load combination according to this code.	Double click or Space bar

Computation of Load Factors for Bridge Evaluation

If you are evaluating a bridge according to code S6-00, the **Load Combinations Generator** will generate the required load combinations.

To do so, select Code S6-00 in the **General Options** page, press "Next". In the **Specific Options** page, press the "Bridge Evaluation" button to call up the **Calculation of Load Factors** dialog box.

Enter the required parameters in this specific dialog box. The reliability Index beta will be calculated, along with αD and αL load factors.



The table below describes the parameters included in the dialog box:

Field	Description
Information on the bridge	
Transitory loads	Specify the type of transitory loads applied to the bridge: Alternative, Level 1, Level 2 or Level 3.
Inspection Level	Choose among INSP1, INSP2 or INSP3.
Traffic Type	Choose among Normal, PA, PB, PC or PS.
Type of span	Specify the type of span: Short or Other.
Calculation Method	Choose a calculation method: Statically determinate, Sophisticated or Simplified.
System Behaviour	Specify the category S1, S2 or S3 that describes the system behaviour.
Element Behaviour	Specify the category E1, E2 or E3 that describes the element behaviour.

Field	Description
Important Structure	Check this option if the bridge is considered as an important structure according to code S6-00.
Calculated Load Factors	
Reliability Index Beta	According to the parameters that you entered, VisualDesign™ calculates a reliability index beta and load factors alpha D and alpha L.
Alpha D	Calculated load factor that will be applied to permanent loads.
Alpha L.	Calculated load factor that will be applied to live loads.

See also

Specific Options page of Load Combination Wizard
Bridge Evaluation Module
Deterioration of Rebars and Cables

Envelopes

Definition of Envelopes

To define envelopes, select headings **Envelopes/Definition** under **Loads** menu. The **Envelopes** dialog box is composed of two tabs: **Envelopes** and **List of Envelopes**. Each one of these tabs includes a spreadsheet to facilitate the data entries.

See also

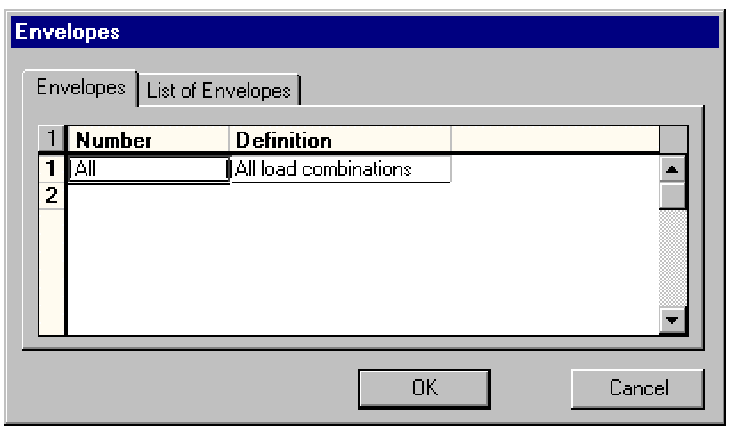
The Envelopes Tab

The List of Envelopes Tab

Copying an Envelope and Load Combinations

The Envelopes Tab

The **Envelopes** tab is part of the **Envelopes** dialog box, which is accessible through the **Loads** menu / **Envelopes**. Insert a line and enter a name for each envelope that you wish to study in your project. Then, select the **List of Envelopes** tab and integrate load combinations that are part of each envelope.



Group: Envelope data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Definition	Comment	Single click

See also

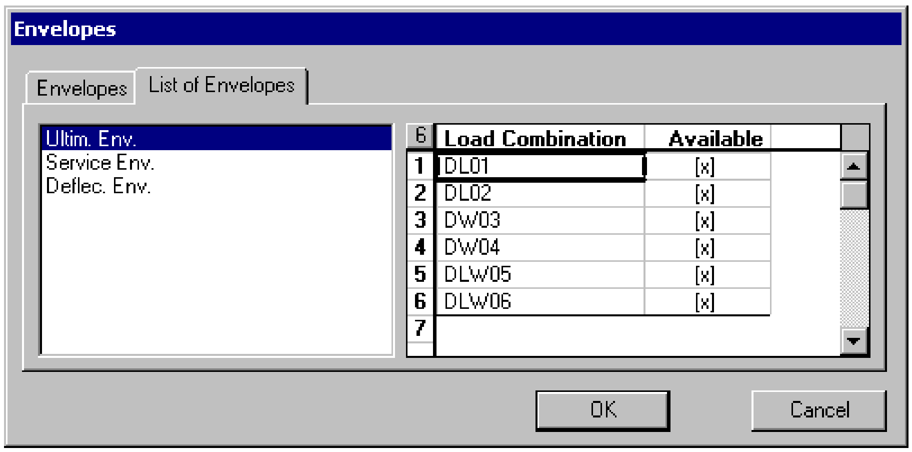
The List of Envelopes Tab

Copying an Envelope and Load Combinations

The List of Envelopes Tab

The **List of Envelopes** tab is part of the **Envelopes** dialog box.

The left part of the dialog box is a list box that contains the name of envelopes. Highlight each envelope and, in the right part of the dialog box, enter the number of lines that correspond to the number of load combinations that are included in the envelope. Then, select each load combination in the "Load Combination" list box. The "Available" column informs the user about the availability of this load combination.



Group: Envelope: (title)

Column	Description	Editing
ID	Calculated automatically	No
Load Combination	Select load combination in the list box.	Double-click
Available	This column informs the user about the availability of this load combination.	Single click

See also

Copying an Envelope and Load Combinations

Copying an Envelope Definition along with Load Combinations

To simultaneously copy an envelope along with load combinations that are part of this envelope, use function **Duplicate** available in the spreadsheet's contextual menu.

Procedure:

- Open the **Definition of Envelopes** spreadsheet.
- Select the line that corresponds to the envelope you want to copy.
- Right click and choose function **Duplicate** in contextual menu.

Chapter

5

STATIC ANALYSIS & FINITE ELEMENTS

TABLE OF CONTENTS

Chapter 5 Static Analysis & Finite Elements

General

The Base Module

The Basic module includes the Static analysis and performs linear and non-linear static analysis of structures composed of members, floors, and triangular or rectangular finite elements. There is no limit for the number of elements and nodes included in the structural model. All types of loads are permitted (concentrated, distributed, trapezoidal, pressure on plates or temperature loads). Furthermore, there is no limit either for the number of load combinations and load cases.

Restrictions

VisualDesign™ cannot perform an analysis if the structure does not include support nodes or if shapes or materials are missing.

VisualDesign™ cannot perform a static analysis if seismic envelopes or moving load envelopes are integrated into load combinations and have not been analyzed beforehand.

Types of Static Analyses

Users must specify the type of static analysis to be run, in the **Analysis** tab of **Project Configuration** dialog box:

Linear Static Analysis

Forces and displacements are obtained without considering second order effects such as P-delta effects.

Non-Linear Static Analysis

Forces and displacements are obtained considering second order effects such as P-delta effects. This type of analysis is a non-linear analysis in the geometric point of view as opposed to a dynamic analysis, which is non-linear analysis in the material point of view where material can reach plasticity.

Static Analysis with Release (Members and/or Supports)

This type of analysis is like a linear static analysis, but is an iterative one: When VisualDesign encounters members that are specified with released conditions, it corrects appropriate members' degrees of freedom and launches another linear analysis until it reaches convergence. Refer to this procedure to learn more: Modeling and analysing a slab-on-grade.

See also

Analysis tab (Project Configuration)

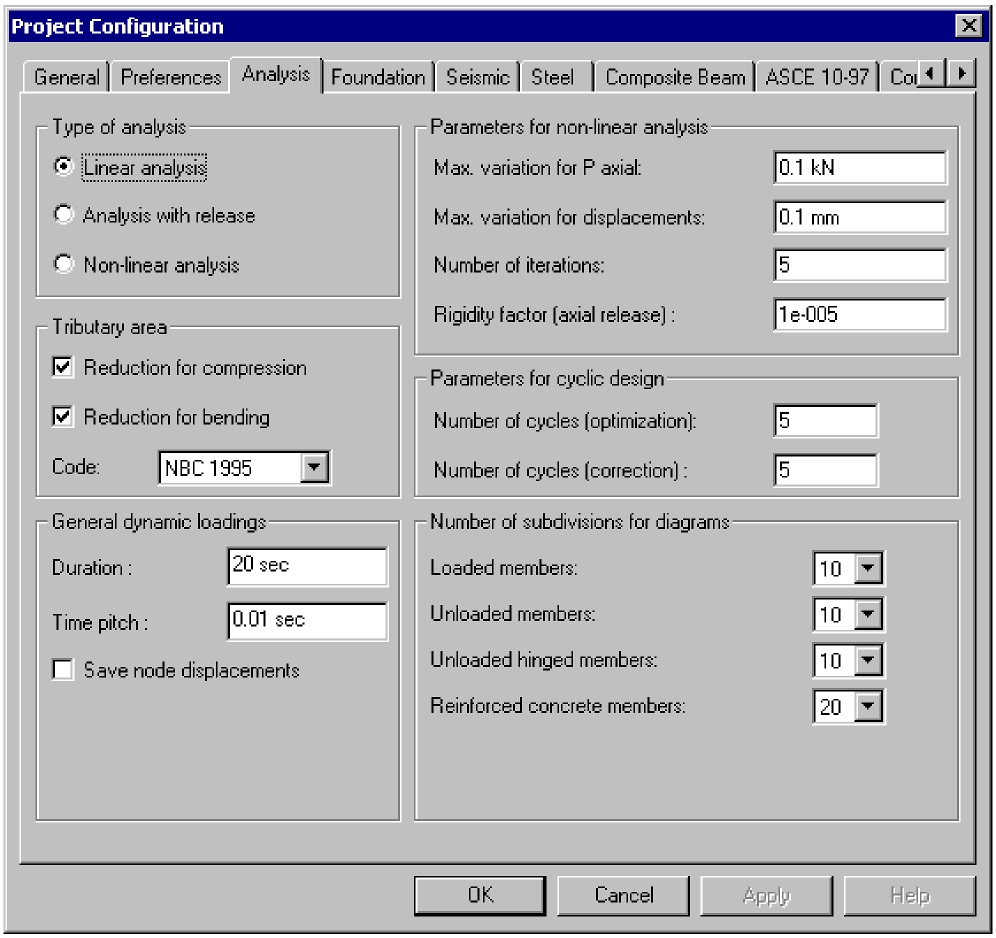
Released Supports (Chapter 2)

Released Members (Chapter 2)
Display Released Members

Project Configuration

Analysis Tab

Specify the type of static analysis to be run (linear, non-linear, or with release), parameters for non-linear analysis, subdivision of members for the display of internal forces, reduction factor for tributary area, and parameters for a general dynamic (transient) analysis.



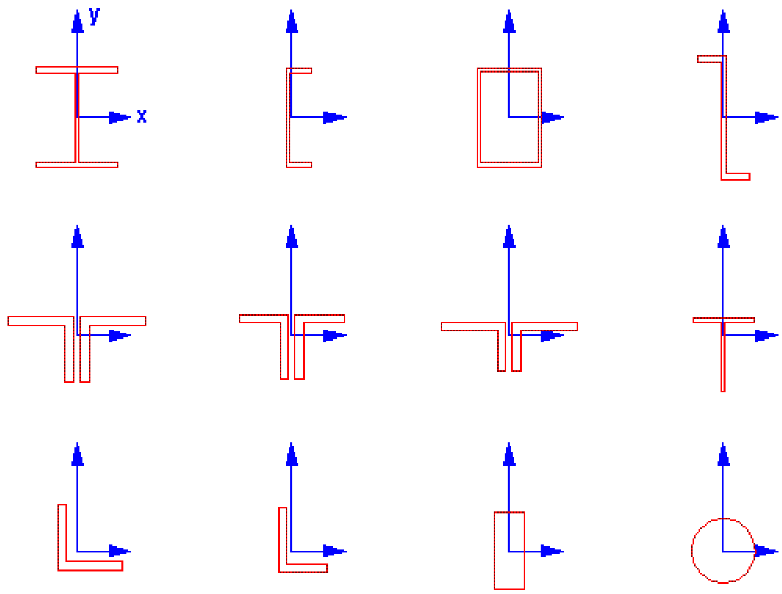
Description of the dialog box:

Field	Description
Type of analysis	Activate a linear or non-linear static analysis. The static analysis with release is required if supports or members need to be released during analysis. Refer to topics: Types of Static Analysis and Tension-only Members

Field	Description
Tributary Area	Specify the type of reduction factor that will be applied to tributary areas: Compression or Bending. With the drop-down list box, select the code that will be apply to reduce overload.
Parameters for Non-linear Analysis	Parameters are shaded if you activated a linear analysis. If you activated an analysis "with release", only the number of iterations can be specified for said analysis.
Max. variation for P axial	The non-linear analysis will end when this variation will fall below this value.
Max. variation for displacements	This tolerance is applied to displacements of axially released supports only.
Number of iterations	The non-linear analysis or the one considering release will end when the specified maximum number of iterations will be reached.
Rigidity factor (axial release)	Specify a rigidity factor for axially released members.
Parameters for cyclic design	
Number of cycles (optimization)	Number of cycles for optimization when designing members.
Number of cycles (correction)	When the optimized number of cycles is reached, members that have not been optimized will be evaluated with the correction mode.
Number of subdivisions for the diagrams	Number of subdivisions applied to all members no matter the load condition. It can be specified for loaded beams, unloaded beams, unloaded pinned beams, concrete members and rectangular plates.
General Dynamic Loadings	
Duration	Fix a maximum time for the application of this type of dynamic loading.
Time pitch	Specify the time pitch. Make sure that "dti" is larger than the time pitch otherwise there will be a warning. Refer to topic General dynamic load diagrams for more details.
Save node displacements	Save the time responses of node displacements in the database (Project_Name.vr1). See the note below.

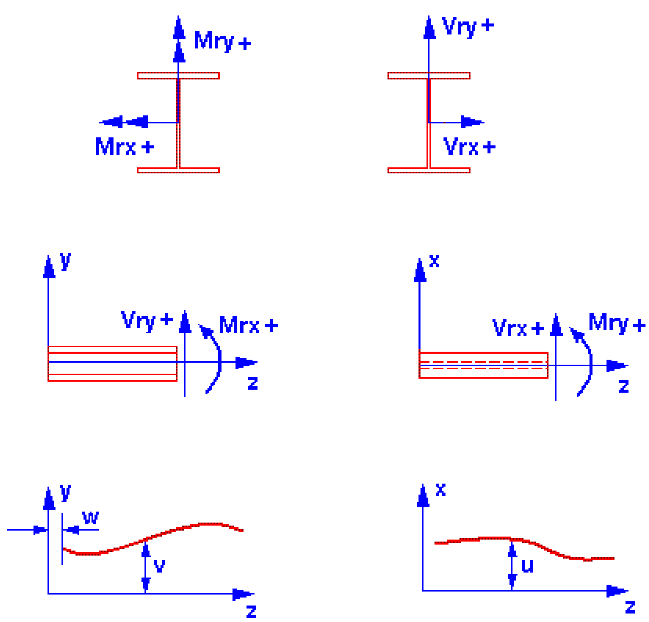
Convention - Forces in Members

Sections' strong and weak axes:

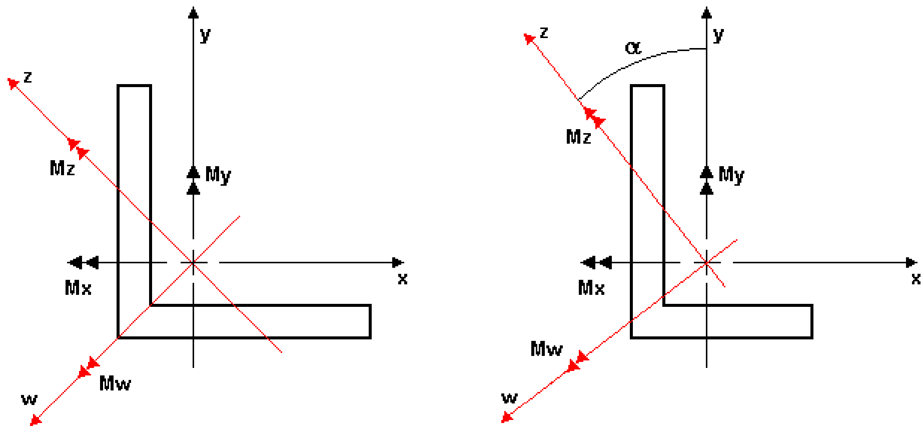


Forces and resistances

Resistances, forces and deflections are given in accordance to the member local axes system (x, y, z). Local z-axis is longitudinal to the member and points towards node j.



For steel angles, Mw and Mz bending moments are always transformed into orthogonal axis system when the design is completed.

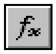


See also

Major/minor and Orthogonal Axis Systems

Static Analysis

Static Analysis Dialog Box



The "Static Analysis" icon of Tools toolbar

Open the **Static Analysis** dialog box by clicking the "Analysis" icon on Tools toolbar or by selecting **Static Analysis** from **Analysis** menu.

The upper part of the **Static Analysis** dialog box displays the project title (as defined in **Project Configuration** under **File** menu)**.** You will also find the number of nodes, members, plates, and floors as well as the number of load types and load combinations that you defined.

Press the "Analyse" button to launch the analysis.

DATE AND TIME

VisualDesign displays the date of analysis in the **Static Analysis** dialog box. You will also find the time it was launched and the time it ended in the upper part and lower part of the dialog box.

See also

Accessing Analysis Results

Null pivot in the stiffness matrix

Activating a Static Analysis when a Spectral Analysis is called for

Starting a Static Analysis

- Click the "Analyse" icon on Tools toolbar or go to **Analysis / Static**.
- Click the "Analyse" button in the **Analysis** dialog box.

Instability Messages

The load combination has not reached the specified level of precision.

This message appears in the **Analysis** dialog box during analysis process when VisualDesign could not reach the convergence criterion that is specified for P axial, in the **Analysis** tab of **Project Configuration**.

Results are not valid because stress/strain compatibility has not being reached.

It could be caused by instability in your model. Look at the magnitude of applied loads. Look carefully at support restraints and look at hinged members (too many hinges = instability). Look at deflections to locate the problem.

Null Pivot in the Stiffness Matrix

During an analysis, this message indicates that a null pivot was detected in the stiffness matrix. It is a strong indication that there is an instability problem within the model. The node number that is given with the message is a hint to locate the instability source.

Good to know:

Don't forget that VisualDesign™ is a 3D program. If you modeled a 2D structure, you must fix the third direction in order to get a stable structure.

A 3D structure that is stable could become unstable if lateral forces are applied and if you ran a non-linear analysis. It means that instability was not detected under gravity loads. Consequently, we recommend that you always check the structure behaviour under lateral loads.

Always check the structure deflection, if possible. An extreme displacement is an important indication that a mechanism is present in the structure. Strengthen the structure where the mechanism took place.

If you have gone through many tries but haven't resolve the problem, here are a couple of suggestions to make sure that the model is stable:

- Create a unit load case on each node of the model. Loads should be applied in the three orthogonal directions (x, y, z, Mx, My and Mz).
  - Create a load combination that corresponds to this load case.
  - Configure your project for a static analysis (**File** / **Project Configuration** / Analysis tab / "Type of analysis").
  - Run a static analysis. If you obtain the null pivot message, the model is still unstable.

- Observe the model deflections to detect instabilities that could be present. Huge deflections are a sign of instability, inferior structural stiffness or excessive loads.

- If there are some Built-Up sections in your project, change all these sections for symmetrical shapes or pre-defined shapes such as HSS, to make sure that instability is not caused by data entry.

Deflections

Shear Energy - Deflection Calculation

VisualDesign™ integrates the shear energy of a deformed section into the calculation of the total deflection of a member. The program adds the deflection due to shear energy to the deflection caused by bending.

Total Deflection = Deflection$|_{\text{BENDING}}$ + Deflection$|_{\text{TORSION}}$.

Deflection Due to Shear:

The deformation due to the shear force is equal to the shear force V divided by the product of the area As and G.

$$\text{Deformation} = \gamma = \frac{V}{(A_s \cdot G)}$$

Where $A_s = K_x \cdot A_{gross}$ K_x is a factor that depends on the type of material and the shape of the section. See Timoshenko for more details.

For a rectangular beam:

$$K_x = K_y = \frac{10 \cdot (1 + \upsilon)}{12 + 11 \cdot \upsilon}$$

The equation for deflection is the integral of the deformation γ.

$$\boxed{\text{Deflection} = \int_0^{\frac{L}{2}} \gamma \, dx} \quad \text{où} \quad \gamma = \frac{V(x)}{A_s \cdot G}$$

$$\text{Then,} \quad \text{Deflection} = \int_0^{\frac{L}{2}} \frac{V(x)}{A_s \cdot G} \, dx$$

Find equation V(x):

at x = 0: V(x) = Vmax

at x = L/2: V(L/2) = 0 and the slope m = 2Vmax/L

$$\text{So,} \quad V(x) = \frac{-2 \cdot V_{max}}{L} \cdot x + V_{max}$$

$$\text{Then, Deflection} = \frac{1}{A_s \cdot G} \cdot \int_0^{\frac{L}{2}} \left(\frac{-2 \cdot V_{max}}{L} \cdot x + V_{max} \right) dx$$

$$\text{Deflection} = \frac{1}{A_s \cdot G} \cdot \left[\frac{(-V)_{max}}{L} \cdot x^2 + V_{max} \cdot x \right]_0^{\frac{L}{2}}$$

$$\boxed{\text{Deflection} = \frac{1}{A_s \cdot G} \cdot \left(\frac{V_{max}}{4} \cdot L \right)}$$

EXAMPLE 1

Calculate the total deflection at the centre of a rectangular beam, simply supported and loaded with a uniform live load.

Beam 405mm x 405mm, simply supported
L = 5 meters
Distributed live load = 50 kN/m
A gross = 164025mm² (calculated by VisualDesign™)
Linear mass = 399,6 kg/m (calculated by VisualDesign™)
Concrete: 30 MPa
E = 26621 MPa (calculated by VisualDesign™)
G = 11092 MPa (calculated by VisualDesign™)
υ = 0,20 (given by VisualDesign™)
I = bd³/12 = 224.2*10exp7
Density = 23.54 kN/m³ (given by VisualDesign™)

Unfactored distributed load:

w = (0.405² * 23.54) + 50 = 53.86 kN/m

So, Vmax = 53.86 * 5 / 2 = 134.65 kN

Calculation of As = Kx * A gross

$$Kx = \frac{10(1+\upsilon)}{12+11\upsilon} = \frac{10*1.2}{12+11*0.20} = 0{,}84507$$

A gross = 164025 mm²

As = 164025 * 0,84507 = 138612.7 mm²

Calculation of the total deflection at centre

$$\Delta centre = \Delta bending + \Delta shear$$

$$= \frac{5\omega L^4}{384EI} + \frac{1}{As*G} * \frac{Vmax*L}{4}$$

$$= \frac{(5*53.86*5000^4)}{384*26621*224.2*10^7} + \frac{(134650*5000)}{138612.67*11092*4}$$

$$= 7.34387mm + 0.10947mm$$

$$= 7.4533mm$$

VisualDesign™ Static Analysis Results: 7.45 mm

EXAMPLE 2: Reduced Stiffness Due to Cracked Concrete.

This is the same example as above except for the elements stiffness that needs to be reduced because of the cracked concrete. The ratio of reduced stiffness is specified in the **Member** tab of **Characteristics of the Member** dialog box. See also the topic Effective Stiffness.

For bending: Ratio for reduction of inertia = 0.8

For torsion/shear: Ratio for reduction of stiffness = 0.5

$$\Delta centre = \Delta bending + \Delta torsion,shear$$

$$= \frac{1}{0.8}\left(\frac{5\omega L^4}{384EI}\right) + \frac{1}{0.5}\left(\frac{1}{As*G} * \frac{Vmax*L}{4}\right)$$

$$= \frac{(5*53.86*5000^4)}{0.8*384*26621*224.2*10^7} + \frac{(134650*5000)}{0.5*138612.67*11092*4}$$

$$= 9.1799mm + 0.21894mm$$

$$= 9.3988mm$$

Results - General

Accessing Analysis Results

As soon as a static analysis is done, VisualDesign™ activates the Load Combination mode. A load combination title can be selected from the scroll list on the Activation toolbar. To look at numerical results, activate a type of element on Elements toolbar and double-click on the element.

To view envelope results, activate the Envelope mode, and then select an envelope title from the scroll list on the Activation toolbar. Activate the type of element on Elements toolbar and double-click on any element of this type.

Alternatively, select one of the load combinations or envelopes spreadsheets under **Results** menu.

See also

Starting a Static Analysis
Static Analysis

Support Results

Results can be viewed by activating the "Load Combination" or "Envelope" mode when the analysis is done. Choose a load combination title from the pull down list of the Activation toolbar.

Select a support and call the **Properties** function. You will then see the **Results** in the **Node** dialog box. Alternatively, you can also double-click on a support to call up the Support Results spreadsheet.

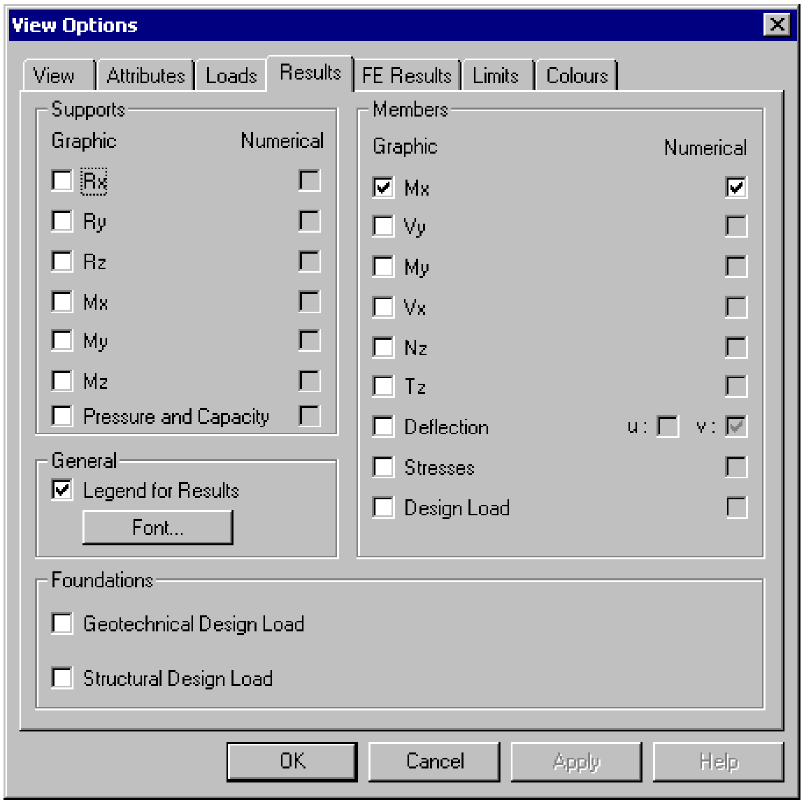
The **Node** tab includes displacement results and the **Support** tab includes support reactions.

See also

Support Characteristics
Support Reactions for a load combination
Support Reactions for an envelope

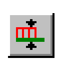
Graphical and Numerical Results

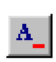
Select the **Results** tab in the **View Options** dialog box and check the box that corresponds to the force, stress or deflection diagrams you want to display on screen. Click [Apply] or OK to exit the dialog box.



- To look at numerical results associated to the diagram, check the box under the "Numerical" heading, in the **Results** tab. You can also click on the Numerical Values icon  of **Diagrams** toolbar.

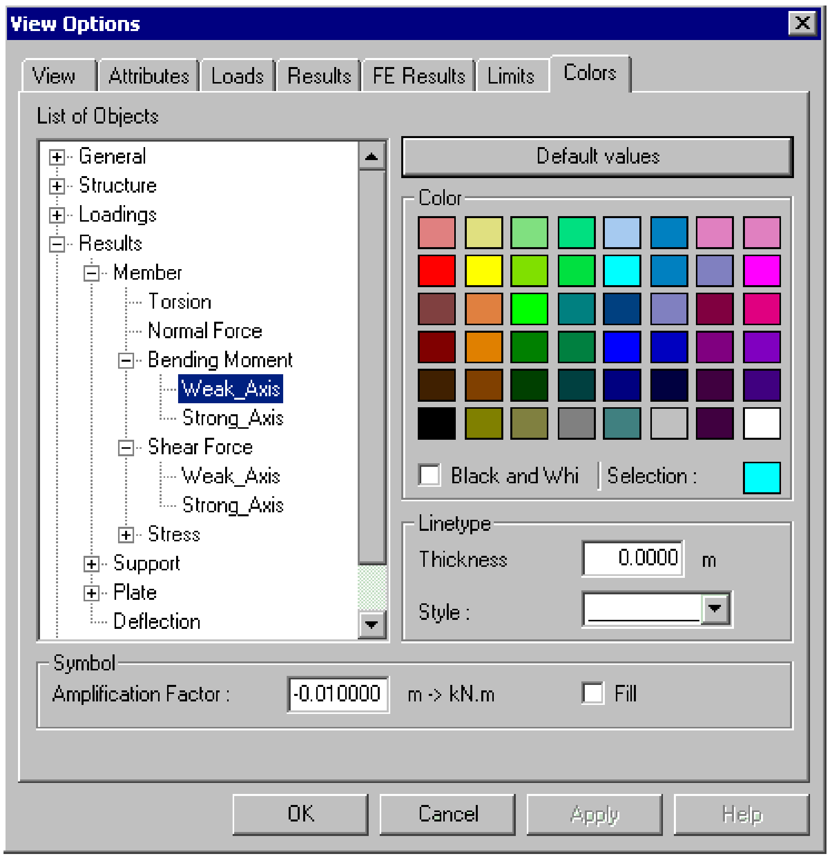
Amplitude of Diagrams and Fonts:

- Use the "Automatic Scaling" icon  of **Diagrams** toolbar if the diagram amplitude is too big or too small.

- Use icons  and  to increase or decrease the displayed text for the activated element on Elements toolbar.

Coloured Results

To modify graphic attributes for results, select the **Colour** tab of **View Options** and expand the *Results* selection tree. With your cursor, highlight a type of results in the list. The colour, for this type of result, will be shown in the "Selection" field. To modify it, click on a coloured square and press OK to exit the dialog box.



See also

Graphical and Numerical Results
The Colour tab of View Options

Load Combination Results

Summary of Load Combinations

Results are not valid if VisualDesign has not reached convergence.

Consult this spreadsheet to look at the obtained precision for each analysed load combination.

This spreadsheet is available through **Results / Load Combinations / Summary**. It is composed of two tabs, *Load Combinations* and *Summary*, which present a summary of results obtained for each load combination such as ΣFx, ΣFy, ΣFz (Sum of applied loads only; envelopes are not considered), maximum displacement of node (node number), message posted after analysis, number of iterations, precision obtained, etc.).

Here is a definition of fields included in these tabs:

Load Combinations tab

Column	Description	Editing
ID	Automatically calculated	No
Number	Number (name) of this load combination	No
Definition	Enter a definition (optional) Ex: 1.25D + 1.5Lx	No
Analysis Message	Message obtained at the end of analysis for this load combination. Ex: *Design OK* or *The load combination has not reached the specified level of precision.*	No
Number of iteration (1)	Non-linear analysis: Number of iterations for this design or non-linear static analysis to attain convergence.	No
Precision obtained (1)	Non-linear analysis: This value represents the maximum variation obtained for P axial.	No

Note (1): These parameters are defined in the **Analysis** tab of **Project Configuration** dialog box, at section *Parameters for non-linear analysis*.

Summary tab

Column	Description	Editing
ID	Automatically calculated	No
Load combination number	Load Combination name or number.	No
ΣFx	Sum of applied forces in the x direction. This result is not available for load combinations that include one envelope or more.	No
ΣFy	Sum of applied forces in the y direction. This result is not available for load combinations that include one envelope or more.	No
ΣFz	Sum of applied forces in the z direction. This result is not available for load combinations that include one envelope or more.	No
Node Number (Max Displ.) (2)	Node that has the maximum displacement in the structure	No
x- displ.	Displacement of critical node in the x direction.	No
y- displ.	Displacement of critical node in the y direction.	No
z- displ.	Displacement of critical node in the z direction.	No
θx	Rotation of critical node on global x-axis.	No
θy	Rotation of critical node on global y-axis.	No
θz	Rotation of critical node on global z-axis.	No

Note (2): VisualDesign uses the following equation to find the maximum displacement in the structure:

Maximum displacement = $\sqrt{x^2+y^2+z^2}$

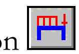
See also

Analysis tab (Project Configuration)

Nodes Displacements Results

Load Combinations Definition

Consulting Load Combination Results

- Activate the "Load Combination" mode using one of the following procedures:
  - Click on the "Load Combination" icon  of Activation toolbar.
  - Select **Edit/Activate Mode/Load Combination**.
- Select a load combination title from the load combination and envelope titles of the pull-down list of the Activation toolbar.
- Select a group of elements of the same type and observe the results using one of the following procedures:
  - Click on the "Properties" icon  of Edit toolbar.
  - Select the **Edit/Properties** functions.
- You can also look at results with one of load combination results spreadsheets from the menu **Results/Load Combinations**. The load combination results spreadsheet for the selected elements appears on your screen. Only the results corresponding to the selected elements are displayed.

See also

Summary

Displacement at Nodes

Reactions at Supports

Internal Forces in Members

Internal Forces in Members (min./max.)

Internal Stresses in Members

Internal Stresses in Members (min./max.)

Nodes Displacements

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Displ. x	x-displacement in the global system	No
Displ. y	y-displacement in the global system	No
Displ. Z	z-displacement in the global system	No
θ x	Rotation of node around global x axis	No
θ y	Rotation of node around global y axis	No
θ z	Rotation of node around global z axis	No

Reactions at Supports

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Value	Indicates the type of results found on this line (Max., Min., …). An empty field indicates a standard analytical result	No
Rx	Reaction in the x direction – local or global system	No
Ry	Reaction in the y direction – local or global system	No
Rz	Reaction in the z direction – local or global system	No
Mx	Moment around x axis – local or global	No
My	Moment around y axis – local or global	No
Mz	Moment around z axis – local or global	No
Orientation	Orientation of the reaction: Local or global axes system	No

Column	Description	Editing
Elastic supports only, with finite elements		
Soil pressure x-dir.	Soil pressure acting on this support in the global x-direction, with respect to the spring stiffness and tributary area. Refer to Automatic Calculation of Tributary Areas	No
(Kp) Soil x-dir	Maximum passive soil pressure on the support, in the global x-direction.	No
Soil pressure y-dir.	Soil pressure acting on this support in the global y-direction, with respect to the spring stiffness and tributary area. Refer to Automatic Calculation of Tributary Areas	No
(Kp) Soil y-dir	Maximum passive soil pressure on the support, in the global y-direction.	No
Soil pressure z-dir.	Soil pressure acting on this support in the global z-direction, with respect to the spring stiffness and tributary area. Refer to Automatic Calculation of Tributary Areas	No
(Kp) Soil z-dir	Maximum passive soil pressure on the support, in the global z-direction.	No

See also

Elastic supports

Calculation of spring supports for piles (foundation design)

Display pressure load on spring supports (pile foundation)

Spring supports' stiffness for lateral backfill on culverts

Internal Forces and Deflections - Members

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
Position	Position on the chord (x=0 at node i)	No
Value	Indicates the type of results found on this line (Max., Min., …). An empty field indicates a standard analytical result	No

Column	Description	Editing
Bending Mx	Bending moment – strong axis	No
Shear. Vy	Shear force – strong axis	No
Bending My	Bending moment – weak axis	No
Shear. Vx	Shear force – weak axis	No
Axial Nz	Axial force (positive in tension)	No
Torsion Tz	Torsion force on the member	No
u (weak axis)	Deflection on weak axis	No
v (strong axis)	Deflection on strong axis	No
w (axial)	Axial deformation (accurate solution only at ends of member)	No

Internal Forces and Deflections (min./max.) in Members

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
Value	Indicates the type of results found on this line (Max., Min., …). An empty field indicates a standard analytical result	No
Bending Mx	Bending moment – strong axis	No
Shear. Vy	Shear force – strong axis	No
Bending My	Bending moment – weak axis	No
Shear. Vx	Shear force – weak axis	No
Axial Nz	Axial force (positive in tension)	No
Torsion Tz	Torsion force on the member	No
u (weak axis)	Deflection relative to weak axis	No

Column	Description	Editing
v (strong axis)	Deflection relative to strong axis	No
w (axial)	Axial deformation (accurate solution only at ends of member)	No

Internal Stresses in Members

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
Position	Position on the chord (x=0 at node i)	No
σ z|Nz	Stress due to axial load	No
σ z|Mx	Stress due to moment – strong axis	No
σ z|My	Stress due to moment – weak axis	No
σ z Max	Maximum stress caused by Nz, Mx and My combined	No
σ z Min	Minimum stress caused by Nz, Mx and My combined	No
τ_yz|Vy	Stress caused by shear - strong axis	No
τ_xz|Vx	Stress caused by shear - weak axis	No
τ|Tz	Shear stress caused by torque	No

Internal Stresses in Members (min./max.)

This spreadsheet is available in the menu **Results / Load Combination** and **Results / Envelope.** It includes internal stresses in steel members OR internal stresses located in the inferior part of a composite section (steel or concrete) that have been analysed with no construction stages

N. B. These results are not valid for a composite beam that have been analysed with construction stages. Please refer to composite beam graphical results, available in menu **Results** / **Load Combination** / **Stresses in Composite Beams**. Graphical results are valid for a composite beam with or without construction stages and include stresses in the section and in the slab.

Group: Load Combination Results: (title)

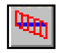
Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
σ z|Nz	Stress due to axial load	No
σ z|Mx	Stress due to moment – strong axis	No
σ z|My	Stress due to moment – weak axis	No
σ z Max	Maximum stress caused by Nz, Mx and My combined	No
σ z Min	Minimum stress caused by Nz, Mx and My combined	No
τ_yz|Vy	Stress caused by shear - strong axis	No
τ_xz|Vx	Stress caused by shear - weak axis	No
τ|Tz	Shear stress caused by torque	No

See also

Interpretation of Composite Beam Results
Composite Beam Graphical Results

Envelope Results

Consulting Envelope Results

- Activate the "Envelope" mode using one of the following procedures:
    - Click on the "Envelope" icon  of Activation toolbar.
    - Select **Edit/Activate Mode/Envelope**.
- Select an envelope title from the load combination and envelope titles of the pull-down list of the Activation toolbar.
- Select a group of elements of the same category and observe the results using one of the following procedure:
    - Click on the "Properties" icon  of Edit toolbar.
    - Select the **Edit/Properties** functions.
- You can also look at results through one of the envelope results spreadsheets from the menu **Results/Envelopes**. The envelope results spreadsheet for the selected elements appears on your screen. Only the results corresponding to the selected elements are displayed.

See also

Properties
Reactions at supports (min/max)
Reactions at supports and Critical Load Combination
Internal Forces in Members and Deflections
Internal Forces in Members and Deflections (min./max)
Members' Internal Forces and Design Groups
Variation of Stresses in Members

Nodes Displacements

Group : Envelope Results: (title)

Column	Description	Editing
Number	Node number	No
Value	Minimum or maximum displacement for this envelope.	No
x- displ.	Displacement of node in the x direction.	No
y- displ.	Displacement of node in the y direction.	No
z- displ.	Displacement of node in the z direction.	No
θx	Rotation of node on global x-axis.	No
θy	Rotation of node on global y-axis.	No
θz	Rotation of node on global z-axis.	No

Reactions at Supports (min./max.)

Group: Envelope Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Value	Indicates the type of results found on this line (Max., Min., …). An empty field indicates a standard analytical result.	No
Rx	Reaction in x direction – local or global system	No
Ry	Reaction in y direction – local or global system	No
Rz	Reaction in z direction – local or global system	No
Mx	Fixing moment around local or global x axis	No
My	Fixing moment around local or global y axis	No
Mz	Fixing moment around local or global x axis	No
Orientation	Orientation of the reaction: local or global axes system.	No

Column	Description	Editing
Elastic supports only, with finite elements		
Soil pressure x-dir.	Soil pressure acting on this support in the global x-direction, with respect to the spring stiffness and tributary area. Refer to Automatic Calculation of Tributary Areas	No
Soil pressure y-dir.	Soil pressure acting on this support in the global y-direction, with respect to the spring stiffness and tributary area.	No
Soil pressure z-dir.	Soil pressure acting on this support in the global z-direction, with respect to the spring stiffness and tributary area	No

Reactions at Supports (min./max.) and Critical Load Combinations

Group: Envelope Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Value	Maximum or minimum forces for this envelope.	No
Load Combination	Critical load combination for the force indicated at column "Value".	
Rx	Reaction in x direction – local or global system	No
Ry	Reaction in y direction – local or global system	No
Rz	Reaction in z direction – local or global system	No
Mx	Fixing moment around local or global x axis	No
My	Fixing moment around local or global y axis	No
Mz	Fixing moment around local or global x axis	No
Soil Pressure x-direction	Soil pressure acting on this spring support towards direction x, considering its tributary surface.	No
Soil Pressure y-direction	Soil pressure acting on this spring support towards direction y, considering its tributary surface.	No

Column	Description	Editing
Soil Pressure z-direction	Soil pressure acting on this spring support towards direction z, considering its tributary surface.	No
Orientation	Orientation of this reaction: local or global axes system.	No

Internal Stresses and Deflections - Members

Group: Envelope Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
Position	Position on the chord (x=0 at node i)	No
Value	Indicates the type of results found on this line (Max., Min.). An empty field indicates a standard analytical result	No
Bending Mx	Bending stress – strong axis plane	No
Shear Vy	Shear stress – strong axis	No
Bending My	Bending stress – weak axis plane	No
Shear Vx	Shear stress – weak axis	No
Axial Nz	Axial stress (positive in tension)	No
Torsion Tz	Torsion stress on member	No
u (weak axis)	Displacement in direction of weak axis.	No
v (strong axis)	Displacement in direction of strong axis.	No
w (axial)	Axial deformation (exact solution only at ends of member)	No

Internal Forces and Deflections (min./max) - Members

Group: Envelope Results: (title)

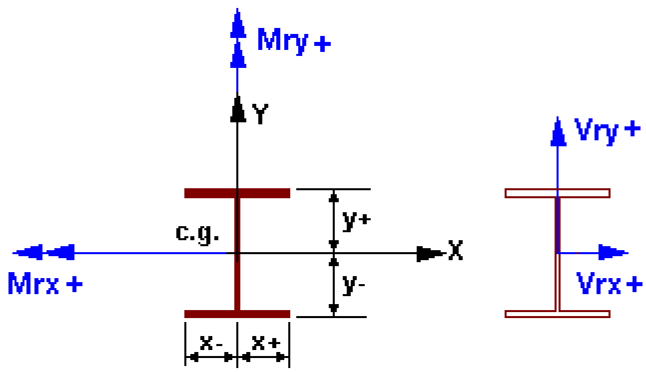
Column	Description	Editing
Number	12 alphanumeric characters	No
Shape	Shape of this member.	No
Value	Indicates the type of results found on this line (Max., Min.). An empty field indicates a standard analytical result.	No
Bending Mx	Bending stress – strong axis plane	No
Shear Vy	Shear stress – strong axis	No
Bending My	Bending stress – weak axis plane	No
Shear Vx	Shear stress – weak axis	No
Axial Nz	Axial stress (positive in tension)	No
Torsion Tz	Calculated automatically	No
u (weak axis)	Displacement in direction of weak axis.	No
v (strong axis)	Displacement in direction of strong axis.	No
w (axial)	Axial deformation (accurate solution only at member ends).	No

Stresses Variations in Members

Interpretation of Results

This spreadsheet supplies the difference between maximum and minimum values for shear forces (Vy) or stresses (Sigma) obtained for the selected envelope.

Stress Variations in a Member



Top	=	**Delta σ |Mx (y+)**
Bottom	=	**Delta σ |Mx (y-)**
Left	=	**Delta σ |My (x-)**
Right	=	**Delta σ |My (x+)**

Group: Envelope Results: (title)

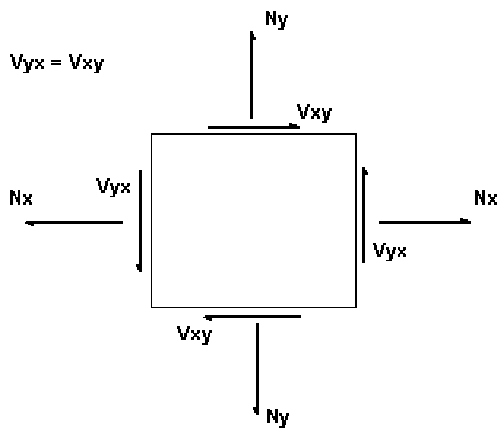
Column	Description	Editing
Number	Member number.	No
Shape	Shape of member.	No
Position	Position on the chord (x=0 at node i)	No
Delta σ|Nz	Stress variation due to axial force.	No
Delta σ|Mx (y+)	Stress variation at the top of section, due to bending moment on strong axis.	No
Delta σ|Mx (y-)	Stress variation at the bottom of section, due to bending moment on strong axis.	No
Delta σ|My (x+)	Stress variation at the right end of section, due to bending moment on weak axis.	No
Delta σ|My (x-)	Stress variation at the left end of section, due to bending moment on weak axis.	No

See also

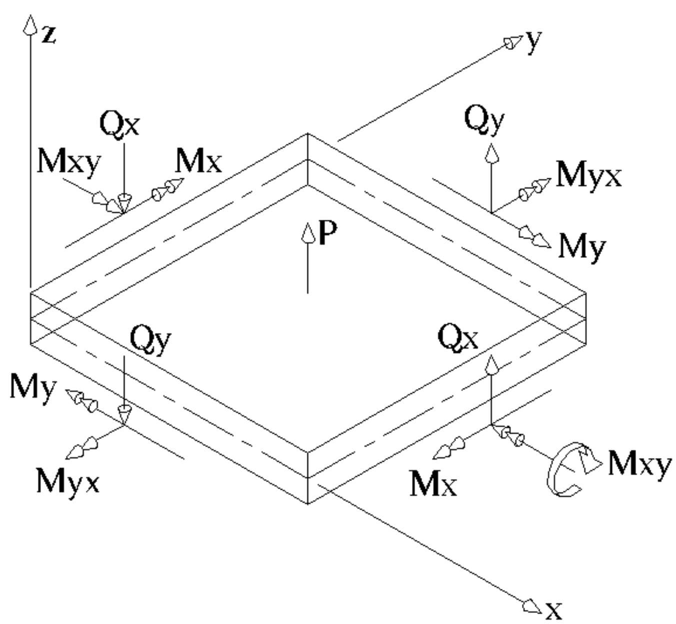
Spacing of Studs – Composite Beam Results
Extra Calculations - Composite Beam

Finite Elements

Convention for Plane Stresses



Sign convention

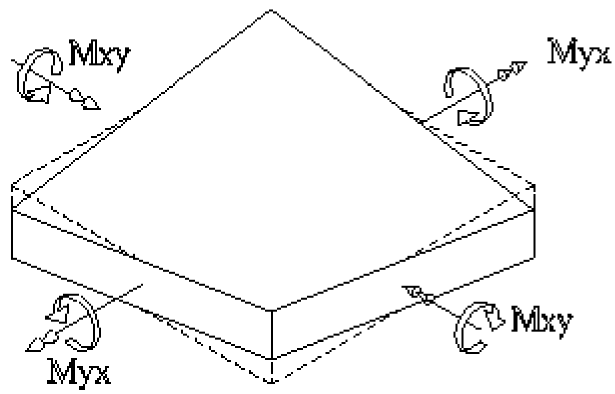


Calculation of Qx and Qy

$$Qx = \frac{\partial Mx}{\partial x} + \frac{\partial Mxy}{\partial y}$$

$$Qy = -\frac{\partial My}{\partial y} - \frac{\partial Mxy}{\partial x}$$

Myx and Mxy Bending Moments

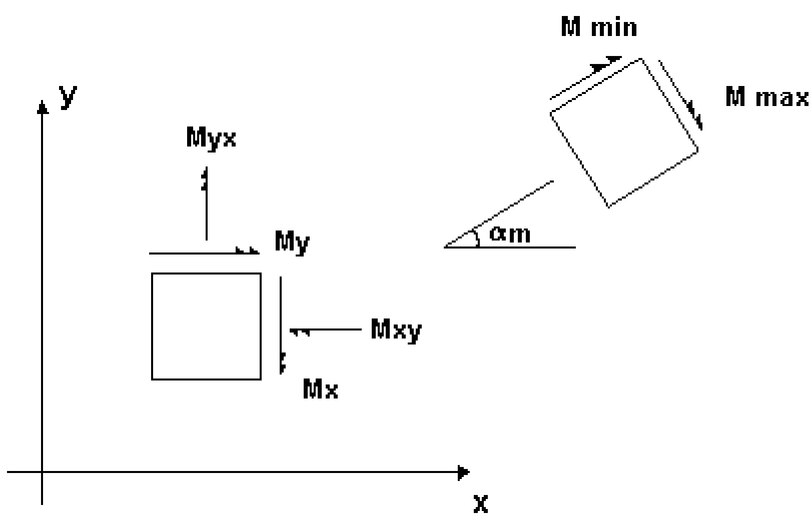


The dotted line shows the original position of the plate.

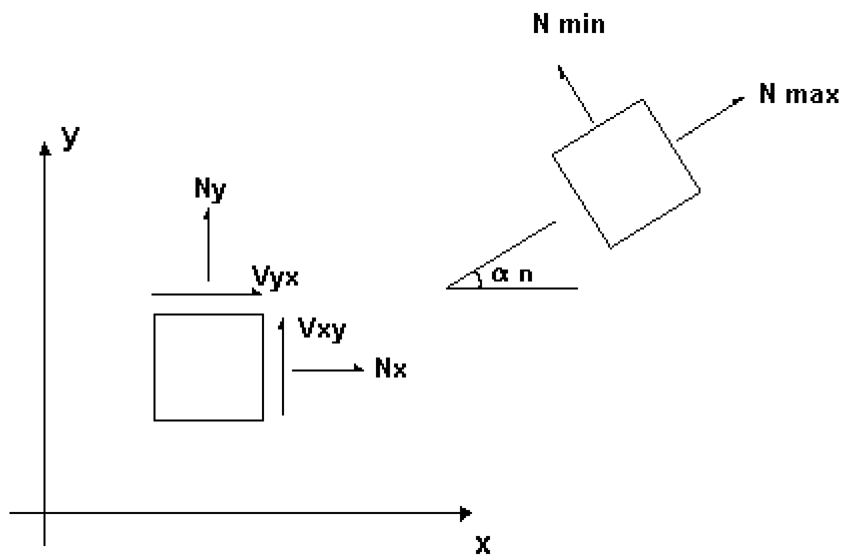
See also
Numbering Convention
Principal Stresses Convention
Interpreting Results

Convention for Principal Stresses

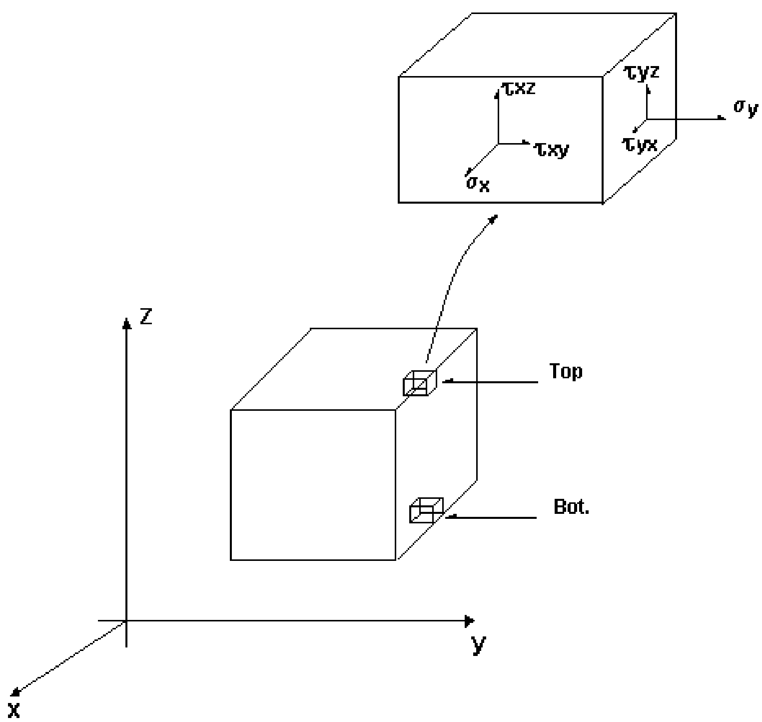
Convention for Bending Moments



Convention for Axial Stress



Convention for Principal Strains



See also

Plane Stress Convention

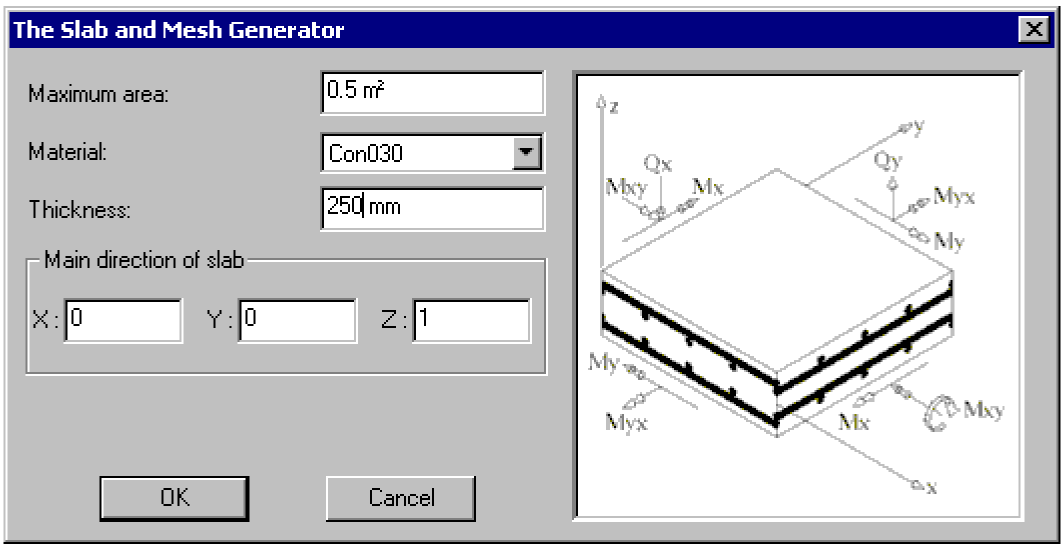
Interpreting Results

The Slab & FE Generator

This powerful tool ((**Structure / Generator / Slab & FE**) creates a slab surface along with finite elements composed of triangular plates. Any slab geometry can be created and meshed with this tool. The mathematical concept used in VisualDesign is called the *Convex Hull*.

The loaded surface can be statically analysed to obtain stress/force contours. It can be designed if users own the Reinforced Concrete Design module.

The slab surface is determined by selected nodes on its outline (members are optional). The user specifies the maximum areas of plates, the material and thickness along with vectors to orient finite elements.

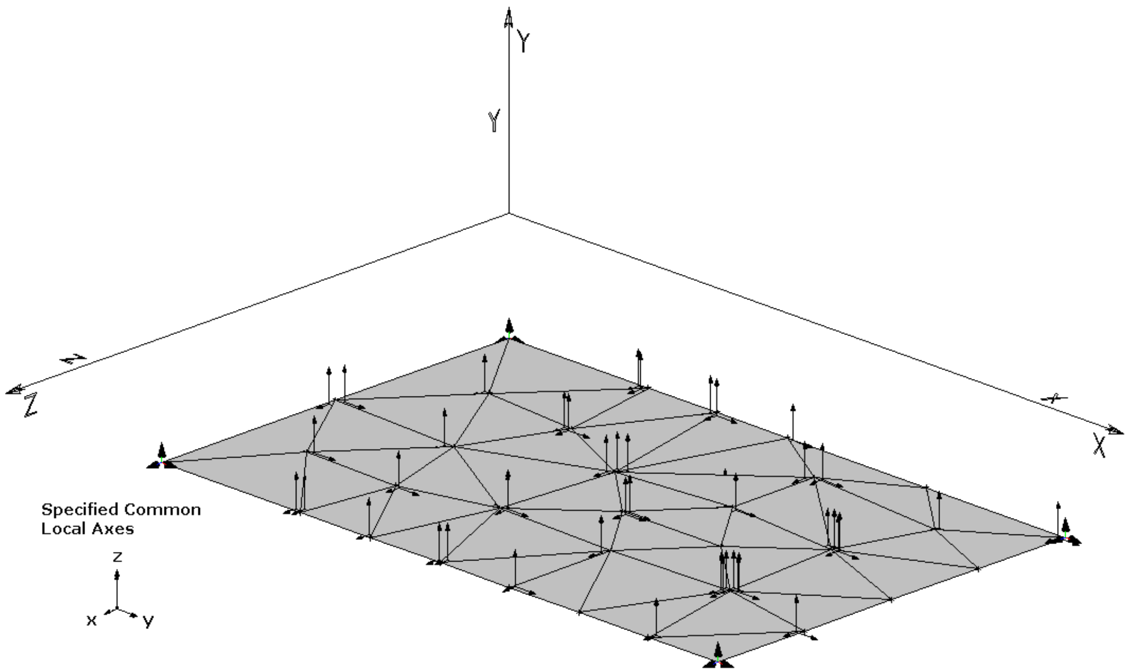


Orientation vectors are automatically initialized in the **Groups of plates – Surfaces** spreadsheet, which is called up when the slab is created. The direction of vectors depends on the order of selected nodes (clockwise or counterclockwise).

Groups of Plates Spreadsheet - Surfaces

1	Number	Origin x m	Origin y m	Origin z m	X-Axis x	X-Axis y	X-Axis z	Y-Axis x	Y-Axis y	Y-Axis z
1	1	1.52	5.00	1.52	0.00	0.00	1.00	1.00	0.00	0.00

Coordinates correspond to the coordinates of the first selected node before calling up the generator.



Openings in the slab

Before calling up the generator, coplanar nodes must be selected. But, members can also be selected to define an opening. The mesher will consider the selected members and no plates should go through these members. However, the generated plates over the opening will have to be erased by hand.

Detecting Columns (Optional)

The tool can search for columns (perpendicular elements) through selected nodes. Nodes at top of columns must then be selected along with nodes on the slab outline. The outline of the biggest column will be added to the imposed boundaries that were created through selected nodes before meshing the convex hull (surface). The biggest column is based on maximum area. It is not necessary to select column along with nodes. Only nodes located at the top of columns and nodes delimitating the floor surface are required.

Saving

We recommend saving the file before calling up the generator. It might be impossible for the mesher to create the slab from selected nodes and members. However, this tool is satisfactory in most cases.

The **Undo** function can be used.

See also

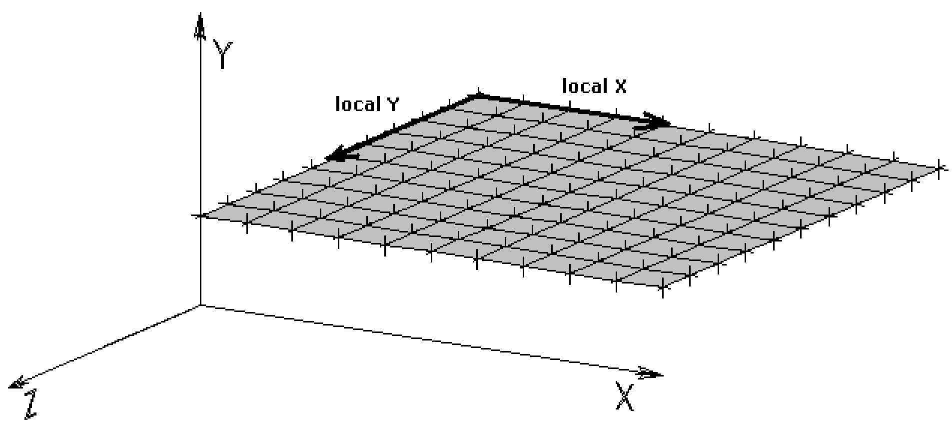
2-way Slab Design
Procedures for 2-way Slab Design
Limitations for 2-way Slab Design

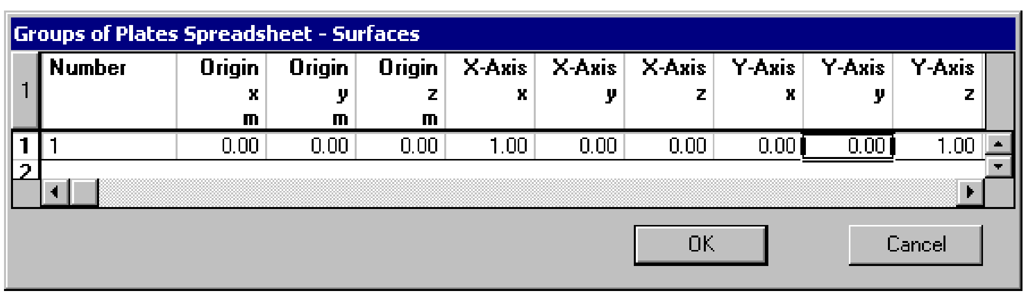
Groups of Plates - Surfaces

Access this spreadsheet by selecting **Plates - Surfaces** under the **Structure/Groups** menu.

This spreadsheet is useful to interpret finite elements results. A local axis system is assigned to a group of horizontal or vertical plates that are composing a slab or a simple wall (not shear walls). Results will be homogeneous and so it will be easier to read results.

Plates are oriented through direction vectors that are normalized by VisualDesign™.




Groups of Plates Spreadsheet - Surfaces

	Number	Origin x m	Origin y m	Origin z m	X-Axis x	X-Axis y	X-Axis z	Y-Axis x	Y-Axis y	Y-Axis z
1	1	0.00	0.00	0.00	1.00	0.00	0.00	0.00	0.00	1.00
2										

OK Cancel

Design of 2-way Slabs

A group of plate must be assigned to plates that are composing a concrete slab. A specification must be selected and concrete covers must be specified.

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Group number or name	Single click
Origin x	Global x-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
Origin y	Global y-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
Origin z	Global z-coordinate of zero point for this group. This data can be useful to localize a group when sorting data.	Single click
X-Axis x	Local x-axis component of the group projected on the global x-axis.	Single click
X-Axis y	Local x-axis component of the group projected on the global y-axis.	Single click
X-Axis z	Local x-axis component of the group projected on the global z-axis.	Single click
Y-Axis x	Local y-axis component of the group projected on the global x-axis.	Single click
Y-Axis y	Local y-axis component of the group projected on the global y-axis.	Single click
Y-Axis z	Local y-axis component of the group projected on the global z-axis.	Single click
Reinforced Concrete Design Module only		
Specification	Design of 2-way slab: Select the concrete specification.	Double click
Top Cover	Specify the concrete cover at the top of the slab.	Single click
Bottom Cover	Specify the concrete cover at the bottom of the slab.	Single click
Lateral Covers	Specify the lateral concrete covers of the slab.	Single click

Groups of Plates - Shear Walls

This spreadsheet is available in **Structure/Groups / Plates – Shear Wall.**

This spreadsheet is required for analyzing or designing a shear wall. Common local axes are assigned to plates that are composing this "section", which can be of any form (T, C, or cubic). The longitudinal axis of the shear wall must be parallel to the gravity axis (up or down), according to the right hand rule. Otherwise, VisualDesign will not be able to build the vertical continuous system, which is represented by a vertical fictitious member.

VisualDesign normalizes vector directions.

Groups are assigned to selected plates through the Plates spreadsheet or **Plate Characteristics** dialog box.

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Group number or name	Single click
Strong axis x	If the strong axis of the shear wall is pointing towards the positive global x direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click
Strong axis y	If the strong axis of the shear wall is pointing towards the positive global y direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click
Strong axis z	If the strong axis of the shear wall is pointing towards the positive global z direction, enter a value of 1.0. If it points towards the negative direction, enter –1.0.	Single click

See also

Interpreting FE Results

Finite Element Analysis Results

Interpreting FE Analysis Results

While visualizing moments and shear diagrams of a structure composed of plates, you will notice that there are sudden jumps where the plates meet. It can be explained by the way that the plates were modeled. The orientation of the local axes system and the plates meshing can influence the results of the analysis.

Important jumps are seen in the shear forces diagram because VisualDesign™ uses a polynomial to approximate plate's deflections. For example, VisualDesign™ will use a 12 terms polynomial to calculate the deflection of a plate composed of four nodes.

Analysis of plates will generally be of good precision for displacements results, a bit less for rotations, fairly well for moments and acceptable for shear forces. These results correspond to the following derivative order:

u	very good	displacements
du → theta	good	rotations
d theta → Moment	fairly good	moments
d Moment → V	acceptable	shear forces

In fact, the more we derive, less good is the precision.

The engineer does the interpretation of results. He can distribute the forces on a given plate width and do some more verification himself to make sure that the plate strength is OK.

Shear forces Vxy

If you want to determine the shear force acting at the base of a shear wall that is composed of rectangular plates, select the horizontal plates located at the bottom and look at the *Vxy* column in the *Internal Forces in Rectangular Plates* spreadsheet.

To get the shear force acting on a given level of the wall, you must add the shear force of plates located at this level. The shear force acting on a plate is equal to Vxy at nodes, which is an average value for the whole plate, multiplied by the dimension of the plate in the local x direction.

So:

Shear force of the wall at level n = sum of [(Vxy for each plate at this level) * dimension of each corresponding plate in the x direction]

Example:

The analysis results for a 0.25m x 0.5m rectangular plate is shown in the table below. The given shear force is the average of the forces at nodes, multiply by the local x-dimension of the plate (0.25m).

Nodes	Vxy (kN/m)
i	147.13
l	147.13
j	147.13
k	147.13

Vxy Plate: 147.13kN/m x 0.25m = 36.78 kN

See also

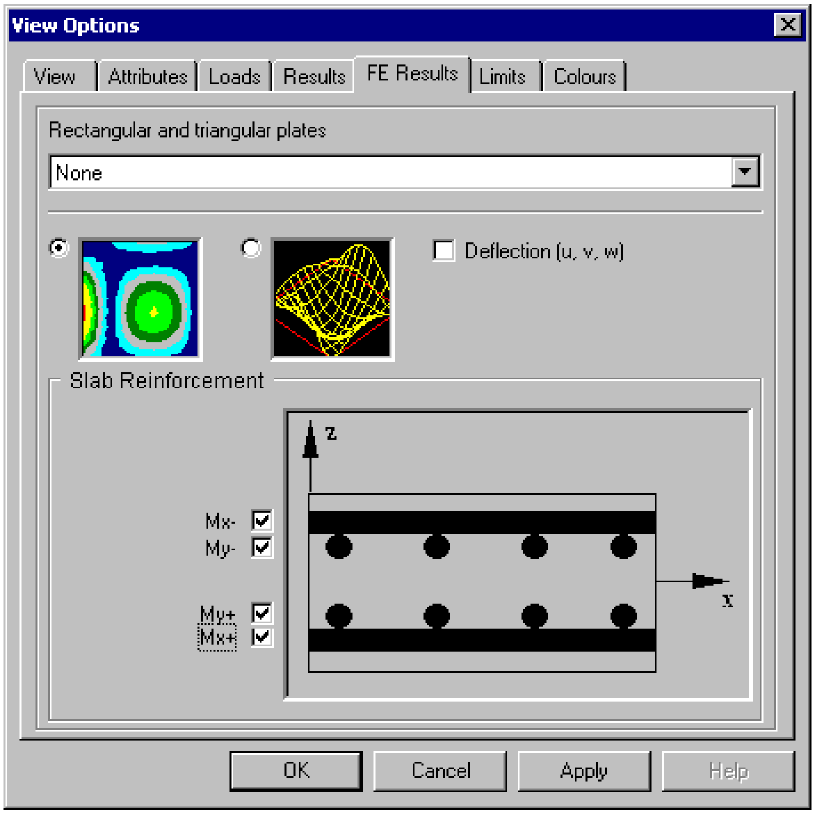
The Plates Spreadsheet
Numbering Convention
Plane Stress Convention
Principal Stresses Convention

The Finite Elements Results Tab

The **Results FE** tab is useful to graphically or numerically display stresses and forces such as, bending moments, shear forces, axial forces, and deflections for rectangular and triangular plates.

Following a static analysis, select a load combination on Activation toolbar and open the **View Options** dialog box. Select the **Results FE** tab.

Select a diagram among the list box. Activate a type of display (stress/force contour or mesh, with or without deflection). Click OK.



Force/Stress Contours

Activate a load combination on Activation toolbar.

The left radio button displays graphic results in the form of stress/force contours if a graph is selected in the list box. Click any icon on Diagrams toolbar to open the **Scaling of Intervals** dialog box. This tool allows modifying the scale (upper and lower limits) for the displayed values (intervals) and colours. Refer to the topic Scaling for Intervals to learn more about this tool.

Mesh and Deflection

Activate a load combination on Activation toolbar.

The right radio button displays the mesh and allows consulting numerical results. Double click on a plate to open the **Internal forces and stresses** spreadsheet or select many plates and press the **Properties** icon to open the spreadsheet.

The check box "Deflection u, v w" displays the deflection of plates for the selected load combination. The deflection can be displayed along with the mesh or coloured stress/force contours.

Slab Reinforcement

The *Reinforced Concrete Design* is required.

This section applies to the display of calculated rebars for 2-way slabs. Four layers of bars can be displayed. The colour of rebars can be modified through respective reinforcement spreadsheets.

See also

Groups of Plates - Surfaces

Scaling for Intervals

View Options

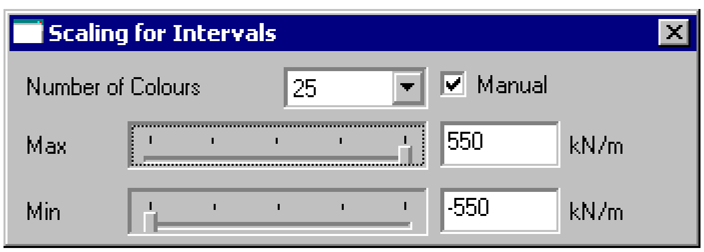
Interpreting Plates Analysis Results

Concrete Design of 2-Way Slabs

Scaling for Intervals

When coloured stress/force contours are displayed for consulting finite elements results, press any icon on Diagrams toolbar to open the **Scaling for Intervals** dialog box.

This tool allows modifying the intervals (values and number of colours) for the displayed legend.



Select a number of colours for the display (9, 25, 45, 105, 225, 525 or 1021).

The dialog box is composed of two scroll bar sliders that allow the scaling of intervals from maximum or minimum values when displaying stress/force contours (legend). Each scroll bar is subdivided into 100 units. Marks indicate a subdivision of 25 units. To move the indicator, click on it and slide the mouse at the right or left on the scroll bar. Displayed intervals will be increased or reduced as a result.

The Manual Mode:

When the *Manual* mode is activated, the absolute minimum and maximum values will still be displayed in the coloured legend but the specified value (min and max) will be used as upper and lower limits for the critical zone. If scroll bar sliders are used to modify the displayed values for intervals, the min and max specified values would be used for the modification instead of absolute values.

Therefore, critical zones (red or blue) are always delimited by the absolute value and specified value if the *Manual* mode is activated. If sliders are used, the modification will be done according to specified values.

The legend displays the lower and upper limits for the two critical zones and intermediate zones (colours) display the middle value of the interval.

Shortcut Keys:

Use keyboard shortcut keys to adjust intervals when your cursor is located in the **Max** or **Min** scroll bar.

Shortcut key	Action
[Home]	The [Home] key moves the indicator at the beginning of the scroll bar slider.
[End]	The [End] key moves the indicator at the end of the scroll bar slider.
→	This arrow moves the indicator one unit right.
←	This arrow moves the indicator one unit left.

Load Combination Results

Activate the Load Combination mode and select a load combination on Activation toolbar. Activate the Plate element on Elements toolbar, select plates and click the **Properties** icon to open this spreadsheet.

Internal Forces in Plates

Group: Load Combination Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Geometry	Geometry of the plate: Rectangular or Triangular.	No
Group	Name of the group where this plate belongs.	No
x	Coordinate relative to the local x-axis.	No
y	Coordinate relative to the local y-axis.	No
Mx	The moment per unit of width, perpendicular to the local x-axis.	No

Column	Description	Editing
My	The moment per unit of width, perpendicular to the local y-axis.	No
Mxy	The torsion per unit of width, around the local x and y-axes.	No
Qx	Shear force per unit of width, in the direction of z-axis, at the y face of local axis system.	No
Qy	Shear force per unit of width, in the direction of the z-axis, at the x face of local axis system.	No
Nx	Normal average force per unit of width, on local x-axis.	No
Ny	Normal average force per unit of width, on local y-axis	No
Vxy	Average shear force per unit of width, on the x face of local axis system.	No
u	Displacement in direction of the local x-axis.	No
v	Displacement in direction of the local y-axis.	No
w	Displacement in direction of the local z-axis.	No
Mmax	Maximum bending moment according to the orientation angle Alpha_M	No
Mmin	Minimum bending moment perpendicular to Mmax	No
αm	Orientation angle of the principal moment Mmax	No
Nmax	Maximum axial force per unit of width, according to orientation angle α n	No
Nmin	Minimum axial force perpendicular to Nmax, per unit of width.	No
αn	Orientation angle of the axial stress Nmax	No
σxTop	Axial strain σ in the x direction, at the top of the plate	No
σyTop	Axial strain σ in the y direction, at the top of the plate	No

Column	Description	Editing
σxyTop	Shear strain τ in the xy direction, at the top of the plate	No
σmaxTop	Maximum axial strain σ in the αTop direction on the top of the plate	No
σminTop	Minimum axial strain σ perpendicular to σmaxTop, at the top of the plate	No
αTop	Orientation angle of the principal axial strain σmaxTop	No
σxBot	Axial strain σ in the x direction, at the bottom of the plate	No
σyBot	Axial strain σ in the y direction at the bottom of the plate	No
τxyBot	Combine shear xy at the bottom of the plate	No
σmaxBot	Maximum axial strain σ in the αTop direction, at the bottom of the plate	No
σminBot	Minimum axial strain σ perpendicular to σmaxTop, at the bottom of the plate	No
αBot	Orientation angle of the principal axial strain σmaxBot	No
~τxz	Average shear strain perpendicular to the plate, in the xz plan	No
~τyz	Average shear strain perpendicular to the plate, in the yz plan	No

See also

Numbering Convention of nodes
Plane Stresses Convention
Principal Stresses Convention
Interpreting Results
Groups of plates - Surfaces
Groups of plates – Shear Walls

Envelope Results

Max/Min Internal Forces in Plates

Activate the Envelope mode and select an envelope on Activation toolbar. Activate the Plate element on Elements toolbar, select plates and click the **Properties** icon to open this spreadsheet.

Group: Envelope Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Geometry	Geometry of the plate: Rectangular or Triangular.	No
Group	Group in which belongs this plate.	No
x	Coordinate relative to local x-axis.	No
y	Coordinate relative to local y-axis.	No
Value	Indicates type of results found on this line (Max., Min., …). Empty field indicates a standard analytical result	No
Mx	Moment per unit of width, perpendicular to local x-axis.	No
My	Moment per unit of width, perpendicular to local y-axis.	No
Mxy	Torsion per unit of width, around local x and y-axes.	No
Qx	Shear force per unit of width, in the direction of z-axis, at the y face of local axis system.	No
Qy	Shear force per unit of width, in the direction of z-axis, at the x face of local axis system..	No
Nx	Normal average force per unit of width on local x-axis.	No
Ny	Normal average force per unit of width on local y-axis.	No
Vxy	Average shear force per unit of width, at the x face of local axis system.	No
u	Displacement in direction of local x-axis.	No
v	Displacement in direction of local y-axis.	No

Column	Description	Editing
w	Displacement in direction of local z-axis.	No

See also

Numbering Convention of nodes
Plane Stresses Convention
Principal Stresses Convention
Interpreting Results
Groups of Plates

Slab-on-Grade

Modeling and Analysing a Slab-On-Grade

Modeling

- Activate the Structure mode.

Create a FE Slab using the Generator

- Activate the Node icon.
- Select coplanar nodes that surround the slab and call up the **Slab & FE Generator** (**Structure / Generator / Slab & FE**).
  - In the dialog box, specify the maximum area of plates, the material, thickness and direction vectors. The slab main direction corresponds to the direction of main layers of bars (layers 1 and 4). Click OK.
  - In the **Groups of plates – Surfaces** spreadsheet, verify the direction vectors. Close the spreadsheet.

Model Elastic Supports

- Activate the Node icon.
- Select all nodes (draw a window around all of them with the cursor). Activate the **Properties** icon (**Edit / Properties**).
- Select a "Support" type of node in the **Node** dialog box. Click Ok.
- Activate the Support icon.
- Select all supports and click the **Properties** icon.
  - In the **Support** tab, select the *Spring* option for Ry and enter the linear rigidity of the soil, K, in the blank field next to it. Free the rotations for Mx, My, and Mz degrees of freedom. Refer to Spring Supports spreadsheet.

Elastic Supports with Axial Release:

- Stay in the **Support** tab and look at the right part that is titled "Release". Reaction Ry must be released for uplift. To release Ry, uncheck the "[-]" option. Refer to Support Release. Click OK.

Calculation of tributary areas for elastic supports:

- Select all supports and go to **Structure / Tools /** Calculation of Tributary Areas.

Loads

- Go to Loads / Load Cases / Definition.
  - Insert a line (click on the number 2 line and press the [Insert] key) in the **Loads Definition** spreadsheet. Give a name to the load case and select a type of load case in the "Type" column.

Apply Loads

- Activate the Load Case mode and select a load case title on Activation toolbar.
- Activate the Plate icon.
- Select all the plates and click on the **Properties** icon.
  - In the **Loads on Plates** dialog box, enter the pressure acting on the plates, according to the global or local axis system of the plates. Click OK.

Load Combination Generator

- Go to Loads / Load Combinations / Automatic Generation.
- Select the appropriate design code in the upper part of the dialog box and activate the generation options. Click on the *Next* button. Look at load combinations that will be generated. If everything OK, click the Finish button. The **Load Combinations** spreadsheet will open. Click OK to exit.

Static Analysis with Axial Release

- Go to **File** menu and open the **Project Configuration** dialog box. Select the **Analysis** tab and activate a linear static analysis with axial release.
- Launch the static analysis.

Load Combination Results

- Activate the Load Combination mode.
- Select a load combination title on Activation toolbar.

Results for Supports

- Go to **Results / Load Combination** menu and open the **Reaction at Supports** spreadsheet. You will find forces on supports and soil pressures, among others. Double-click on a support to open the same spreadsheet.
- Go to **Results / Load Combination** menu and open the **Node Displacements** spreadsheet. You will find the displacement of each support according to global axis system. Double-click on a node to open the same spreadsheet.

Finite Elements Results

Graphic Results

- Open the **View Options** dialog box and select the **FE Results** tab.

- To look at deflections, activate the radio button at the right and check the "Deflection" box (u, v, w).

- To look at stress/force contours, activate the radio button at the left and select a type of graph in the list box. A coloured legend will be posted on screen along with stress/force contours. To edit colours and displayed values that go along with them, click on any icon on Diagrams toolbar (or go to **View / Diagrams** / and select any function). The Scaling of Intervals dialog box will open. Use this tool to modify the number of displayed colours and modify values of intervals (a minimum and maximum value can be specified if the "Manual" mode is activated). Use the "Max" and "Min" scroll bar sliders to modify minimum and maximum displayed values for the selected graph.

Numerical Results

- Go to **Results / Load Combination / Internal forces in Plates**.

Or,

- Click the Plate icon.

- Double-click on any plate to open the results spreadsheet for this plate only or select many plates and click the Properties icon to open the spreadsheet including results for these plates.

- Refer to topic Interpreting Plates Results, to help you interpreting data in this spreadsheet.

Chapter 6

FOUNDATION DESIGN

TABLE OF CONTENTS

Chapter 6 Foundation Design

General

The Foundation Design module

The **Foundation Design** module includes the S6-88, S6-00, S16-01, and AISC-LRFD standards for the computation of structural and geotechnical forces for shallow and deep foundations. These calculations are based on the recommendations given in the *Canadian Foundation Engineering Manual* and on other models used in the practice.

All calculations are done using the limit states design allowing a better integration of load cases already used by structural engineers. In all cases, the ground water elevation can be specified.

The static analysis is used for the calculation. However, when a design module is used, the design of foundations takes part in the cyclic process.

A stratigraphical profile and foundation specification must be created before defining a foundation model, except in the case where the Foundation Modeling Wizard is used. The Modeling Wizard is provided to help users to quickly define foundation models. In this case, the software fixes parameters. However, they can be modified in all time through spreadsheets.

When shallow foundation design is carried out, VisualDesign calculates the required dimensions for the footing but it does not modify the footing thickness, which is specified by the user. If the bearing capacity is sufficient, VisualDesign will place reinforcing bars in the footing. The user can change the rebars dimension and results will be automatically recalculated within this results spreadsheet.

Piles foundations can only be verified, for the moment. The structural capacity of the pile is verified and the geotechnical capacity also. Graphic results are also supplied for verification of piles.

Soil-structure interaction can be considered during analysis. Footing settlements induce forces in the structure, which will be redistributed by VisualDesign until convergence is reached.

See also

Axis System Convention
Configuration of the Project Foundations Parameters
Rocks spreadsheet
Cohesive Soils spreadsheet
Granular Soils spreadsheet
Important parameters to define soils
Foundation Modeling Wizard
Stratigraphical Profiles spreadsheet

Design Limitations

Sloped footings

VisualDesign does not consider sloped footings.

Reactions

The calculated reactions are applied to the centre of footings.

Several nodes on a footing

It is possible to place several nodes on a footing by defining very rigid elements (10 to100 times more rigid than the most rigid element in the structure) at the support node located at the centre of the footing.

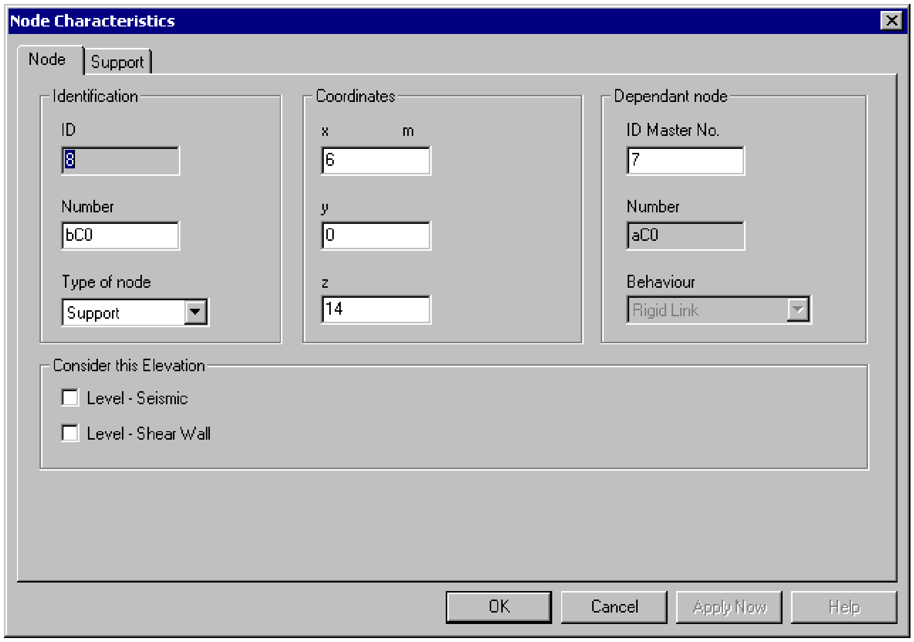
If a shallow foundation is supporting two columns, VisualDesign™ will not calculate the extra reinforcement needed around columns. The engineer must use his own discretion in these cases.

Rigid footing

Shallow foundations are calculated in keeping with the basic hypothesis that footings are rigid and not flexible elements.

Rigid Links

To automatically create rigid links, select a node that you wish to be dependent. Call the **Node Characteristics** dialog box by double-clicking on it and specify the master node's ID in the appropriate field. You will notice that "Rigid link" is posted in the "Behaviour" box and that it is shaded, meaning that you cannot edit this characteristic. Example: A support node is the master node of four dependent nodes corresponding to the beams starting nodes at the foundation level.



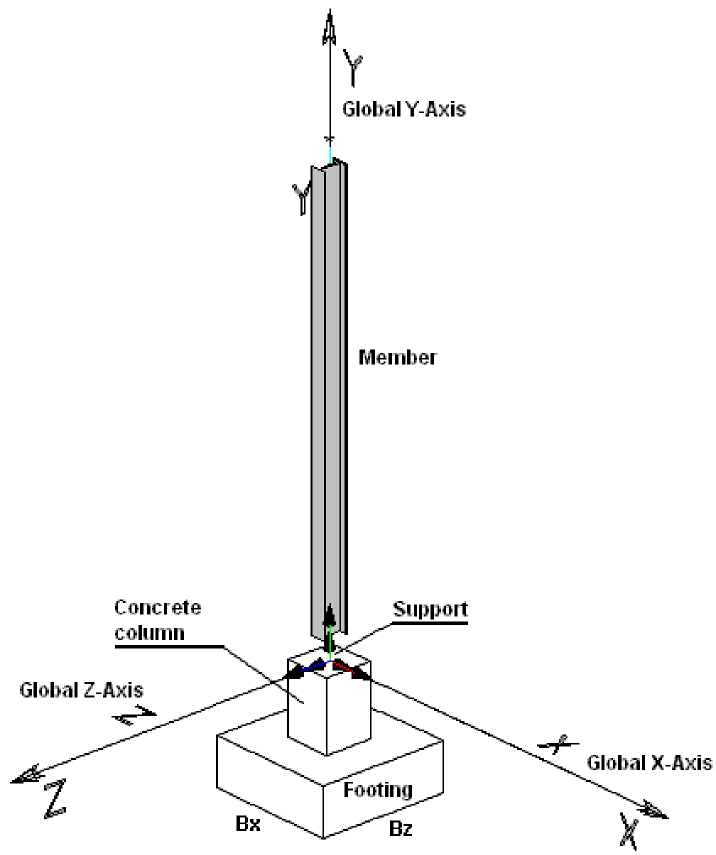
Configuration for Foundations Parameters

The first step to do when you use the foundation design module is to configure the project foundations in the **Foundation** tab of **Project Configuration** dialog box (**File** menu).

Axis System Convention

The foundation column rests just below the support node and is centred on both the column and footing.

Gravity can be represented by the x, y or z-axis, depending on the user's preference (**Preferences** tab- **Project Configuration**).



Shallow Foundations

The **Foundation Design** module allows optimizing or checking shallow foundations.

In the design mode, VisualDesign™ optimizes the footing dimension for a more economical design that satisfies safety criteria and all load combinations. The thickness of the footing is not modified. If the shear stress is too high, or the calculated bearing capacity is insufficient, a new model has to be defined.

In the verification mode, VisualDesign™ checks the footing bearing capacity and the settlements for all load combinations.

The following calculation models for bearing capacity are included in VisualDesign™: Meryerhof, Hansen, Vesic, Terzaghi, CNBC/CFEM/S6-00, SPT, vane shear test and static penetrometer test.

This module considers the soil/structure interaction. In this case, the internal forces created by footing settlements are redistributed into the structure. The software performs iterations until it reaches convergence between the settlements and forces to attain force/settlement compatibility. In Design mode, footings are optimized according to the soil capacity (ultimate limit states) or settlement (serviceability limit states). Eccentricities caused by bending moments or eccentric loads are considered for the design or analysis. Eccentricities may be limited as specified.

After analysis, the following results are available: footing dimension, effective footing dimension, bearing capacity, soil pressure beneath the foundation, settlements, service reactions and ultimate reactions and percentage of used capacity for each footing.

A spreadsheet allows the structural design of the footing. The engineer specifies the code, the reinforcing material, the type of concrete and the minimum concrete cover. The spreadsheet will automatically calculate the required number of rebar in the two directions, the spacing of rebar, the footing bending strength, shear strength and punching shear strength.

See also

Definition of Foundation Model

Shallow Foundation Spreadsheet

Piles Foundations

The **Foundation Design** module also analyzes a foundation consisting of a sole pile or a group of piles. Geotechnical and structural strength are verified in the design of the pile foundation. The pile location, steel shape, inclination and beta angle are defined in the **Foundation Model Spreadsheet**. Any number of irregular layouts can be defined. However, the engineer may use the geometric layout generator available in VisualDesign™. VisualDesign™ has predefined steel shapes that may be used for pile sections. The pile section may also be filled up with concrete, if desired.

The pile lateral strength is calculated using a finite element model that uses the soil *Kh* or *nh* value (soils secant module) according to the defined stratigraphical profile. The computation of the soils secant moduli takes into account the different soil layers that define the stratigraphical profile.

Following analysis, the results can be seen in the form of spreadsheets or graphically on the screen. One of the available results spreadsheets contains geotechnical results for each foundation support node. It includes the following information: geotechnical axial strength of a pile or group of piles, geotechnical lateral strength, settlements, ultimate and service reactions forces, and the design load of the foundation. A second spreadsheet includes the forces acting in each of the piles (axial forces, shear forces, bending moments and deflections) and the calculation of their structural strength (axial strength, bending strength, shear strength, etc).

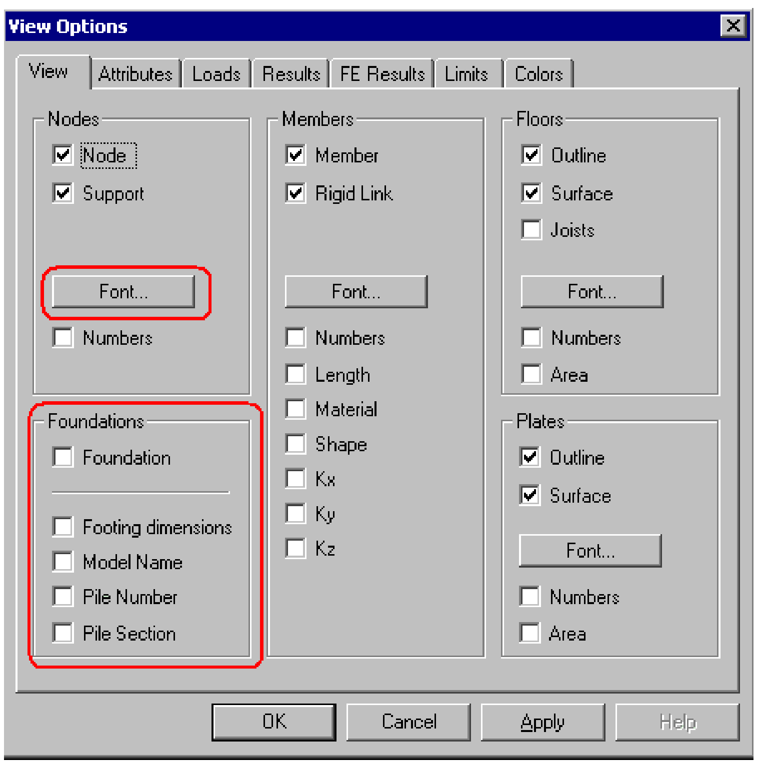
See also

Pile Foundation Spreadsheet

Definition of Foundation Model

View Options – Foundations

To display the foundation attributes on the screen, click on the **View Options** icon and check the "Foundation" box in the **View** tab. You can also display footing dimensions, model name, pile number and/or pile section. Press the "Font" button that is posted in the "Nodes" section to modify the font as you wish.



Before the analysis is performed, the maximum or actual dimensions of the footing entered in the foundations spreadsheet will be displayed in keeping with the chosen mode of analysis: In **Design** mode, foundations will display the maximum dimensions specified in the **Foundation Models** spreadsheet, whereas in **Verification** mode, foundations will display their actual dimensions.

To visualize the name of the foundation model associated with a support node, tick the box "Model name". As for pile foundations, display pile numbers or pile sections by ticking off the appropriate boxes.

See also

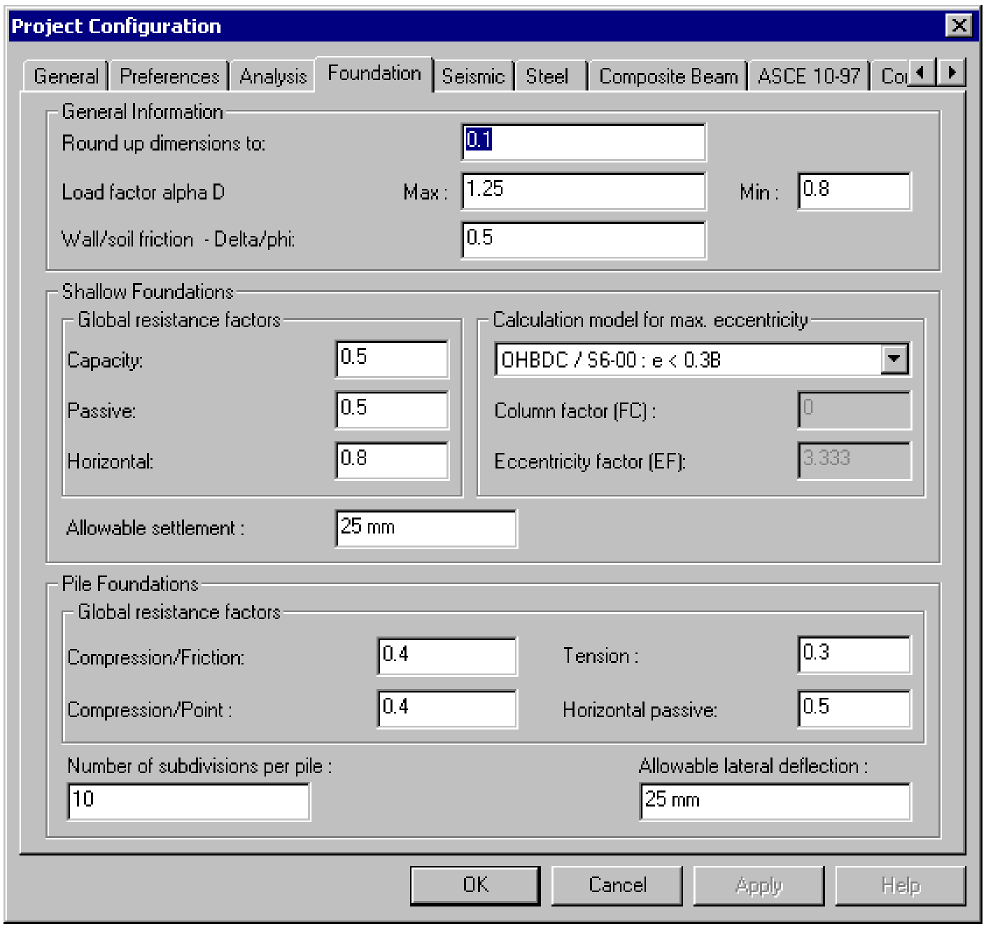
View Options

The Colours tab

Project Configuration

Foundation tab

Parameters and global resistance factors used for the design of shallow foundations as well as those applied to the verification of pile foundations are listed in the **Foundation** tab of **Project Configuration**.



Field	Definition
General Information	
Round-off Dimensions to:	Allow you to round off dimensions of the foundation according to a specified value.
Load Factor Alpha D	Max: If the load factor for dead load is greater than 1.0 for ULS, VisualDesign™ will used this load factor. Min: If the load factor for dead load is smaller than 1.0 for ULS, VisualDesign™ will used this load factor.

Field	Definition
Friction Wall/Soil - Delta/Phi	Specify the reduction (in decimals) to apply to the angle of friction, delta (δ), of a granular soil against a concrete wall. According to *Bowles*, this angle can be considered as the angle of internal friction of the soil, phi (Φ), for concrete materials. The delta angle is used to calculate the active (Pa) and passive (Pp) earth pressure. Some standards reduce this angle by a factor of 2/3 or 50%.
Shallow Foundations	
Global Resistance Factor	USD global resistance factors replace the safety factor used in the allowable stress method. For a shallow foundation, a global resistance factor of 0.50 equals to a safety factor of 3 when the ASD method is used. Default values are: ϕ capacity = 0.5 ϕ passive = 0.5 ϕ horizontal = 0.8
Calculation Method for Max Eccentricity	Choose a model for calculating the maximum allowable eccentricity. Three models are proposed: 1. Model OHDBC: e < 0,3B 2. Model ACI 318: B > 4e + w 3. Custom model: B > FE(e) + FC(w) where "w" is the column width.
Column Factor (FC) and	If you chose a custom model, enter FC value needed to compute B.
Eccentricity Factor (EF)	If you chose a custom model, enter FE value needed to compute B.
Allowable Settlement	Allowable settlement of spread footings for the analysis of limit conditions during service. (Generally equal to 25mm)
Pile Foundations	
Global Resistance Factor	USD global resistance factors replace the safety factor used in the allowable stress method. For a pile foundation, a global resistance factor of 0.40 equals a safety factor of 3 when the ASD method is used. Default values are: ϕcs compression/friction = 0.4 ϕcb compression/point = 0.4 ϕts tension = 0.3 ϕ passive horizontal = 0.5

Heading	Definition
Number of subdivisions per pile	Specify pile number of subdivisions for the display of internal force and deflection diagrams.
Allowable Lateral Deflection	Specify allowable pile lateral deflection. Generally equal to 25 mm

See also

Project Configuration
The Foundation Design Module
The Foundation Factor F

Soils

Typical Values for Soils' Properties

Note Typical values given in the tables below may vary. They cannot be used for a detailed design.

SOFT AND SENSITIVE CLAYS§

Properties	Typical value
Effective cohesion, c’	5 - 10 kPa
Effective angle of internal friction, ϕ’	25 - 31°
Poisson's ratio (saturated), μ	0.4 – 0.5 (Bowles, 1988)
Young modulus, E	2 - 25 MPa (Bowles, 1988).
Undrained shear resistance, cu	20 - 70 kPa
Humid unit weight, γhum	16 -18 kN/m3
Saturated unit weight, γsat	16 -18 kN/m3

♣Soft and sensitive clays of Champlain Sea included.

FINE AND LOOSE SAND

Properties	Typical value
Effective cohesion, c’	0 kPa
Effective angle of internal friction, ϕ’	30- 34°
Poisson's ratio (saturated), μ	0.3 – 0.4 (Bowles, 1988)
Young modulus, E	10 -30 MPa (Bowles, 1988)
Humid unit weight, γhum	16 -18 kN/m^3
Saturated unit weight, γsat	16 -18 kN/m^3

STIFF CLAYS

Properties	Typical value
Effective angle of internal friction, ϕ'	30 - 40°
Poisson's ratio (saturated), μ	0.4 – 0.5 (Bowles, 1988)
Young modulus, E	50 -100 MPa (Bowles, 1988)
Undrained shear resistance, cu	75- 200 kPa
Humid unit weight, γhum	17 - 19 kN/m^3
Saturated unit weight, γsat	17,5 -20 kN/m^3

DENSE SAND

Properties	Typical value
Effective cohesion, c'	0 kPa
Effective angle of internal friction, ϕ'	36- 45° (Meyerhof,1974; Peck, 1974)
Poisson's ratio (saturated), μ	0.3 – 0.4 (Bowles, 1988)
Young modulus, E	50 - 80 MPa (Bowles, 1988)
Humid unit weight, γhum	18 –22 kN/m^3
Saturated unit weight, γsat	19 –23 kN/m^3

SEDIMENTARY ROCK

Properties	Typical value
Effective cohesion, c'	0,5 – 5,0 MPa
Effective angle of internal friction, ϕ'	33° - 37°
Poisson's ratio (saturated), μ	0.3 – 0.35
Young modulus, E	40 - 70 GPa
Humid unit weight, γhum	27,0 kN/m^3

GRANITIC ROCK

Properties – Granitic rock	Typical value
Effective cohesion, c’	1 – 10 MPa
Effective angle of internal friction, ϕ’	33° - 40°
Poisson's ratio (saturated), μ	0.3 – 0.4
Young modulus, E	50 – 100GPa
Humid unit weight, γhum	27,0 kN/m^3

Important Parameters for Soils

If personalized soils are added in one of VisualDesign Soils spreadsheets, you must, at least, define the following parameters:

Cohesive Soils	**Calculation of Bearing Capacity**	**Calculation of Settlements**
	Name of the soil Undrained shear resistance (not necessary if qult is known) Saturated and Unit Humid Weights	Young's modulus Poisson's ratio
Granular Soils	**Calculation of Bearing Capacity**	**Calculation of Settlements**
	Name of the soil Effective angle of internal friction Saturated and Unit Humid Weights	Young's modulus Poisson's ratio

See also

Cohesive Soils spreadsheet
Granular Soils spreadsheet

Rocks Spreadsheet

Pre-defined rocks are not editable but personalized ones can be added at the end of the spreadsheet. This spreadsheet is located the **Common** menu, at heading **Soils**.

Group: Shared data: Vdbase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Name	Soil name	Single click
Cohesion	Roc cohesion: Enter the undrained cohesion for calculating short-term shifts.	Single click
ϕ	Angle of internal friction of the soil	Single click
q ultimate	Ultimate bearing capacity of soil, if known	Single click
γ Humid	Specific humid weight of soil	Single click
γ Saturated	Specific saturated weight of soil	Single click
E	Elastic modulus of soil (Young)	Single click
Poisson	Poisson's ratio of soil	Single click
RQD	Rock quality index	Single click

See also

Typical Properties for Soils

Cohesive Soils Spreadsheet

Pre-defined cohesive soils are not editable but personalized ones can be added at the end of the spreadsheet. This spreadsheet is located the **Common** menu, at heading **Soils**.

Important The humid and saturated weights of a soil are used to calculate the weight of soil located above and below water table. Usually, the saturated weight is slightly higher that the humid weight. ***Do not subtract the weight of water from the saturated weight of soil because VisualDesign already does that automatically.***

Group: Shared data: Vdbase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Name	Soil name	Single click
Undrained Shear Resistance, Cu	Undrained shear resistance for this soil. This variable is represented by "cu" or "su", according to codes.	Single click
ϕ	Angle of internal friction of the soil	Single click
q ultimate	Ultimate bearing capacity of soil, if known.	Single click
γ Humid	Specific humid weight of soil.	Single click
γ Saturated	Specific saturated weight of soil	Single click
E	Elastic modulus of soil (Young)	Single click
μ	Poisson's ratio of soil	Single click
qc	End bearing of static penetrometer	Single click
Ip	Plasticity number	Single click
N1-60	Corrected N index corresponding to 60% of energy of the SPT test.	Single click

Granular Soils Spreadsheet

Pre-defined granular soils are not editable but personalized ones can be added at the end of the spreadsheet. This spreadsheet is located the **Common** menu, at heading **Soils**.

Important The humid and saturated weights of soils are used to calculate the weight of soil located above and below water table. Usually, the saturated weight is slightly higher that the humid weight. ***Do not subtract the weight of water from the saturated weight of soil because VisualDesign already does it automatically.***

Group: Shared data: VDbase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Name	Name of soil	Single click
Cohesion	Undrained cohesion of this soil. Letter "c" in most of the codes represents this variable.	Single click
ϕ	Angle of internal friction of the soil	Single click
q ultimate	Ultimate bearing capacity of soil, if known	Single click
γ Humid	Specific humid weight of soil	Single click
γ Saturated	Specific saturated weight of soil	Single click
E	Elastic modulus of soil (Young)	Single click
μ	Poisson's ratio of soil	Single click
qc	End bearing of static penetrometer	Single click
N1-60	Corrected N index corresponding to 60% of energy of the SPT test	Single click
β Driven piles	Combined resistance factor along the driven pile	Single click
Nt Driven piles	Bearing capacity factor for driven piles.	Single click
β Bored piles	Combined resistance factor along the bored pile	Single click
Nt Bored piles	Bearing capacity factor for bored piles.	Single click
nh	Factor relative to the unit weight of the soil (kN/m3)	Single click

See also

Typical Properties for Soils

Foundation Modeling Wizard

Foundation Modeling Wizard

The **Foundation Modeling Wizard** has been created to help engineers to quickly generate a shallow or deep foundation model. VisualDesign creates a stratigraphical profile, according to the design soil that is chosen by the user. In addition, the program automatically generates specifications for shallow or deep foundations. This tool is available in the **Structure** menu, at heading **Foundation Models**.

Once that the required parameters are entered in the dialog boxes that are composing this tool, you can open the **Deep** or **Shallow Foundations** spreadsheet and the **Stratigraphical Profiles** spreadsheet and modify default parameters set by the program. The data that were entered through the modeling wizard will be included in spreadsheets.

You are allowed to use to modeling wizard the number of times that fits you. The program will number the foundation models starting from 1 with an increment of 1. The user only has to select the appropriate foundation model number to be assigned to supports. Refer to topic Assign a foundation model to a support

If you think that you made a mistake in the modeling wizard, you can create a new foundation model and erase the first one in appropriate spreadsheets. Alternatively, you can open the spreadsheets and modify data.

Data to be specified in dialog boxes

Shallow Foundation:

- Design Soil;
- Ground Elevation (it must be compatible with elevation of supports);
- Height and dimensions of concrete column;
- Footing dimensions and thickness. (Enter current dimensions for verification or enter maximum dimensions for a design).

Pile Foundation:

- Design Soil;
- Ground elevation (it must be compatible with elevation of supports);
- Height and dimensions of concrete column;
- Dimension and thickness of footing. Enter current dimensions for a verification and maximum dimensions for a design).
- Piles steel shape, material, and length;

- Definition of piles layout.

Generated Data

Stratigraphical Profile

VisualDesign creates a stratigraphical profile according to the specified ground elevation and design soil. The depth of this soil layer is 21.45 meters. The ground water table is located at –21.45 m. The name of this profile is the same as the foundation name. To modify the generated stratigraphical profile, go to **Structure** / **Stratigraphical Profiles**.

Shallow Foundation Model

SPECIFICATION FOR SHALLOW FOUNDATION

VisualDesign creates a specification having the same name as the foundation name. The CAN/CSA-A23.3-94 standard is used for design and the rebars material is G30.18-400R. To modify these data, go to **Structure / Specifications / Shallow Foundations**.

DESIGN CRITERIA FOR SHALLOW FOUNDATIONS

VisualDesign chooses the "known q ult" method for the calculation of bearing capacity (the value of q ult is written in the Cohesive soils spreadsheet). Adhesion, Ca, is equal to 250 kPa for a cohesive soil. Footing concrete covers are set to 75 mm. The rebars layout favours Mx and the type of rebars are 25M for both directions.

Deep Foundation Model

SPECIFICATIONS FOR DEEP FOUNDATION

VisualDesign creates a specification for the deep foundation model. The name of this specification is the same as the foundation name. The standard that is applied for the structural verification of piles is CAN/CSA-A23.3-95. To modify these data, go to **Structure** / **Specifications / Deep Foundations.**

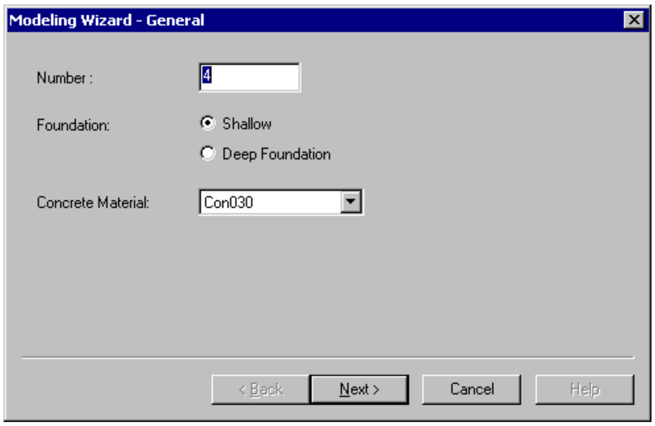
DESIGN CRITERIA FOR DEEP FOUNDATIONS

Piles are driven in place and they work as point bearing. Piles end conditions are fixed at both ends and the buckling length is 0%. These default parameters are the same for a cohesive or granular soil.

Foundation Modeling Wizard Dialog Boxes

Modeling Wizard – General

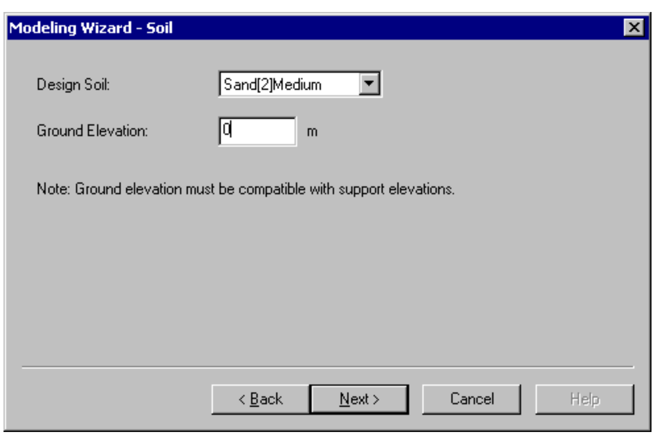
First Wizard dialog box for both deep and shallow foundation models:



Field	Description
Number	Keep the default number or enter a name for this foundation model.
Foundation	Activate the appropriate radio button that corresponds to the foundation model you want to generate: Shallow foundation or Deep Foundation.
Concrete	Keep the default concrete material or click the arrow and select another one.

Modeling Wizard – Soil

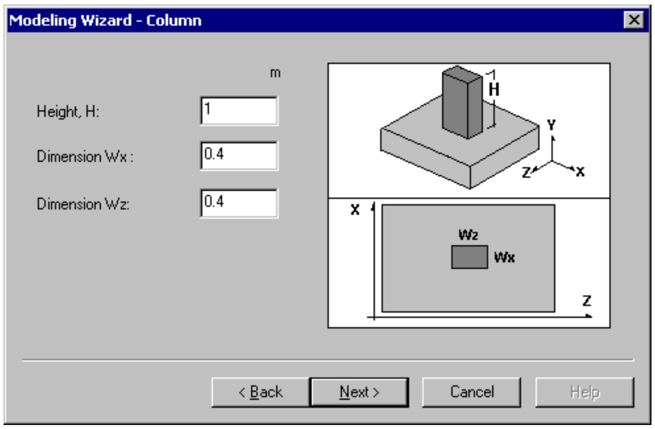
Second Wizard dialog box for both deep and shallow foundation models. VisualDesign will generate a stratigraphical profile according to these data.



Field	Description
Design Soil	Click on the arrow and select the design soil located under the footing.
Ground Elevation	Enter the ground elevation. Make sure that this elevation is compatible with elevation of supports in your model.

Modeling Wizard – Column

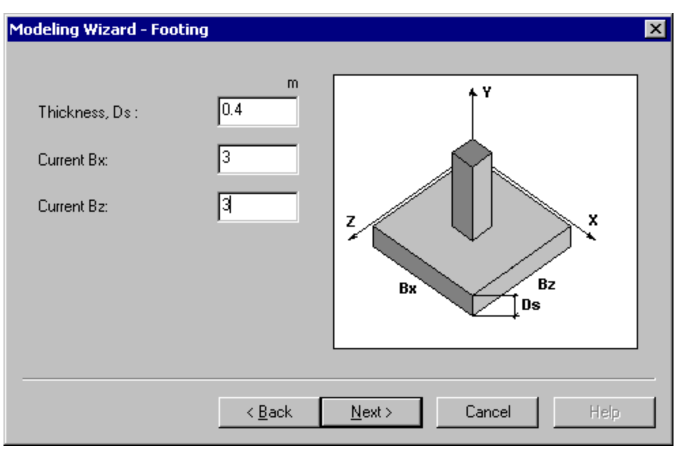
Third Wizard dialog box for both deep and shallow foundations:



Field	Description
Height H	Enter the height of concrete column that s located above the footing.
Dimension Wx	Enter the dimension of column in the global x-axis.
Dimension Wz	Enter the dimension of column in the global z-axis.

Modeling Wizard - Footing (Deep foundation)

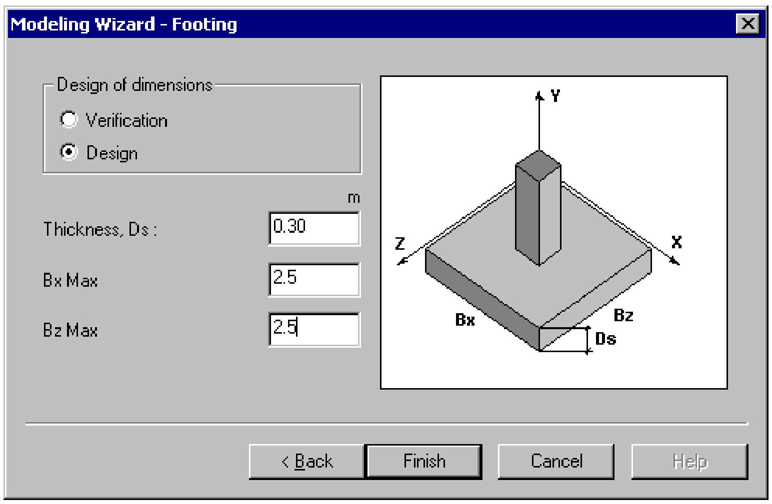
Fourth Wizard dialog box for a deep foundation model:



Field	Description
Thickness Ds	Enter the footing thickness located above piles head.
Current Bx	Enter the current dimension Bx, according to global x-axis.
Current Bz	Enter the current dimension Bz, according to global z-axis.

Modeling Wizard - Footing (Shallow Foundation)

Fourth and last Wizard dialog box for a shallow foundation model:

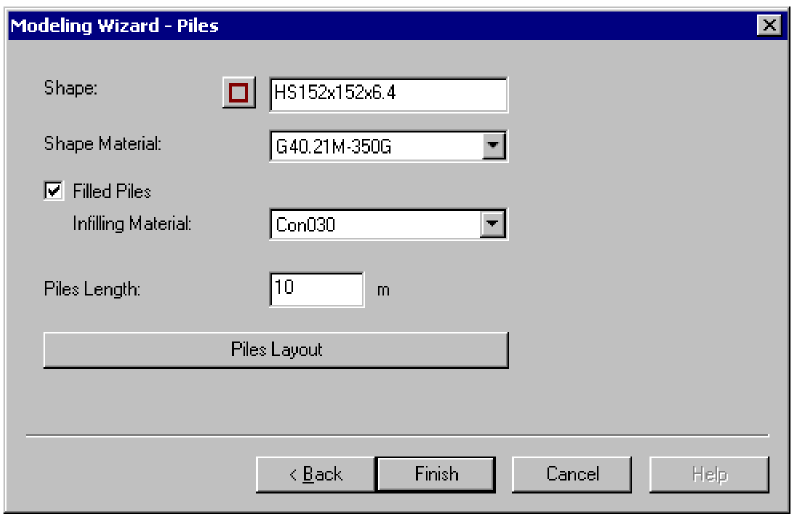


Field	Description
Design of Dimensions	Activate the appropriate radio button for a design or a verification of the footing dimensions.
Verification	This is a geotechnical verification.
Thickness Ds	Enter the footing thickness. After analysis is completed, check the minimum thickness, *H min*, written in the Results spreadsheet for shallow foundation.
Current Bx	Enter the current dimension Bx, according to global x-axis.
Current Bz	Enter the current dimension Bz, according to global z-axis.
Design	This is a structural design. VisualDesign will place reinforcement in the footing.
Thickness Ds	Enter the footing thickness. VisualDesign does not modify this thickness. After analysis is completed, check the minimum thickness, *H min*, written in the Results spreadsheet for shallow foundation.

Field	Description
Bx Max	Enter a maximum dimension for the design, according to global x-axis.
Bz Max	Enter a maximum dimension for the design, according to global z-axis.

Modeling Wizard – Piles

Fifth Wizard dialog box for a deep foundation:



Field	Description
Shape	Click the I-Beam icon and choose a shape among the Shape Selection tree.
Shape Material	Click the arrow and choose a material in the among the Steel Material Selection tree.
Filled Piles	By default, this box is activated. If you chose a shape other than a hollow section, VisualDesign will not consider this field.
Infilling Material	By default, the infilling material is a 30 MPa concrete. To modify this data, click the arrow and select another material. . If you chose a shape other than a hollow section, VisualDesign will not consider this field.
Pile Length	Enter the length of piles.
Button "Pile Layout"	Press this button to open the Pile Layout dialog box. It is the same dialog box as the one included in the Deep Foundation Model dialog box.

Modeling Wizard - Piles Layout

This is the sixth, and last Wizard dialog box for a deep foundation model. This dialog box is identical to the **Piles Layout** tab included in the **Deep Foundation Models** Dialog box.

See

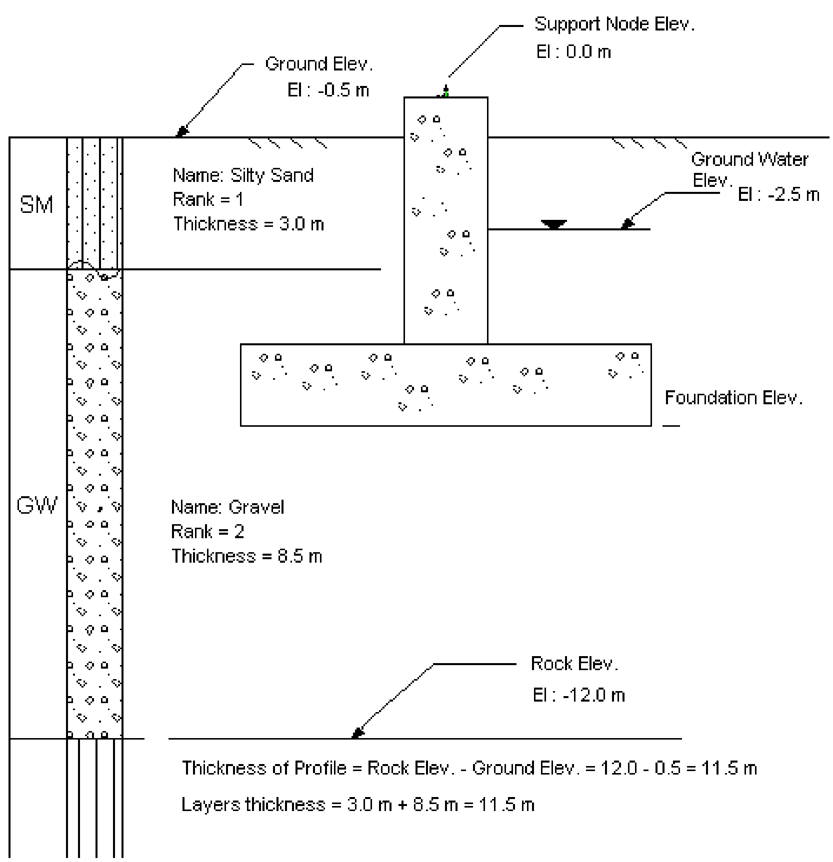
Piles Layout tab

Stratigraphical Profiles

Stratigraphical Profiles Dialog Box

You must define the stratigraphical profile on which the foundations will rest. The soils report may indicate that the foundations may be erected on different strata. Indicate, for each of these strata, elevation of topsoil, groundwater table and rock elevation. To do so, select the **Stratigraphical Profiles** heading under **Structure** menu, enter values in the first tab. Switch to the **Layer Definition** tab, and enter each soil layer parameters.

Example of a Stratigraphical Profile



See also

Stratigraphical Profiles tab

Layer Definition tab

Stratigraphical Profiles tab

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Profile	Name of stratigraphical profile	Single click
Elevation Topsoil	Elevation of natural ground with respect to support nodes elevation (Ex. 0.0 m)	Single click
Elevation Water	Elevation of water table (Ex: -12.0 m)	Single click

N.B. Elevations can be negative or positive.

To have a look at a stratigraphical profile cross-section, refer to topic below.

See also

Invalid Stratigraphical profile
Stratigraphical Profile Cross-Section
Stratigraphical Profiles spreadsheet
Layer Definition tab

Layer Definition tab

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Rank	Rank of soil layer. Rank 1 is just below topsoil.	Single click
Thickness	Thickness of layer (ex.: 12 m)	Single click
Soil Name	Name of soil as defined in one of available **Soils Spreadsheet**, which makes up the layer (ex.: Clay)	Double-click

See also

Stratigraphical Profiles tab
Layer Definition tab

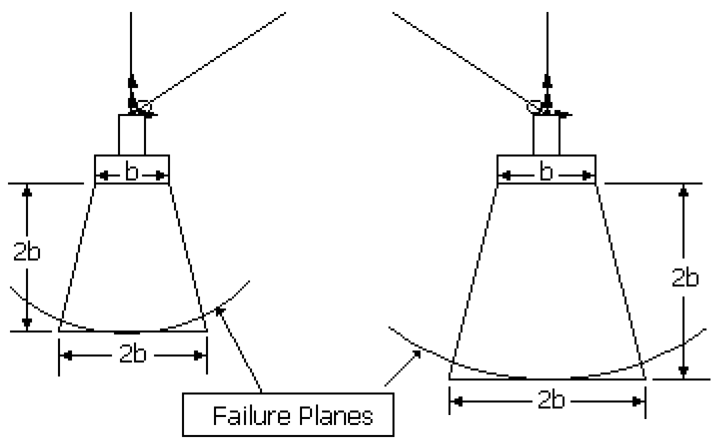
Invalid Stratigraphical Profile

If you obtained the following message:

"Elevation of the soil failure plane is below the bottom of the deepest soil layer. Please modify the thickness of soil layers or reduce footing dimensions.

Explanation:

VisualDesign calculates the failure plane under each footing. This soil volume is trapezoidal and its depth corresponds to about 2 times the footing width (each direction). If the bottom of the deepest soil layer is located within this volume, you get this message. Look at the figure below.



Rock elevation must be located below the soil failure planes

What can you do?

- Check the elevations that you entered in the **Stratigraphical Profiles** spreadsheet. They must be compatible with the elevation of support nodes;
- Check if bottom of the deepest soil layer is deeper that you expected;
- Reduce footing dimensions in order to reduce the depth of the soil failure plane and launch another analysis;
- If the message still appears, add piles under the footings.

See also

Stratigraphical profiles spreadsheet

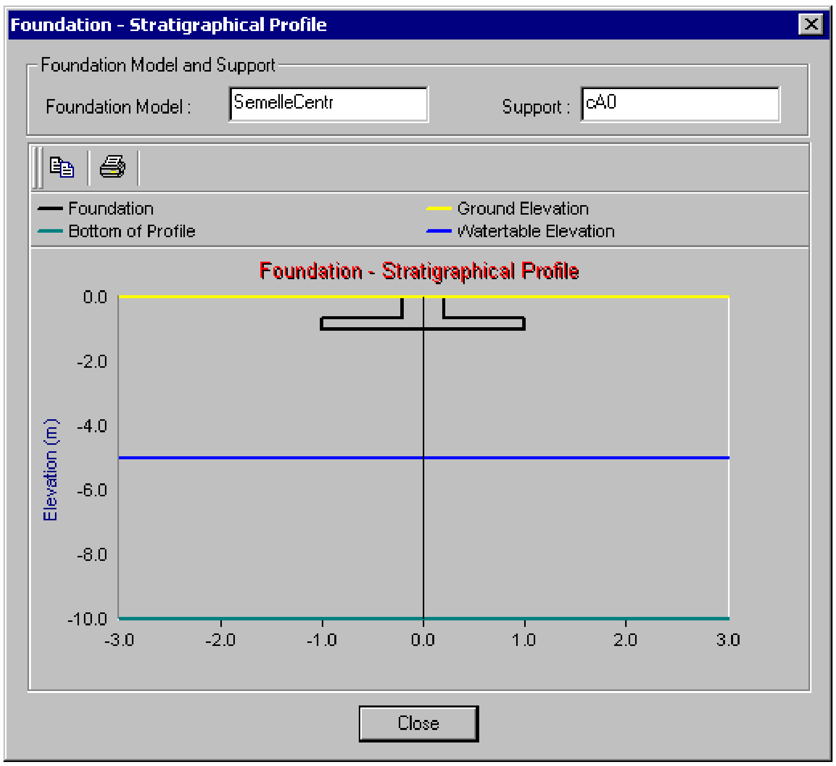
Layer Definition tab

Display the Stratigraphical Profile Cross-Section

Cross-section of Stratigraphical Profile

Check elevations of soils composing the stratigraphical profile at a specific support. Follow the procedure below:

- Activate the Structure mode;
- Click on a support (or more) and go to menu **Structure** / **Supports**.
- Click in any cell located in the line representing the support in the **Standards Supports** spreadsheet and right click. The spreadsheet contextual menu will appear on the screen. Select the *Details* function.



You can print this graph by clicking the *Print* button available in this dialog box.

See also

Stratigraphical Profiles spreadsheet

Standards Supports spreadsheet

Specifications

Specifications for Shallow Foundation

To design or verify a shallow foundation, you must create a **Shallow Foundation Specifications** spreadsheet. Afterwards, a specification must be selected in the **Design** tab of **Shallow Foundation Models** dialog box.

This spreadsheet is available in the **Structure** menu, under **Specifications**.

Here is a description of parameters that you need to define:

Group: Structural data

Column	Description	Editing
Specification ID	Automatically calculated	No
Number	Specification number (16 alphanumerical characters)	Single click
Type of Analysis	Indicate in which case the specification will be used for: Design or Verification. **Verification of footings**: The footing dimensions are verified and bearing capacity is calculated. If the footing thickness is insufficient, vf >vr. If so, modify the thickness and launch another analysis. Look at calculated *Hmin* in the **Shallow Foundation Results** spreadsheet. **Design of footing**: Footing dimensions and reinforcement will be designed. The footing thickness is not modified. Look at minimum thickness, *Hmin,* calculated by VisualDesign.	Double-click
Bx max	Design: Enter a maximum dimension of footing according to global axis system. VisualDesign will not exceed it during the design. Verification: Enter the actual footing dimension according to global axis system.	Single click
Bz max.	Design: Enter a maximum dimension of footing according to global axis system. VisualDesign will not exceed it during the design. Verification: Enter the actual footing dimension according to global axis system.	Single click
Fix a dimension	Fix dimension Bx or Bz for a design, or choose n/a.	Double-click

Column	Description	Editing
Saf	Forces amplification factor. Refer to topic Saf and Waf Factors.	Single click
Waf	Amplification factor for the effective width of footing. See Saf and Waf Factors	Single click
Design		
Code	Choose the standard that will be used for the design of reinforcement in the footing.	Double-click
Rebar Material	Specify the steel grade for rebars that will be placed in the footing.	Double-click

See also

Shallow Foundation Models Spreadsheet

The Design tab

Specifications for Deep Foundations

Go to **Structure / Specifications** and create a deep foundation specification that is going to be applied for the geotechnical and structural verification of the deep foundation.

Afterwards, while modeling the foundation, you will select this deep foundation specification in the **Piles** tab of the **Deep Foundation Models** dialog box.

Here is a description of parameters that you need to define:

Group: Structural data

Column	Description	Editing
Specification ID	Automatically calculated	No
Number	Specification number (16 alphanumerical characters)	Single click
Type of Analysis	For the moment, only the (geotechnical and structural) verification of piles is available. The program does not design piles.	Double-click
Steel Specification	Choose a steel specification for the structural verification of piles.	Double-click

Shallow Foundation Models

Read the following topics before defining a shallow foundation model: "Fields of application for theoretical calculation models" and "Fields of application for empirical calculation models".

If you decide not using the Foundation Models Wizard, we recommend that you enter data through corresponding dialog boxes instead of spreadsheets because dialog boxes are more explicit and contains more fields than spreadsheets. To open the dialog box when the spreadsheet is open, select any cell in the Foundation spreadsheet (first one), right click, and choose function "Details".

See also

Foundation Modeling Wizard

Shallow Foundation Models Dialog Box

Fields of application for theoretical calculation models

Fields of application for empirical calculation models

Saf and Waf Factors

Rebar Layout in the Footing

Shallow Foundation Models spreadsheet

Fields of Application of Empirical Calculation Models

Standard penetration test (SPT)

The standard penetration test (SPT) is widely used in the field for the study of foundation sites on granular soils. This test measures the number of blows required for a standardised split core sampler to penetrate 300 mm into the soil by striking the respective soil layer. The number of blows corresponds to the standard penetration index (N). According to the NBC 1995, the allowable contact pressures obtained through this method must be compared to the pressures due to unfactored loads. In addition, in accordance with the NBC 1995, this method is useful in the initial calculation of foundations as well as the final calculation for most ordinary buildings. However, foundations for tall buildings and towers, and special buildings that are sensitive to movement or built on sensitive soil must be assessed using theoretical calculation models.

Vane shear test

For a footing on cohesive soil, the bearing capacity is generally governed by undrained resistance (cu). In practice, this resistance is suitably estimated by means of the vane shear test, which simulates the undrained load adequately.

The vane shear test applies to sensitive and non-sensitive clays. Strip, rectangular or square footings may be used. Footings may be either sunken or not sunken.

Static penetrometer test

End bearing (qc) is measured in the interval from B/2 on top of the base of the foundation to 1.1B below it.

A static penetrometer can be used for:

- Cohesive and coarse-grained soils;
- Strip or square footings;
- Df / B ≤ 1.5.

Fields of Application of Theoretical Calculation Models

	Terzaghi*	Meyerhof	CFEM	Hansen	Vesic
Df <= B	●	●	●	●	●
Df >= B		●	●	●	●
Shallow foundation	●	●	●	●	●
Deep foundation				●	●
Strip, square or rectangular footing		●	●	●	●
Vertical or sloped load		●	●	●	●
Footing next to a slope				●	●
Cohesive soils	●	●	●	●	●
Granular soils		●	●	●	●
Moment		●	●	●	●

* Above all for estimating q ultimate.

See also

Shallow Foundation Model Dialog Box
Deep Foundation Model Dialog Box

Bearing Capacity Equations

General Equation:

$$q_{ult} = \overbrace{c \cdot N_c \cdot \zeta_c}^{\text{cohesion term}} + \overbrace{\frac{1}{2} \cdot B \cdot \gamma_H \cdot N_\gamma \cdot \zeta_\gamma}^{\text{depth term}} + \overbrace{\gamma_{Df} \cdot D_f \cdot N_q \cdot \zeta_q}^{\text{live load term}}$$

where,

qult = ultimate bearing capacity of the soil (kN/m^2);

c = cohesion (kPa); please note that:

c = c', effective cohesion, used for effective stresses analysis (long term analysis).

c = cu, undrained cohesion, used for total stresses analysis with cohesive soils (short term analysis).

B = the foundation effective width (m).

γ'H = effective unit weight of the soil under the footing, inside the rupture zone (H) (kN/m^3).

γ'Df = effective unit weight of soil at the top of the footing, inside a zone located between the ground surface and the footing base (kN/m^3).

Nc, Nγ, Nq = bearing capacity factors used for the calculation of cohesion, depth and live load term (factors depending on the chosen calculation model).

ζc, ζγ, ζq = correction factors used for the calculation of cohesion, depth and live load terms (factors depending on the chosen calculation model).

Standard Penetration Test (SPT)

$$N_1 = C_n \cdot N_{in-situ} = \sqrt{\frac{P_{atm}}{\sigma'_{vo}}} \cdot N_{in-situ}$$

where:

Nin-situ = Value of N index, measured in the field.

Cn = Correction factor for the effective stress.

N1 = Corrected N index for effective stress.

Patm = 101,3 kPa (or 1 ton /square foot).

σ'vo = effective vertical stress at the depth corresponding to the measure of the N index.

BOWLES MODEL (1988):

1) If B is ≤ 1,2 m, then:

$$q_{allowable-net} = \frac{N'}{F_1} \cdot K_d$$

2) If B is ≥ 1,2m, then

$$q_{allowable-net} = \frac{N'}{F_2} \cdot \left(\frac{B+F_3}{B}\right)^2 \cdot K_d$$

where:

q allowable = allowable net bearing capacity (kPa) corresponding to a settlement of 25 mm.

N' = adjusted standard penetration test index for an effective stress of 100 kPa and an effective rate equal to 70% of the delivered energy through the hammer (i.e. N'70; you must follow the procedures proposed by Bowles (1988, chap.3.7);

Kd = 1 +0,33·(Df / B)

F factors depend on the energy rate and are given below:

TABLE 1.1 FACTOR F FOR THE BOWLES MODEL (1988)

Factor F	N'55	N'70
F1	0,05	0,04
F2	0,08	0,06
F3	0,30	0,30

Field Vane Test

$$q_{ult} = 5 \cdot \mu_f \cdot c_u \cdot \left[1 + 0{,}2 \cdot \frac{D_f}{B}\right] \cdot \left[1 + 0{,}2 \cdot \frac{B}{L}\right] + \gamma \cdot D_f$$

where,

qult = gross ultimate bearing capacity (kPa);

cu = undrained resistance measured in the field vane test (kPa);

μf = reduction factor proposed by Bjerrum (1973);

Df = footing depth (m);

B = footing width (m);

L = footing length (m);

γ = unit weight of material above the base of footing (kN/m^3)

Saf and Waf Factors

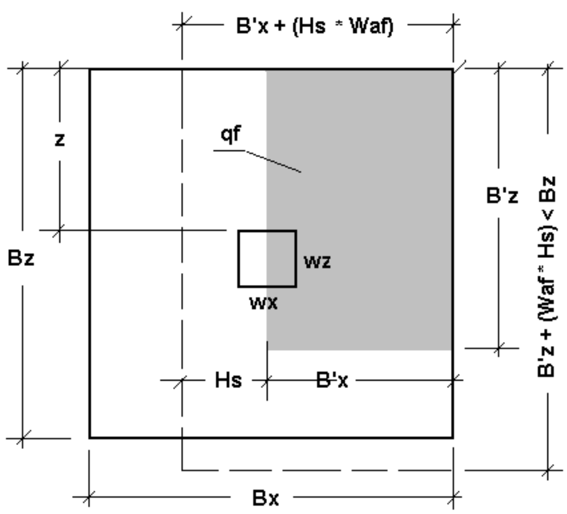
Factor "Waf" allows the redistribution of the calculated qf on a larger width. We recommend the following width:

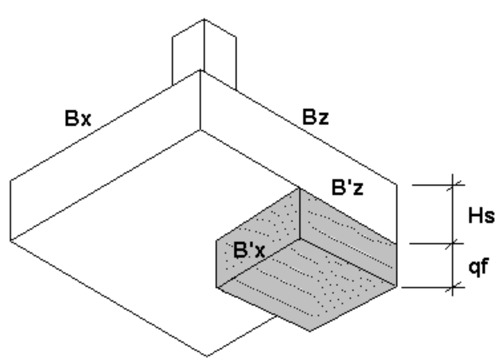
New width = effective footing width for the analyzed direction + at least 1Hs (Footing thickness).

If you wish to redistribute the forces on a greater width, all you have to do is to increase factor Waf:

Effective width = B eff + (Waf * Hs)

If the axis of gravity is the Y axis:





B’x = Bx - 2·eccentricity_x

B'z = Bz - 2·eccentricity_z

$$\text{Mfz (kN} \cdot \text{m / m)} = \frac{qf \cdot z^2 \cdot B'x}{2(B'x + Hs \cdot Waf)} \cdot \textbf{factor}$$

$$\text{Mfx (kN} \cdot \text{m / m)} = \frac{qf \cdot x^2 \cdot B'z}{2(B'z + Hs \cdot Waf)} \cdot \textbf{factor}$$

$$\text{Qfx (kN/m)} = \frac{qf \cdot x \cdot B'z}{(B'z + Hs \cdot Waf)} \cdot \textbf{factor}$$

$$\text{Qfz (kN/m)} = \frac{qf \cdot z \cdot B'x}{(B'x + Hs \cdot Waf)} \cdot \textbf{factor}$$

factor = alpha · m + b

alpha = B'x or B'z (according to the analyzed direction) / Bx or Bz

b = (Saf – 0.3) / 0.7

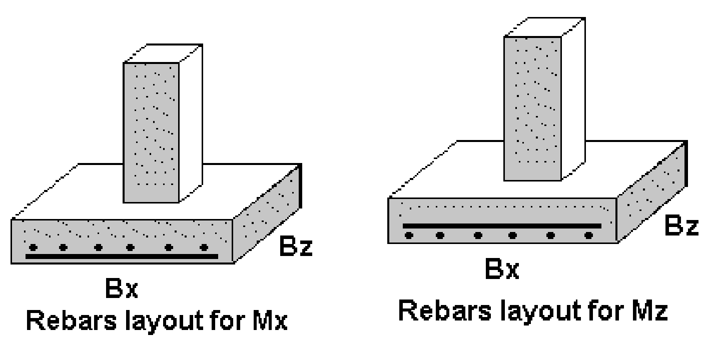
m = 1 - b

If **factor** > Saf, then the **factor** = Saf

We introduced the forces amplification factor to make up for the fact that VisualDesign™ distributes a uniform pressure under the footing. Actually, we should consider a triangular or trapezoidal distribution of forces under the footing, which is caused by eccentricities towards one or two directions. A triangular distribution of stresses can only be obtained from a non-linear analysis. Then, we fixed the following default values: Saf = 1.25 and Waf = 1.50. When there is not any eccentricity in the calculation of a footing, the Saf and Waf factors are not considered.

Rebars Layout in the Footing

If axis of gravity is the Y axis:



Shallow Foundation Models Dialog Box

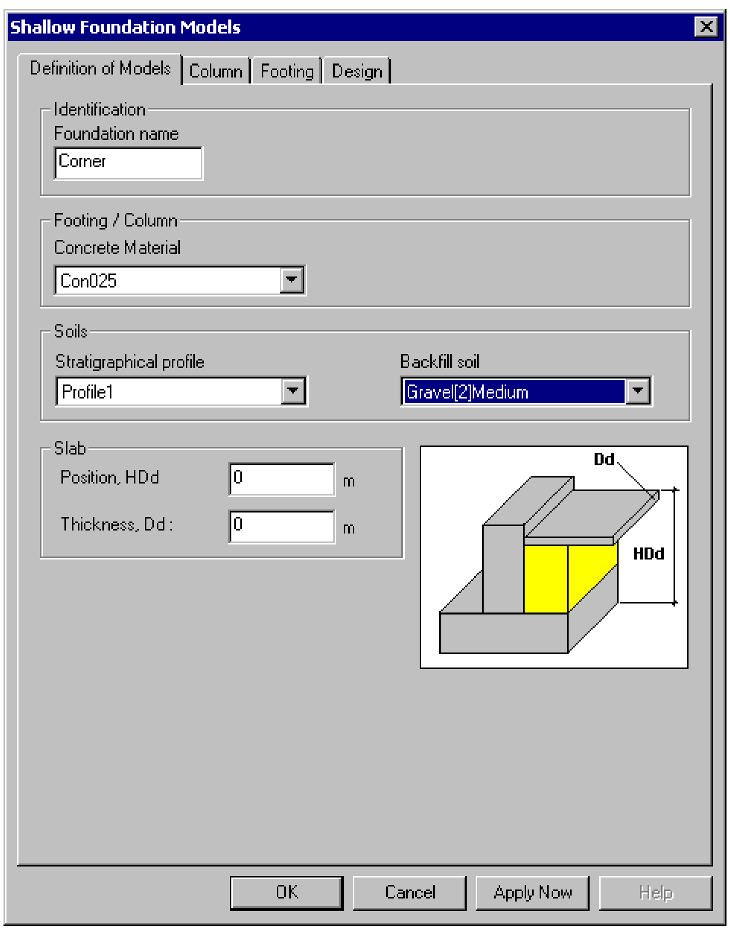
It is important that you enter data in the dialog box, because it contains more fields and it is much more explicit than columns included in the **Shallow Foundations** spreadsheet. Spreadsheets are useful for printing, editing and sorting data.

To open the **Shallow Foundation Models** Dialog box, do as follows:

- Create a stratigraphical profile (**Structure / Stratigraphical Profiles**).
- Define a specification for the shallow foundation model (**Structure / Specifications / Shallow Foundations**.
- Go to **Structure / Foundation Models** / **Shallow**.
- Insert a line in the spreadsheet. Click in any cell, right click, and choose the *Details* function in the contextual menu.

This dialog box includes the following tabs: **Model Definition**, **Column, Footing** and **Design** tabs.

The Model Definition Tab



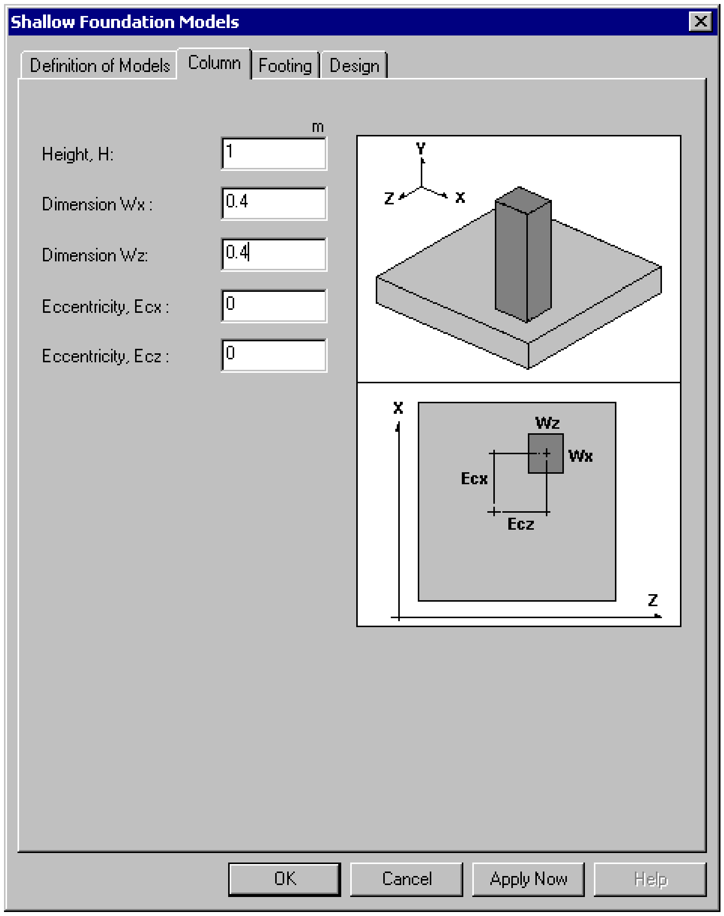
Here is a description of the **Model Definition** tab:

Field	Description
Identification	
Foundation Name	Name of this shallow foundation model
Footing/Column	
Concrete Material	Click on the arrow and select a concrete material for the footing and column.
Soils	
Stratigraphical Profile	Click on the arrow and select a stratigraphical profile as defined in the Stratigraphical Profiles spreadsheet
Backfill Soil	Click on the arrow and select a backfill soil. The soil must be defined in one of the **Soils** spreadsheets.
Slab	
Position HDd	Position of the slab: Height from bottom of footing to top of slab
Thickness Dd	Thickness of the slab.

The Column Tab

You must complete this tab to define the concrete column or wall above the footing.

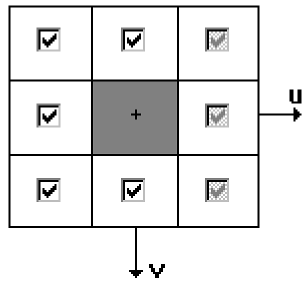
Field	Description
Column	
Height H	Column height measured from top of footing.
Wx Dimension	Column x-dimension (according to global x-axis).
Wz Dimension	Column x-dimension (according to global x-axis).
Eccentricity Ecx	Column eccentricity x (calculated from centre of footing) according to global x-axis.
Eccentricity Ecz	Column eccentricity z (calculated from centre of footing) according to global z-axis.

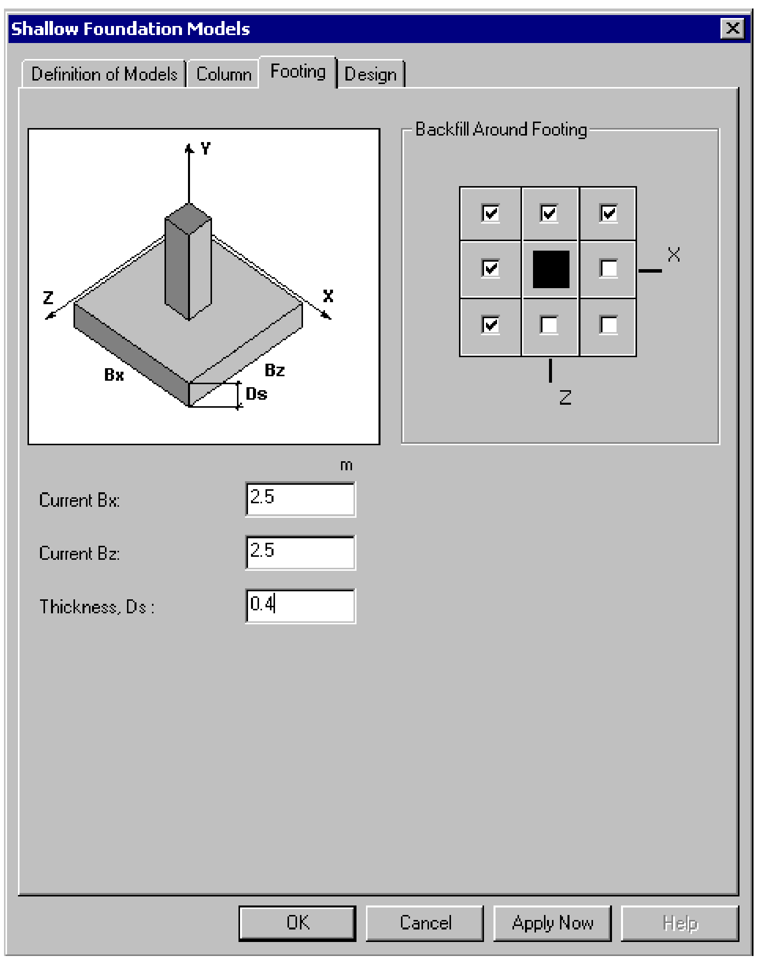


The Footing Tab

Field	Description
Dimensions	
Bx max or Current Bx	Design: Enter the maximum dimension of the footing according to the global x-axis. Verification: Enter the current footing dimension.
Bz max or Current Bz	Design: Enter the maximum dimension of the footing according to the global z-axis. Verification: Enter the current footing dimension.
Thickness Ds	Footing thickness
Concrete material	Select the concrete material for this footing model.
Slab	
Position HDd	Position of top slab from bottom of footing.

Field	Description
Angle of Friction Soil/Footing	Angle of friction between soil and footing.
Backfilled part(s) of the footing	See the explanation below.





Backfill over the footing

This tool describes the location of backfill material over the footing. The plane view of the footing is surrounded by check boxes. They can be of three kinds: checked, checked and shaded, or unchecked. Here is a description of each type:

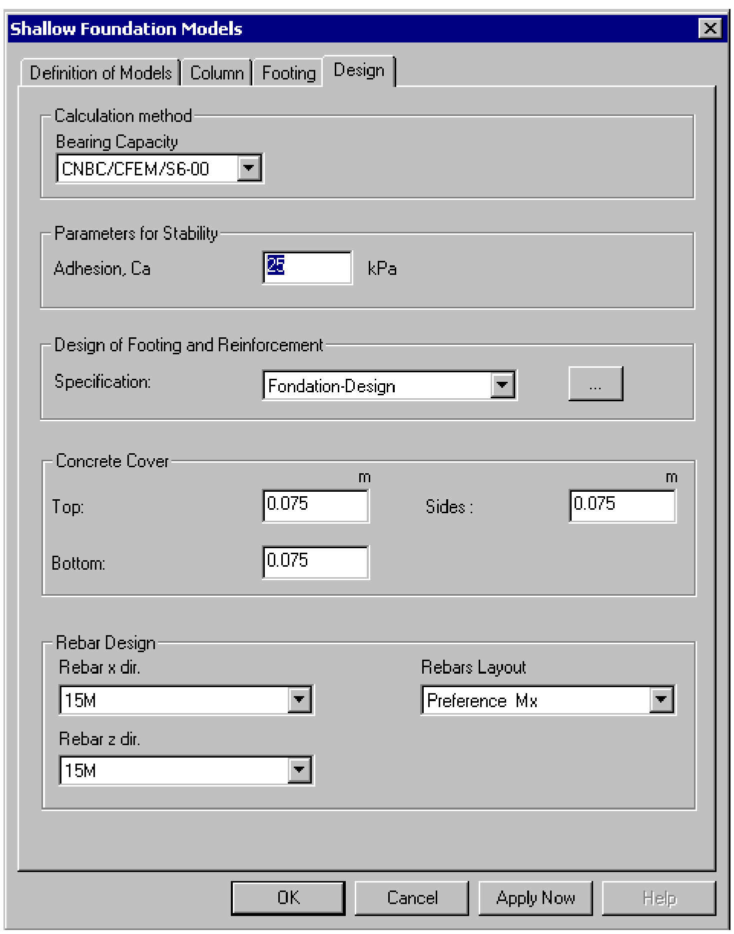
Checked: there is backfill over the footing and it is located outside of the building. The backfill thickness is calculated from natural ground to top of footing.

Checked and shaded: there is backfill over the footing and it is located inside of the building. The backfill thickness is taken from bottom of slab to bottom of footing. If the slab is located directly above the footing, there is no backfill considered.

Unchecked: there is no backfill.

The Design Tab

This tab includes shallow foundation design criteria that you must complete in order to run a verification or design of footing dimensions Bx and Bz, and reinforcement.



Field	Description
Calculation Method	
Bearing Capacity	Choose a method for the calculation of bearing capacity. See also "Fields of application for theoretical calculation models" and "Fields of application for empirical calculation models".
Stability Parameter	
Adhesion, Ca	Adhesiveness of footing and soil (kPa).
Design of Footing and Reinforcement	
Specification	Select a specification for the design or verification of this foundation model. To add or modify a specification, press the button […] located at the right of this field. It will open the **Shallow Foundation Specifications** spreadsheet.
Concrete Covers	
Top	Specify the concrete cover at the top of footing.
Bottom	Specify the concrete cover at the bottom of footing.
Sides	Specify the concrete cover around the lateral sides of the footing.
Design of Rebars	
Dir. x	Select a type of rebar that will be placed along the footing global x-axis.
Dir. z	Select a type of rebar that will be placed along the footing global z-axis.
Rebar Layout	Select an option for the placement of rebars: *Favour Mx* or *Favour Mz*. Refer to topic Rebar Layout

See also

Shallow Foundation Model Dialog Box

Shallow Foundation Spreadsheet

Go to **Foundation Models** in the **Structure** menu and select the **Shallow Foundation** spreadsheet. The spreadsheet is divided into four spreadsheets to facilitate the consultation of data: **Foundation**, **Footing**, **Column** and **Design**.

Spreadsheets are useful to consult data, copy foundation models or sort data.

Foundation Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Name	Foundation name	Single click
Profile	Corresponding stratigraphical profile defined in the Stratigraphical Profiles tab	Double-click
Concrete Material	Concrete material for the footing, as defined in the **Concrete Materials** spreadsheet**.**	Double-click
Backfill material	Backfill material placed above the footing.	Double-click
Slab Thickness Dd	Slab thickness	Single click
Slab Position HDd	Height between bottom of footing and top of slab	Single click

The Footing Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Name	Name of the foundation model.	Single click
Bx max or Actual Bx	Design: Maximum dimension of the footing in the global x direction. Verification: Actual dimension of the footing in the global x direction.	Double-click

Column	Description	Editing
Bz max or Actual Bz	Design: Maximum dimension of the footing in the global z direction. Verification: Actual dimension of the footing in the global z direction.	Double-click
Thickness Ds	Footing thickness	Single click

The Column Spreadsheet

This spreadsheet is part of both shallow and deep foundations spreadsheets.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Name	Name of the foundation model.	Single click
Dimension Wx	Column dimensions according to global x-axis.	Single click
Dimension Wz	Column dimensions according to global z-axis.	Single click
Height H	Height of column measured from top of footing.	Single click
Ecx x Direction	Eccentricity of concrete column (calculated from centre of footing) in the global x-direction.	Single click
Ecz z Direction	Eccentricity of concrete column (calculated from centre of footing) in the global z-direction.	Single click

The Design Spreadsheet

This spreadsheet includes parameters that are needed to design the footing dimensions and reinforcement.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Name	Name of the foundation model.	Single click
Specification	Choose a shallow foundation specification for the design or verification of this footing.	Double-click

Column	Description	Editing
Bearing Capacity Equation	Choose the method used for the calculation of bearing capacity. Please refer to topics Fields of application for theoretical calculation models and Fields of application for empirical calculation models.	Double-click
Adhesion Ca	Soil/Footing adhesiveness (kPa).	Single click
Concrete Cover Top	Specify the concrete cover at the top of footing.	Single click
Concrete Cover Bottom	Specify the concrete cover at the bottom of footing.	Single click
Concrete Cover Sides	Specify the concrete cover around (sides) of footing.	Single click
Rebars Layout	Choose a rebar layout: Favour Mx or Favour My. Refer to topic Rebar Layout	Double-click
Rebars x - Dir.	Select rebars that will be placed along the footing x-global axis.	Double-click
Rebars z- Dir.	Select rebars that will be placed along the footing z-global axis.	Double-click

See also

Shallow Foundation Model dialog box

Shallow Foundations spreadsheet

Deep Foundation Models

If you decide not using the Foundation Models Wizard, we recommend that you enter data through corresponding dialog boxes instead of spreadsheets because dialog boxes are more explicit and contains more fields than spreadsheets. To open the dialog box when the spreadsheet is open, select any cell in the Foundation spreadsheet (first one), right click, and choose function "Details".

See also

Foundation Modeling Wizard

Calculation of Friction and Point Bearing Capacity

Piles' Spring Supports

Deep Foundation Models Dialog Box

Deep Foundation Models spreadsheet

Calculation of Friction and Point Bearing Capacity for Piles

Ref: *Manuel Canadien des fondations*, clause 20.2.1.2 (3)

Calculation of Friction Bearing Capacity

Granular Soils

The Beta method is always used with these types of soils. Consequently, the parameters Beta for driven and bored piles shall be defined and not equal to zero in the **Granular Soils** spreadsheet. If so, the friction bearing capacity of piles in contact with this type of soil will be equal to zero.

Then, the compression/friction global resistance factor shall be specified in the **Foundation** tab of **Project Configuration** dialog box.

Qsu =Sum (Beta * Sigma' * Pile perimeter * delta L) per soil layer in contact

Qsr = Phi compression/friction * Qu

Where "delta L" is the thickness of the soil layer considered

Note: The perimeter of a pile composed of a W shape is equal to (b+d)*2

Cohesive Soils

The Alpha method is always used with these types of soils. The soil cohesion is included in equations, so this parameter shall be different from zero in the **Cohesive Soils** spreadsheet.

The soil cohesion must be less than 100 kPa, with respect to the *Manuel Canadien des Fondations*.

1. Steel Piles:

Alpha = 0.00005 * Cohesion ^2 - 0.0125 Cohesion + 1.1339 - 0.3367 / Cohesion

2. Concrete Piles:

If cohesion > 25 kPa:

Alpha = 0.00008 * Cohesion ^2 - 0.0201 * Cohesion + 1.9008 - 11.111 / Cohesion

If cohesion < 25 kPa:

Alpha = 1.0

Calculation of Point Bearing Capacity

Values for Nt (driven and bored piles) shall be specified in the **Granular Soils** spreadsheet. Also, the global resistance factor for Compression/Point shall be specified in the **Foundation** tab of **Project Configuration** dialog box.

Granular Soils

Qpu = Nt * Sigma' *At

Qpr = Phi Compression/Point * Qpu

If Nt values are not specified in the **Granular Soils** spreadsheet, the SPT method will be used. Thus, the value for N1-60 is required:

Qpu = m*N1-60*At

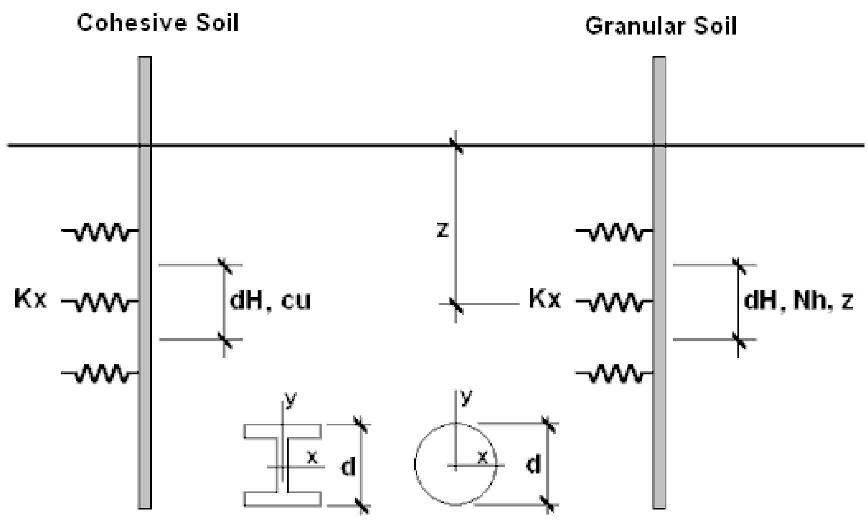
Where the value for "m" is given at page 324, *Manuel Canadien des fondations* (MCF).

Cohesive Soils

Qpu = Nt * Cohesion * At

Where values for Nt is given at article 20.2.1.2 (4) (MCF)

Piles' Spring Supports



In accordance with the Canadian Foundation Manual (page 347), the spring support stiffness Kx and Ky is determined as follows:

Kx = Ky = Ks * d * dH (N/m)

Where

Ks = 67 cu / d (N/m^3) for cohesive soils;

Ks = nh z / d (N/m^3) for granular soils.

When many layers are composing the soil, a factored value for Ks is calculated, considering each layer of soil and its respective thickness.

In current version 5.9, the projected area is considered for each direction, allowing the calculation of spring stiffness in the proper way: Kx will be related to "d" and Ky, to "b".

See also

The Piles tab (Deep Foundation Models)

Deep Foundation Models Dialog Box

It is important that you enter data in the dialog box, because it contains more fields and it is much more explicit than columns included in the **Pile Foundations** spreadsheet. Spreadsheets are useful for printing, editing and sorting data.

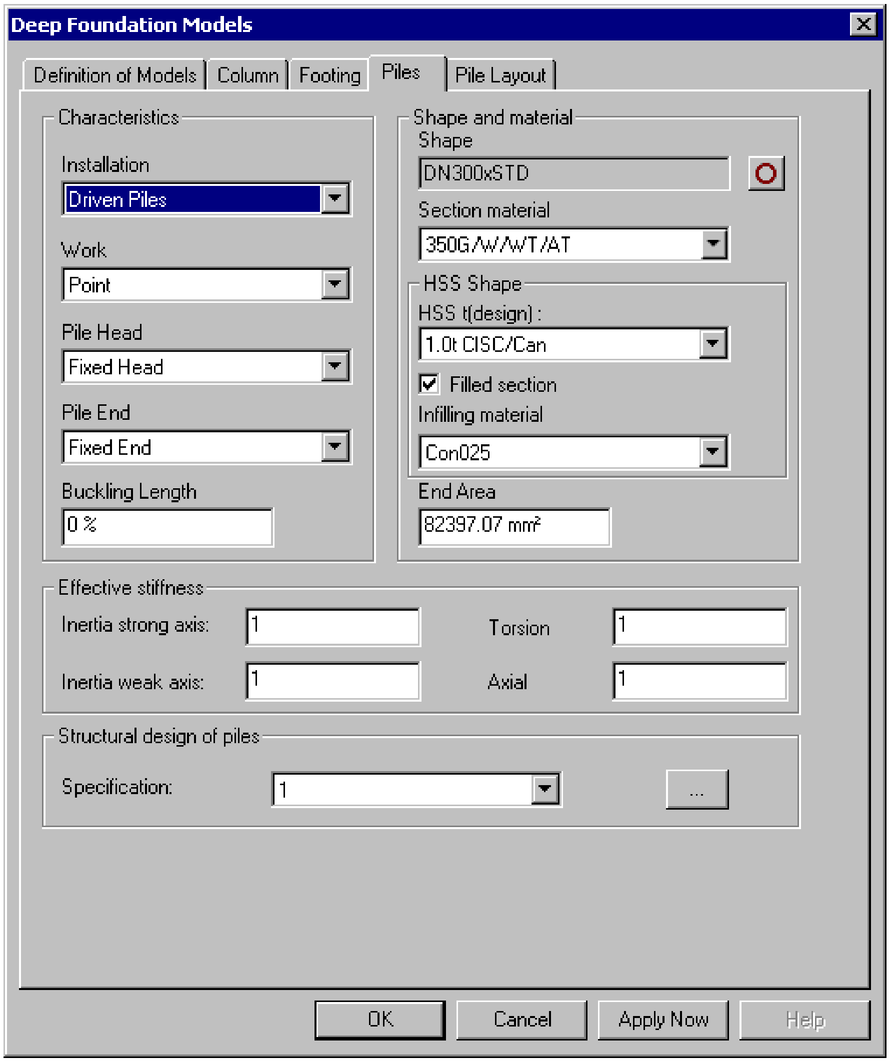
To open the **Deep Foundation Models** Dialog box, do as follows:

- Create a stratigraphical profile (**Structure / Stratigraphical Profiles**).
- Define a specification for a deep foundation (**Structure / Specifications / Deep Foundations**).
- Go to **Structure / Foundation Models** / **Deep**.
- Insert a line, click in any cell, right click, and select the *Details* function in contextual menu.

This dialog box includes the following tabs: **Model Definition**, **Footing**, **Column**, **Piles** and **Piles Layout** tabs.

The three first tabs are identical to those described for Shallow Foundation Models.

The Piles Tab

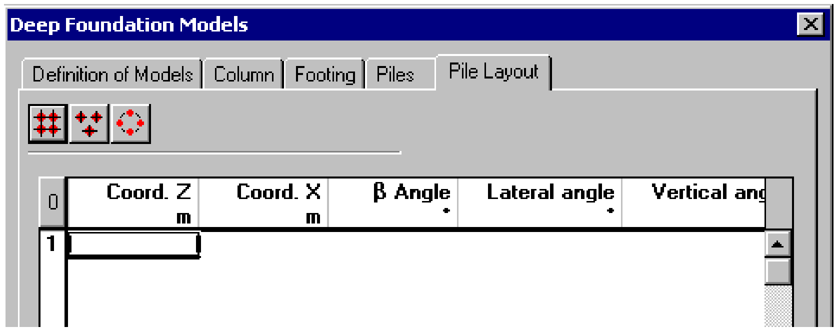


Field	Description
Characteristics	
Installation	Type of installation of piles: Driven or bored
Work	Specify if the piles are working in point, friction, or both point and friction.
Piles Head	Specify if the pile head is fixed or unrestrained.
Piles End	Specify if the pile end is fixed or unrestrained.
Buckling length	Unsupported length of the pile expressed in % or fraction of total length.

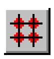
Field	Description
Shape and Material	
Shape	Choose a pile section by pressing the I beam icon. Expand the shape selection tree and double click to select one.
Shape Material	Click on the arrow and expand the Material selection tree. Double-click to select a material.
End Area	Area at the end of the pile. This area is recalculated whether the section is filled or not
HSS Shape	
HSS with 0.9t	Check this box if the chosen HSS has been manufactured according to grade ASTM A500.
Filled Section	Tick off this box if the tubular section is filled with concrete.
Infilling Material	Click on the arrow and select the concrete material that filled the tubular section.
Effective stiffness	
Inertia strong axis	Specify the ratio of effective inertia on the section strong axis for the structural verification in bending.
Inertia weak axis	Specify the ratio of effective inertia on the section weak axis for the structural verification in bending.
Torsion	Specify the ratio of effective stiffness for torsion/shear for the structural verification of the section.
Axial	Specify the ratio of effective axial stiffness for the structural verification of the section in compression.
Structural Design of Pile	
Specification	Click on the arrow and choose a specification for the verification of the deep foundation model. To add or modify a specification, press the button […] located at the right of this field. It will open the **Deep Foundation** Specifications spreadsheet.

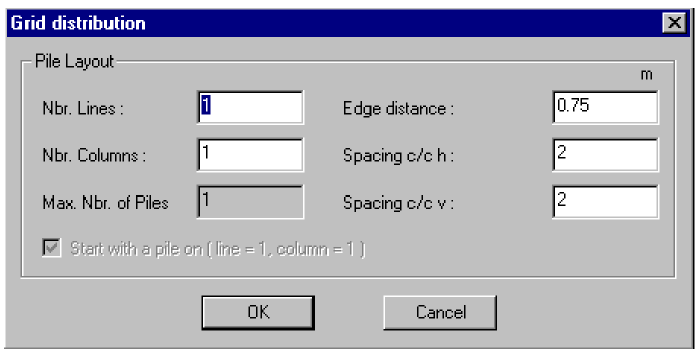
Piles Layout Tab

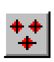
The **Piles Layout** tab allows you to enter, visualize, and print the piles layout according to the coordinates entered in the spreadsheet.

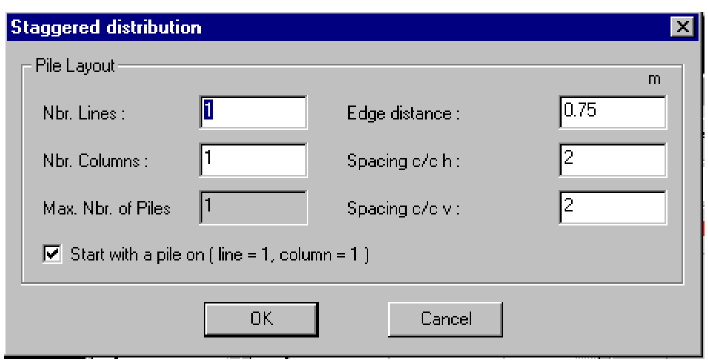


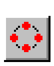
Use one of the three buttons to quickly define the piles layout coordinates. Each one is representing a particular piles layout and is calling up a dialog box.

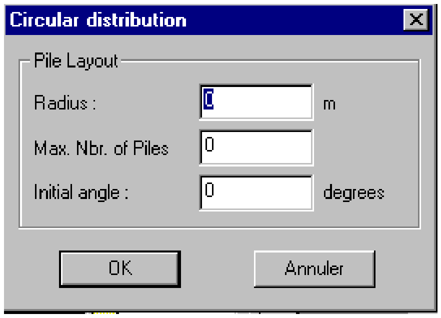




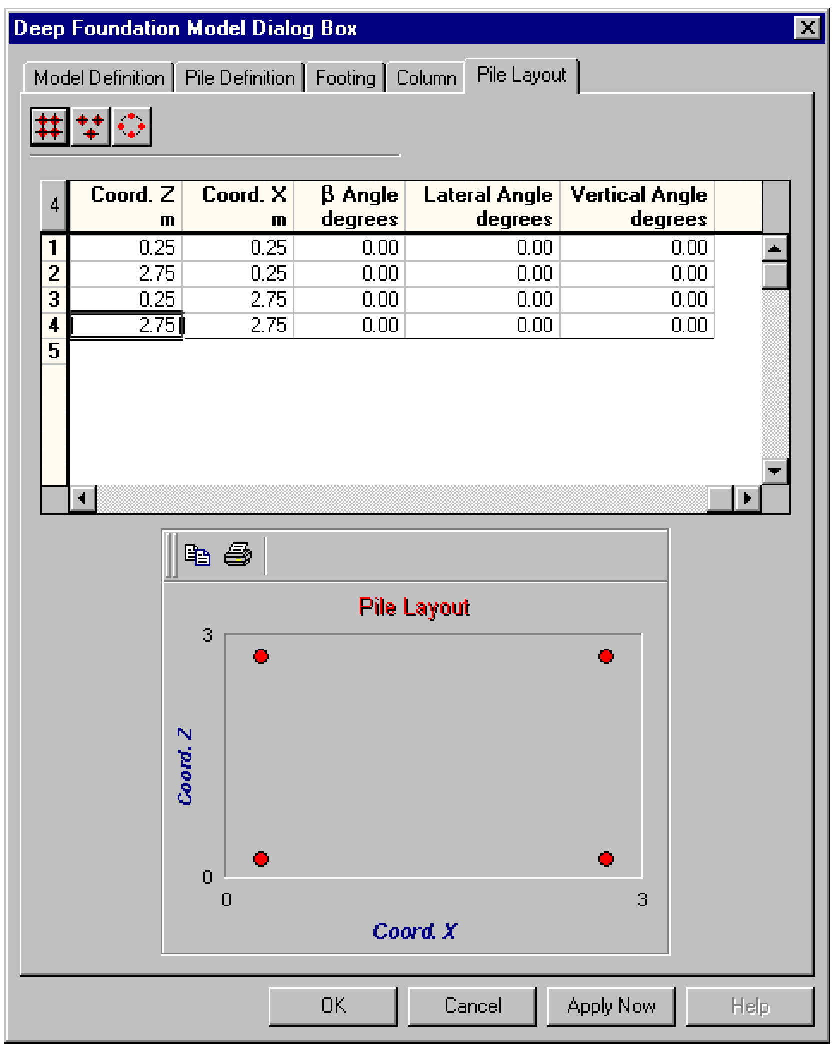








Following input, visualize and print the piles layout that will be drawn in the dialog box, as the example shown below:



The spreadsheet includes the following data:

Field	Description
ID	Automatically calculated
Coord. X, Y, and Z	Pile coordinates, according to the convention described below.
Beta angle	Beta angle of the pile section
Lateral angle	Lateral slope of the pile, expressed in angle
Vertical angle	Vertical slope of the pile, expressed in angle

Deep Foundations Spreadsheet

This spreadsheet, which is located in the **Structure/Foundation Models** menu, is divided into four tabs: **Foundation**, **Footing, Column** and **Piles**. The three first tabs are identical to those part of the shallow foundation models spreadsheet.

Spreadsheets are useful to consult or sort data and to copy foundation models using the **Duplicate** function of contextual menu.

Piles Spreadsheet

Group: Structural data

Column	Description	Editing
ID	Automatically calculated.	No
Name	Foundation name	Single click
Specification	Choose a specification for the verification of the deep foundation.	Double-click
Shape	Chosen shape for the pile. To modify this choice, double-click in this cell to open the Shape selection tree and double click to select a shape.	Double-click
Material Pile	Chosen material for the pile.	Double-click
Filled	If the shape is filled with concrete, choose option [x].	Double-click or Space bar
Infilling Material	Concrete material that fills the tubular section.	Double-click

Column	Description	Editing
End Area	Area of pile end.	Single click
Installation	Type of installation for the piles: Driven or Bored.	Double-click
Work	Indicate the way the piles are working: in point bearing, friction bearing or both.	Double-click
Piles Cap	Indicate the pile head end condition: fixed or unrestrained.	Double-click
Piles End	Indicate the end conditions of pile end: fixed or unrestrained.	Double-click
Buckling length	Buckling length: Percentage or fraction of pile length.	Single click
Effective inertia for bending strong axis	Factor (or percentage) that indicates the effective inertia that must be considered for bending on the strong axis. A factor of 1.0 or percentage of 100% means that there is no reduction.	Single click
Effective inertia for bending weak axis	Factor (or percentage) that indicates the effective inertia that must be considered for bending on the weak axis. A factor of 1.0 or percentage of 100% means that there is no reduction.	Single click
Effective stiffness for torsion/shear	Factor (or percentage) that indicate the effective stiffness of the pile that must be considered for torsion and shear. A factor of 1.0 or percentage of 100% means that there is no reduction.	Single click
Effective axial stiffness	Factor (or percentage) that indicate the effective axial stiffness of the pile that must be considered for compression and tension. A factor of 1.0 or percentage of 100% means that there is no reduction.	Single click
HSS t (design)	Wall thickness that will be considered during analysis of this HSS.	Double-click

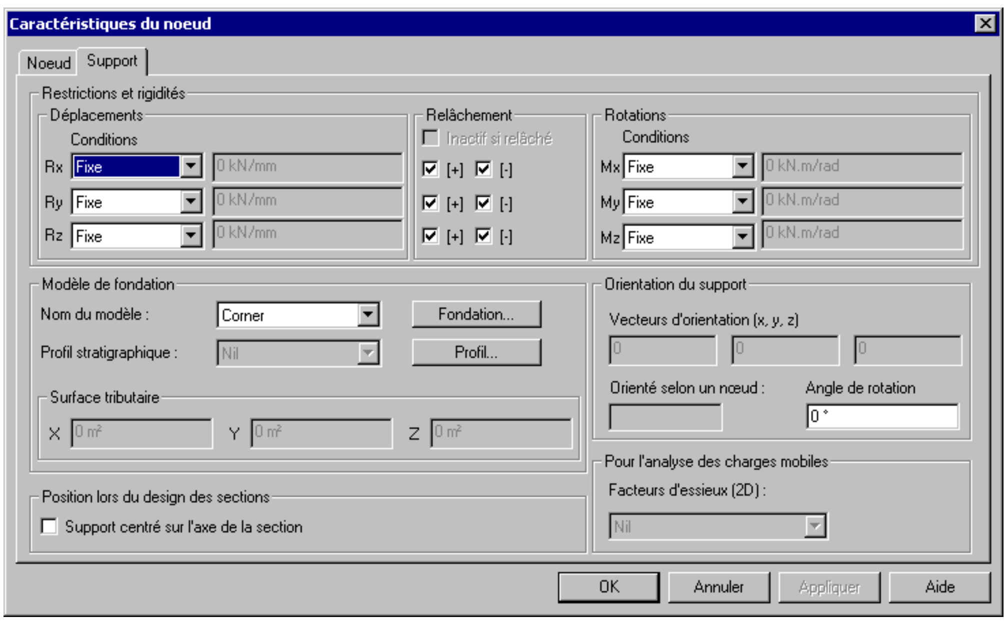
Foundation Supports

Assigning Foundation Models to Supports

A foundation model can be assigned to one support or more. Generally, the same foundation model is assigned to supports that would transmit loads of similar magnitude to the foundation.

To assign a foundation model to one support or more:

- Activate the "Support" icon on Elements toolbar.
- Double-click on a support or select many and click the **Properties** icon. Go to the **Support** tab of **Node Characteristics** dialog box.
- In the "Foundation model" heading, select a foundation model among the drop-down list box or click the "Foundation" button to define one. (This button opens one of the Foundation Models dialog box.)
- Click OK.



Tributary Areas for Spring Supports

Use this tool, which is located in **Structure** menu / **Tools**, to automatically calculate tributary areas in the x-, y- and z-direction, for spring supports that are associated to plate elements.

Procedure:

- Activate the Structure activation mode.
- Select spring supports.
- Go to **Structure / Tools** and select **Calculation of Tributary Areas**.

Calculated areas will be written in the **Support** tab (**Node Characteristics** dialog box).

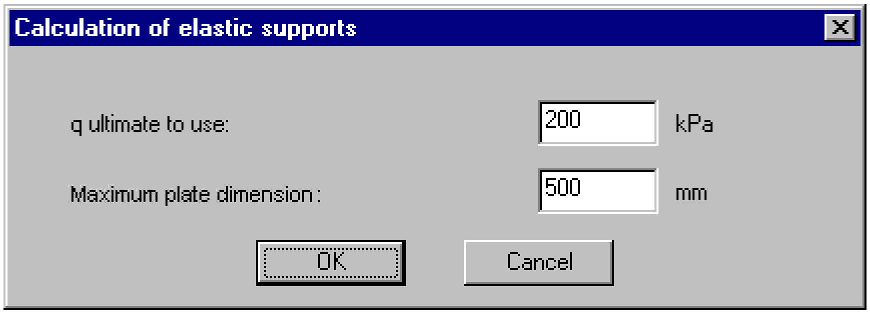
N. B. For the Generation of Abutments, Piers & Retaining Walls module, spring support tributary areas are automatically calculated from the stratigraphical profile data and are indicated in the **Support** tab.

Foundation Transformation

The function **Foundation Transformation** is available in **Structure/Tools** menu and is only used with the Abutment, Pier & Retaining Wall module, for the moment. The function transforms a footing into plates and adds elastic supports. For pile foundation, the function transforms the pile into members and adds rigid links from pile heads to footing.

- Rename your file to keep bearing capacity results.
- Activate the Structure mode.
- Activate the Support element on Elements toolbar.
- Select the foundation central support.
- Select the **Foundation Transformation** function.

The following dialog box will appear on screen. VisualDesign indicates the bearing capacity that will be used for the calculation of spring supports. You can specify the maximum dimension of plates that will be composing the footing after the transformation.



See also

Abutments, Piers, and Retaining Walls Generation and Design

Soil/Structure Interaction

Analysis with soil-structure interaction

The secant moduli K of soils is used to calculate footing settlements.

To model the soil/structure interaction, select the option "Secant mod. K" as degrees of freedom for selected supports in the **Support** tab (**Node Characteristics** dialog box) and then, assign a foundation model to these supports. Please refer to the topic Secant modulus K for Foundation Supports.

When the soil-structure interaction is considered, the program performs iterations until the results of the foundation settlement, the stresses in the structure and the reactions at the supports caused by the settlement converge.

The process for calculating soil-structure interactions rarely requires more than five iterations. If more than five iterations are required, the solution cannot be optimized. The results, however, are valid since the calculated percentages for capacity are greater than 100%.

Procedure:

- Select all the supports of only some of them, and press the properties icon to call up the **Support** tab of **Node Characteristics** dialog box. Choose the option Secant modulus K for appropriate degrees of freedom.
- Create a load case and select an *Interaction* type of load in the **Loads Definition** spreadsheet.
- Go to the **Load Combinations Generation Wizard**. Select a code and generate ultimate and serviceability load combinations.
- Run a static analysis.
- Select a load combination on Activation toolbar and look at results in menu **Results** / **Foundations**.

Secant Modulus K for Foundation Supports

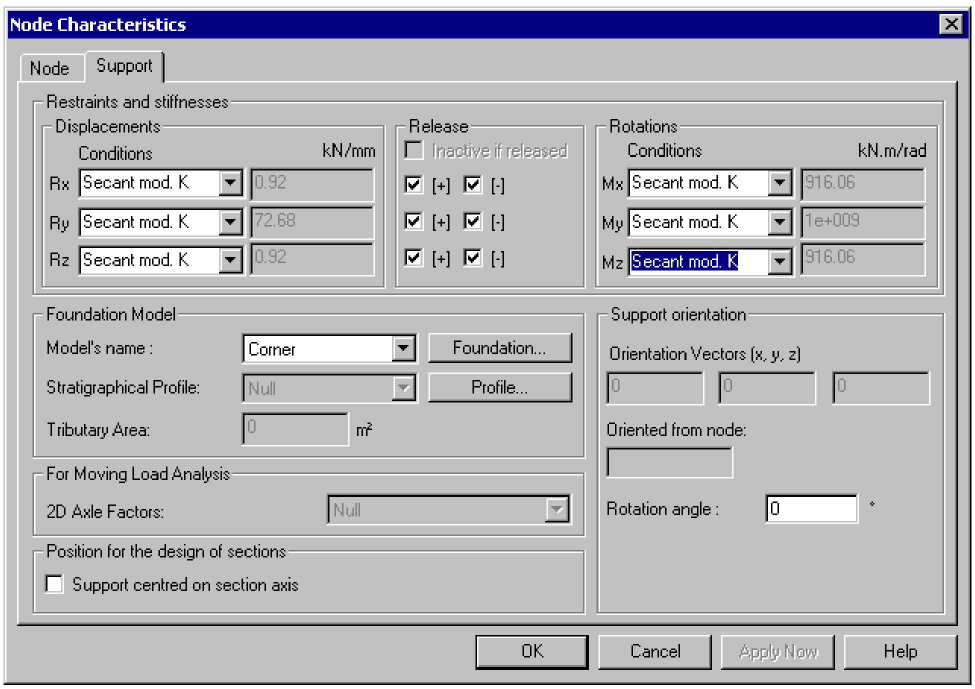
If you own the Foundation Design module, VisualDesign can automatically calculate the linear and torsional rigidity of spring supports. It is done using the properties of soils composing the stratigraphical profile, which is specified in a foundation model. Lateral and vertical displacements of the footing or slab can be obtained from these calculated values.

The secant modulus must be selected as degrees of freedom for a support if soil/structure interaction needs to be considered.

Procedure:

- Define a shallow foundation model.
- Select supports and press the **Properties** icon.
- In the **Support** tab, choose option *Secant mod. K* as restraints for Rx, Ry, Rz, Mx, My, and/or Mz degrees of freedom.
- Assign supports to foundation models.

The linear and torsional rigidity of the support will be calculated and written in the shaded field next to each degree of freedom corresponding to *Secant modulus K*.



See also

Support's Degrees of Freedom

Analysis without Soil-Structure Interaction

Soil-structure interaction is not considered if the option "Secant Modulus K" is not assigned to support degrees of freedom.

Analysis

Limit States Design

The limit conditions calculation is used to design foundations. The limit conditions calculation uses a **global bearing factor** rather than a global safety factor. At limit conditions, a global resistance factor of 0.5 is equivalent to a safety factor of 3.0 when calculating allowable stresses. Global resistance factors must be specified in the **Foundation** tab of **Project Configuration** dialog box.

A limit condition occurs at the point at which a structure ceases to fulfil the function for which it was designed. Limit conditions that affect safety are called ultimate limit conditions and they occur when the load bearing capacity of the structure is surpassed, resulting in reversal, slipping, rupture or fatigue. The service limit conditions pertain to the use for which the structure is designed and concern bending, vibrations, permanent deformations and cracking.

See also

Serviceability Limit State
Definition of Load Combinations

Serviceability Limit States

Serviceability limit states pertain to the use for which the structure is designed and concern bending, vibrations, permanent deformations, and cracking. Service limit conditions are verified with specific load factors.

Foundation Design Procedures

Project Configuration

- Configure the project foundation parameters in the **Foundation tab** of **Project Configuration.**

Definition of Soils

- Define the soils that you need for your project, in the soils spreadsheets (**Soils / Common).** Create personalized soils if needed, by inserting lines at the end of spreadsheets.

Definition of Foundation Models

- Use the **Foundation Modeling Wizard**. In addition to the foundation model, this wizard generates a stratigraphical profile and a specification that applies to this model.

OR

- Define the stratigraphical profiles in the **Stratigraphical Profiles** dialog box **Structure** menu. Make sure that the depth under footings is sufficient to consider the soil plane of failure under them.
- Define a foundation specification.
- Define the foundation model in one of the **Foundation Models** dialog box (**Structure** menu).

Supports

- Assign foundation models to supports.

Load Combinations

- Define load combinations using the **Load Combination Generator**. The dead load of backfill over footings is automatically considered in dead load cases.

Analysis

- Run a static analysis or a design.

Results

- Go to **Results / Foundations**.

Foundation Results - General

All results obtained during the analysis are stored. There are two ways to view these results: Graphically or through spreadsheets. Results spreadsheets are listed under **Results/Foundations**.

Graphic View Options

Spreadsheet Shallow Foundation Results

Footing Reinforcement

Piles Foundations Results

Piles Reactions

Note: Results must be analysed carefully to detect the slightest data entry error.

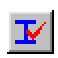
See also

The Foundation Design module

Results tab of View Options

Display the structural and geotechnical design load

Display the Foundation Structural or Geotechnical Design Load

- Activate the *Design Results* mode  and open the **View Options** dialog box.
- Select the **Limits** tab and activate the "Legend" box. Then, select the **Results** tab and activate one option among the two available: "Structural Design Load" or "Geotechnical Design Load".
- To modify the font style and text height, go back to the **Limits** tab and click the "Font" button.

Shallow Foundations

Structural Design Loads:

- The displayed design load will be the maximum among those calculated for overturning, sliding, uplift, and shear.

Geotechnical Design Loads:

- The displayed design load will be the maximum among those calculated for shear, bending moment, and reinforcing bars in the footing.

Deep Foundations

Structural Design Loads:

- Piles: The displayed design load will be the maximum among those calculated for shear, bending moment & compression, and the strength of the steel shape.
- Footing: N/a

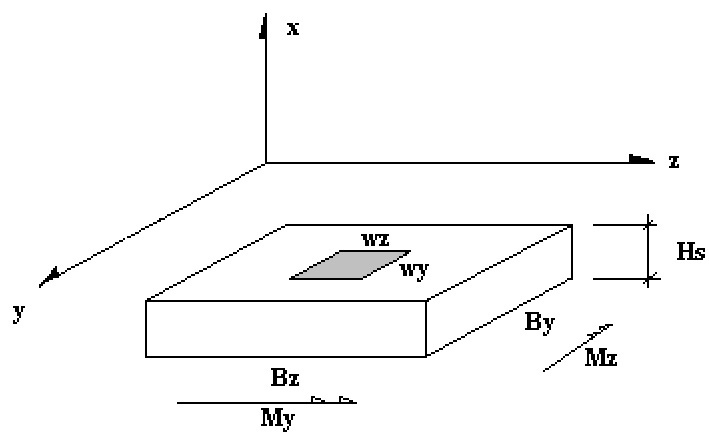
Geotechnical Design Loads:

- Piles: The displayed design load will be the maximum among those calculated for point bearing, friction bearing, or both.
- Footing: The displayed design load will be the maximum among those calculated for global design load, compression, or tension.

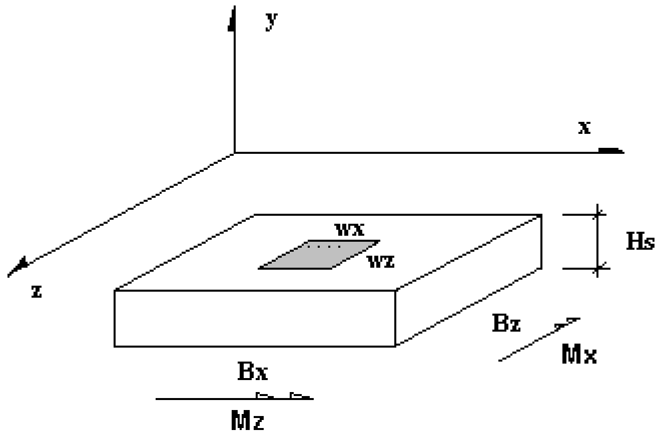
See also

Results tab of View Options
The Limits tab
The Results tab
Display the Pressure and Capacity of Spring Supports

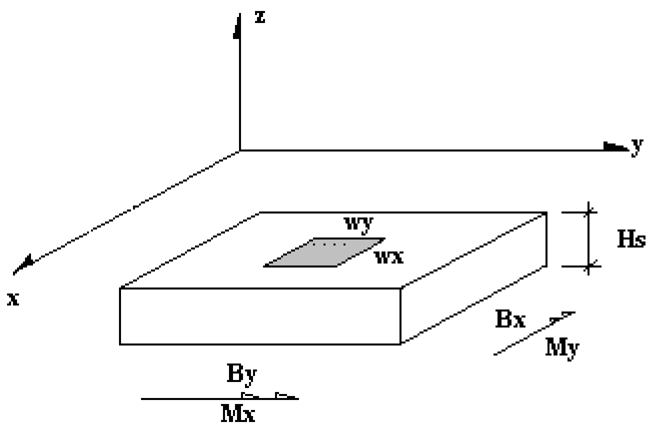
Sign Convention of Forces in the Footing



Gravity Axis towards X



Gravity Axis towards Y



Gravity Axis towards Z

See also

Saf and Waf Factors

Position of rebars in the footing

Footing reinforcement spreadsheet

Specification for Shallow Foundations

Shallow Foundation Results

Shallow Foundation Results Spreadsheet

Once that the static analysis is completed, open this spreadsheet and look at the calculated bearing capacity of the foundation, for each load combination. If it is insufficient, qr = 0. Increase the footing dimensions or add piles.

To consult this spreadsheet, go to **Results/Foundation / Footings**.

Group: Design Results

Column	Description	Editing
Model	Name of the shallow foundation model.	No
Support	Support node associated with the foundation model	No
Load combination	Analysed load combination	No
Bx Bz	Check: Current dimensions of the footing Design: Calculated dimensions of the footing	No
Bx eff Bz eff	Effective dimension of the footing relative to global x, and z-axis.	No
Bearing Capacity qr	Factored bearing capacity of soil under the footing. If this value is zero, the bearing capacity is insufficient. Look also at *Hmin*, in the **Footing Reinforcement** spreadsheet.	No
Contact Pressure qf	Pressure of soil under the footing	No
Settlement dy	Footing settlement. Settlements are automatically calculated for service load combinations if foundations supports are modeled using the secant modulus of soil. Refer to Secant modulus of soil	No
Rfx Rfy Rfz	Reaction (ultimate or service) transmitted to the foundation, relative to the x, y and z-axis.	No
Mfx Mfy Mfz	Bending moment (ultimate or service) transmitted to the foundation, relative to the x, y and z-axis.	No

Column	Description	Editing
Design load Bearing capacity	Footing Design Load =(qr / qf)*100	No
Hrs (1)	Geotechnical horizontal resistance of footing in the soil near the footing/soil interface.	No
Hri (1)	Geotechnical horizontal resistance of footing at footing/soil interface.	No
Hf x-dir.	Horizontal sliding force acting on the foundation, in the global x direction.	No
Hr x-dir.	Resistance of the foundation against sliding, in the global x direction.	No
Design load Sliding x-dir.	Design Load of footing considering sliding: = (ΣF horiz / ΣF vert.) x 100	No
Hf z-dir.	Horizontal sliding force acting on the foundation, in the global z direction.	No
Hr z-dir	Resistance of the foundation against sliding, in the global z direction.	No
Design load Sliding z-dir.	Sliding Design Load of footing: = (ΣF horiz/ΣF vert.) x 100	No
Eccentricity Ratio x-dir.	This ratio is equal to the calculated eccentricity x over the allowed eccentricity x, according to chosen model (Foundation tab of Project Configuration).	No
Eccentricity Ratio z-dir.	This ratio is equal to the calculated eccentricity z over the allowed eccentricity z, according to chosen model. (Foundation tab of Project Configuration)	No
Uplift ratio	This ratio is equal to the factored uplift reaction of support over the factored dead load of footing.	No
Settlement ratio	Design load relative to allowable settlement: Settlement (%) = (allowable settlement/calculated settlement)*100	No
Result	Result of analysis: O.K., Eccentricity error, Settlement error, etc	No

Note (1): Refer to clause 6.7.5 "Factored Horizontal Resistance" of CAN/CSA-S6-00 Standard and clause 11.4.9.1 of CAN/CSA-A23.3-95 Standard.

See also

Foundation tab (Project Configuration)

Display Structural and Geotechnical Design Load

Footing Reinforcement Spreadsheet

Open this spreadsheet when that static analysis is completed and look at calculated footing reinforcement. Some columns are editable (yellow ones). You are allowed to modify dimensions of rebars and rebars layout in the footing. When you edit yellow columns, reinforcement is automatically recalculated.

Access the results of calculated reinforcement in the footing by selecting **Results / Foundation / Reinforcement.**

Group: Design Results

Column	**Description**	**Editing**
ID	Automatically calculated	No
Name	Name of the foundation model	No
Quantity	Number of footing of this model	No
Bx By Bz	Footing dimension according to global x- y, or z direction.	No
Hs	Footing thickness, as specified by the user.	No
Hmin	Minimum thickness of footing for shear and punching shear.	No
Concrete Material	Footing material	Double-click
Steel	Material of reinforcing bars.	Double-click
Rebars Layout	Rebars layout in the footing to favour Mx or Mz, as specified by user in the Design tab. See Rebars Layout.	Double-click
Concrete Cover Minimum	Minimum concrete cover for reinforcing bars in this footing.	Single click
Concrete Cover Sides	Concrete cover at the sides of footing.	Single click

Column	Description	Editing
Saf	Amplification factor for forces. Default value = 1.25. Refer to Saf and Waf Factors.	Single click
Waf	Amplification factor for the effective width of the footing. Default value = 1.50. Refer to Saf and Waf Factors	Single click
Rebars dimensions x-, y or z-dir.	Selection of Rebar in the specified direction.	Single click
No. Rebars x-y, or z dir.	Calculated number of rebar in the x, y and z direction.	No
Spacing Rebars c/c x-, y, or z dir.	Centre to centre spacing of rebar.	No
ρ x , ρ y, ρ z	Percentage of steel reinforcement in the specified directions.	No
Bar length x-, y-, or z-dir.	Length of reinforcement in the direction considered.	No
d x-, y-, or z- dir.	Calculated depth from the top of footing to the centre of gravity of reinforcement, in the x, y and z direction	No
Mfx Mfy Mfz	Bending moment (ultimate or service) in the footing, according to the specified direction.	No
Mrx Mry Mrz	Bending strength of footing, according to the specified direction.	No
Qfx Qfy Qfz	Shear force acting on footing, in the specified direction.	No
Qrx Qry Qrz	Shear strength of footing	No
vf	Punching shear stress in the footing	No
vc	Punching shear strength of footing	No
Structural Design Load	Structural design load of foundation according to the selected code (specification for shallow foundation).	No
Message	Result: OK, Qr < Qf, Mr < Mf, vc <vf.	No

Column	Description	Editing
Reinforcement	Weight of footing reinforcement (kg).	No
Concrete	Volume of concrete footings	No

Note If the user modifies some data into the spreadsheet, VisualDesign™ will automatically recalculate new data.

See also

Sign Convention for Forces in the Footing

Saf and Waf Factors

Position of Rebars into the Footing

Specifications for Shallow Foundations

Deep Foundation Results

Pile Foundation Results Spreadsheet

Access the results of pile foundation design by selecting **Results_**menu**->Foundations ->Piles**.

Group: Foundation Design Results

Column	Description	Editing
Model	Name of deep foundation model.	No
Support	Support node assigned to the foundation model	No
Load combination	Analysed load combination	No
Length	Length of Piles	No
Cr Geotech. Group	Geotechnical factored compression resistance of the pile group (kN).	No
Tr Geotech. Group	Geotechnical factored tension resistance of the pile group (kN).	No
dy	Settlement of the group of piles.	No
Rx, Ry, Rz	Reaction (ultimate or service) transmitted to the foundation, relative to the x, y and z-axis.	No
Mx, My, Mz	Bending moment (ultimate or service) transmitted to the foundation, relative to the x, y and z-axis.	No
Geotechnical Design Load Compression %	Design load in compression for the group of pile (%): (Reaction of the pile group / Compression strength of the pile group) *100	No
Design Load Settlement %	Percentage of allowable settlement: Allowable settlement = (calculated settlement / allowable settlement) *100	No
Results	Result of analysis for the considered load combination.	No

Graphic Results for Piles

Select a line in this spreadsheet, right click and select the function **Details** in the contextual menu.

Graphic Results for Piles

Select a line in the **Pile Foundation Results** spreadsheet, right click, and select the function *Detail* in contextual menu to have access to the following graphical results:

Pile Ultimate Capacity relatively to its Depth

Select a line in the spreadsheet, which corresponds to the support (pile foundation) that you want to look at, and press the button. The graph shows the pile capacity in point bearing, friction bearing, and both.

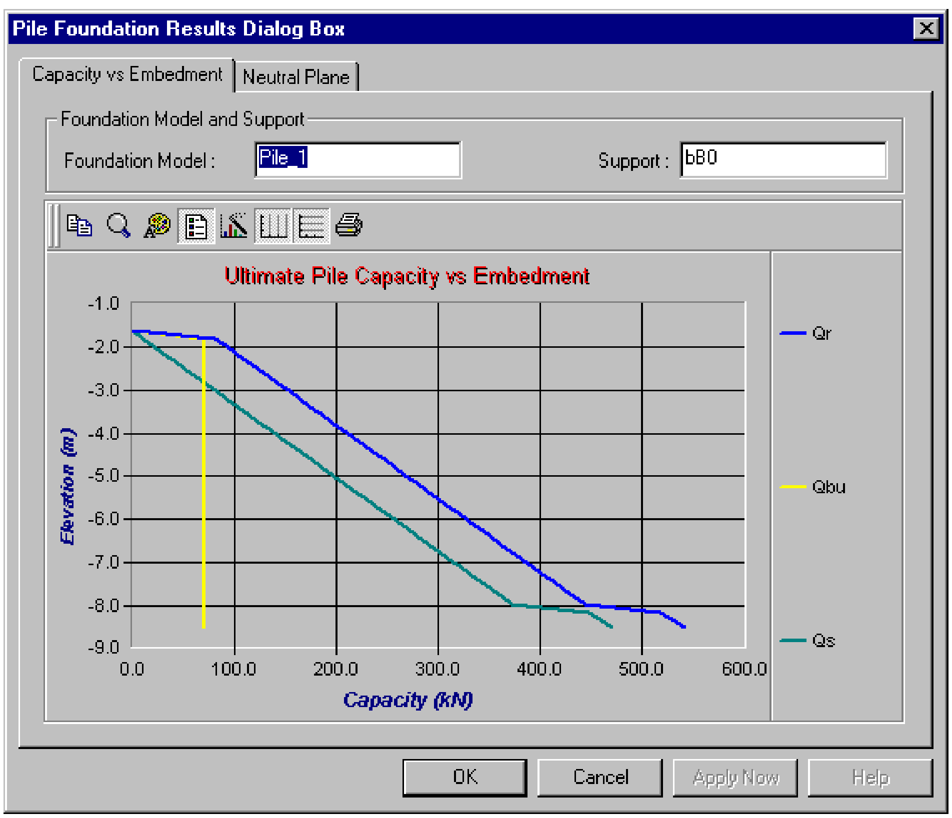
Qbr = ϕcp * Qb

Qb = nominal capacity in point bearing;
ϕcp = global resistance factor for compression/point;

Qsr = ϕcs * Qs

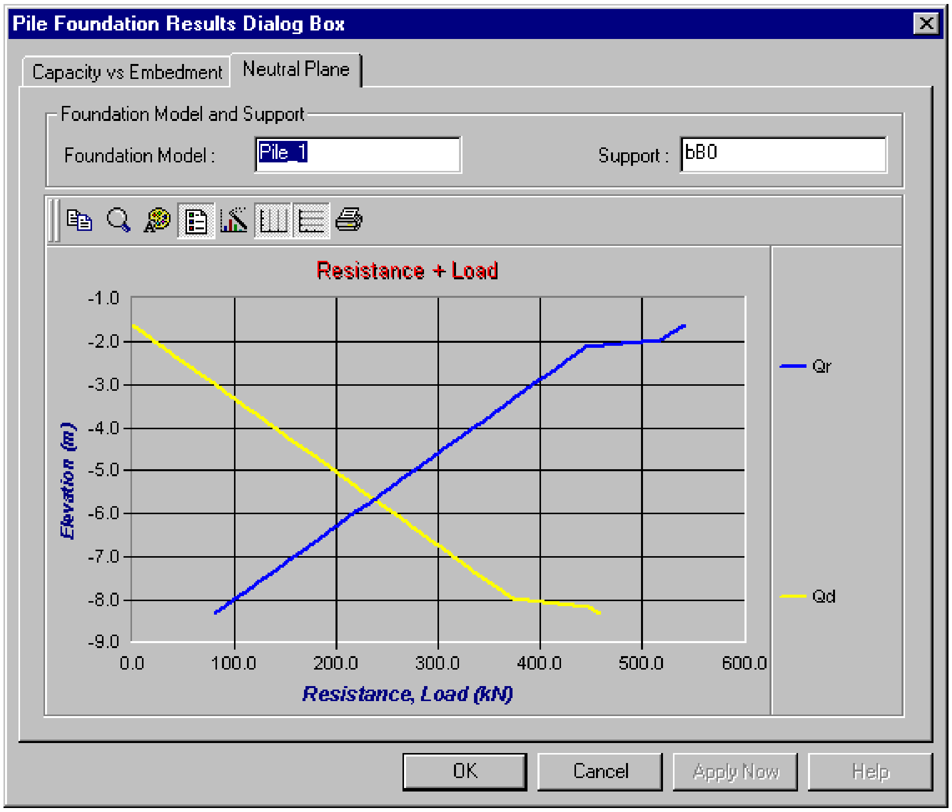
Qs = nominal capacity in friction bearing
ϕcs = global resistance factor for compression/friction;

Qr : Qbr + Qsr



Pile Neutral Plane

The location of neutral plane corresponds to the depth where negative lateral friction is equal to positive lateral friction.



Qd represents the dead load distribution along the pile depth; Qr is the variation of the total pile capacity (Qbr + Qsr) with the depth of the pile.

See also

Pile Foundation Results

Pile Forces and Reactions

Display the Pressure and Capacity of Spring Supports

Pile Forces and Resistance Spreadsheet

Access the results for the verification of piles foundation by selecting **Results / Foundations/Piles Reactions**.

Group: Design Results

Column	Description	Editing
Model	Name of the model of foundation	No
Support	Name of the support node	No
Load combination	Analysed load combination	No
Pile number	Pile number	No
Nz max	Maximum axial force in the pile	No
Nz min	Minimum axial force in the pile	No
Mfy	Bending moment concomitant with Nz, on weak axis.	No
Vfx	Shear force concomitant with Nz, on weak axis.	No
x	Lateral deflection of pile according to local x-axis.	No
% x	Ratio of pile deflection to allowable lateral deflection, according to local x-axis.	No
Mfx	Bending moment concomitant with Nz, on strong axis.	No
Vfy	Shear force concomitant with Nz, on strong axis.	No
y	Lateral deflection of pile according to local y-axis.	No
% y	Ratio of pile deflection to allowable lateral deflection, according to local y-axis.	No
Soil Pressure x Dir.	Soil pressure acting on the pile, according to local x-axis.	No
Kp sol x Dir.	Passive soil pressure on pile, according to local x-axis.	No
P / Kp x Dir.	Soil pressure divided by the passive soil pressure, according to local x-axis.	No

Column	Description	Editing
Soil Pressure y Dir.	Soil pressure acting on the pile, according to local y-axis	No
Kp sol y Dir.	Passive soil pressure, according to local y-axis.	No
P / Kp y Dir.	Soil pressure divided by the passive soil pressure, according to local y-axis.	No
Nr Structure	Axial strength of the pile	No
Mrx Structure	Bending strength of the pile on strong axis.	No
Vry Structure	Shear strength of the pile on strong axis.	No
Mry Structure	Bending strength of the pile on weak axis.	No
Vrx Structure	Shear strength of the pile on weak axis.	No
Structural Design load	Structural design load of the pile: (Critical force / pile strength)*100. A value greater than 100 means that the pile section is incorrect.	No
Geotechnical Resistance of Pile		
Cr Geotechn.	Compression strength of the pile.	No
Tr Geotechn.	Tension strength of the pile.	No
Geotechn. Design load Compression	Geotechnical design load of pile, considering compression force.	No
Geotechn. Design load Tension	Geotechnical design load of pile, considering tension force.	No

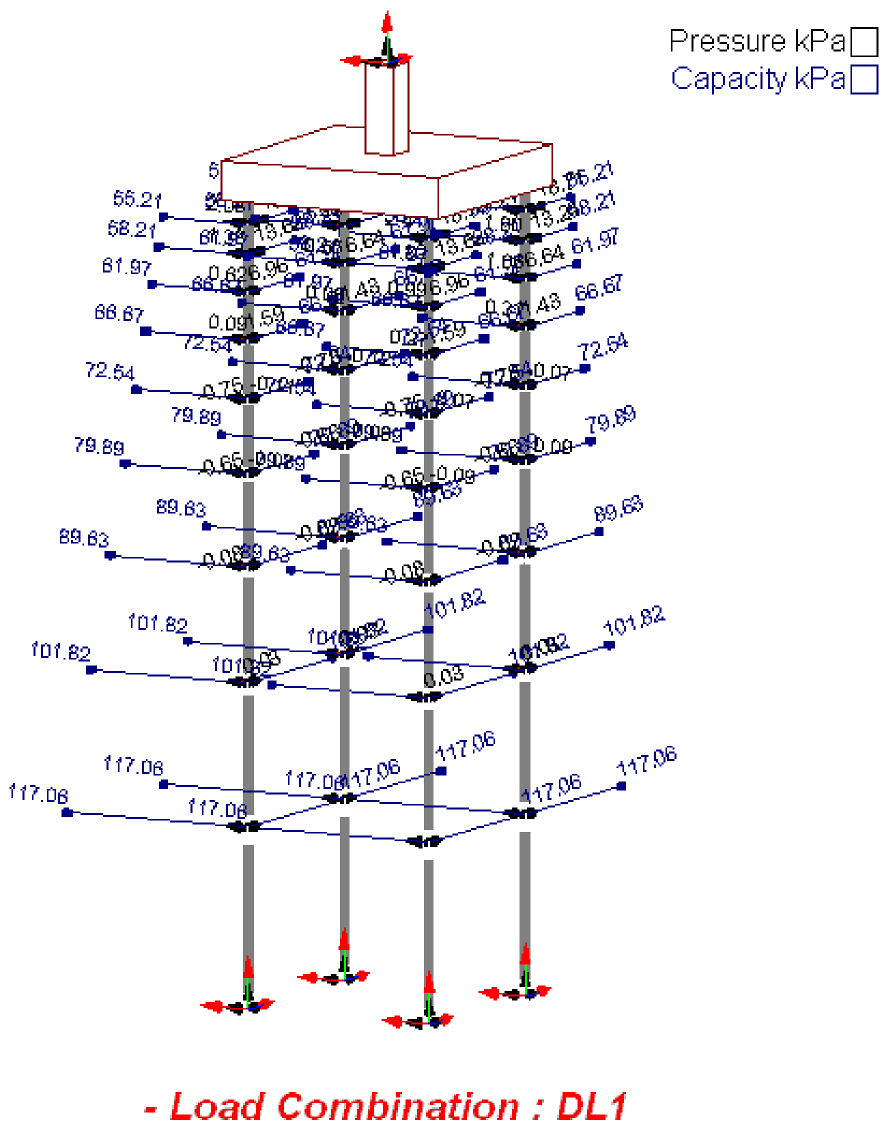
See also

Foundation Design Module
Piles tab
Piles Layout tab
Specifications for Deep Foundations
Display the Pressure and Capacity of Spring Supports

Pressure and Capacity of Spring Supports along Piles

Once that a pile foundation analysis is done, you can display pressures acting on each spring support located along piles along with the capacity of spring supports.

This option is available in the **Results** tab of **View Options** dialog box. Check the "Pressure and Capacity" box in the *Supports* section. A coloured legend will also appear to differentiate pressure diagram (orange) from the capacity diagram (blue). If you display numerical values, they will be the same colour as their respective diagram. The displayed values correspond to the envelope minimum and maximum forces.



References

AMERICAN ASSOCIATION OF STATE HIGHWAY AND TRANSPORTATION OFFICIALS (2004). *AASHTO LRFD Bridge Design Specifications* (SI Units), 3rd Edition, Washigton, D.C.

AMERICAN ASSOCIATION OF STATE HIGHWAY AND TRANSPORTATION OFFICIALS (1998). *AASHTO LRFD Bridge Design Specifications* (SI Units), 2nd Edition, Washigton, D.C.

AMERICAN CONCRETE INSTITUTE (1997). *ACI Design Handbook* (ACI 340R-97), Publication SP-17 (97), 6th Edition, ACI International, Farmington Hills, Michigan, 482 pp.

AMERICAN CONCRETE INSTITUTE (1999). *Building Code Requirements for Structural Concrete (318-99) and Commentary (318R-99)*, Reported by ACI Committee, ACI International, Farmington Hills, Michigan, 391 pp.

AMERICAN CONCRETE INSTITUTE COMMITTEE 436 (1966). *Suggested Design Procedures for Combined Footings end Mats*, ACI journal, Vol. 63, No.10, pp. 1041-1057, with discussion, pp.1537-1544.

AMERICAN INSTITUTE OF STEEL CONSTRUCTION, INC. (1995). *Manual of Steel Construction – Volume II – Connections,* ASD 9th Edition/LRFD 1st Edition.

AMERICAN INSTITUTE OF STEEL CONSTRUCTION, INC. (1992). *Manual of Steel Construction – Load Resistance Factor Design, Volume 1, Structural Members, Specifications, & Codes*, 2nd Edition.

AMERICAN INSTITUTE OF STEEL CONSTRUCTION, INC. (1989). *Manual of Steel Construction – Allowable Stress Design,* 9th Edition, Chicago, Illinois.

AMERICAN NATIONAL STANDARDS INSTITUTE (1996). *Structural Standards for Steel Antenna Towers and Antenna Supporting Structures*. ANSI/TIA/EIA-222-F Standard, Published by Telecommunications Industry Association, Arlington, VA. 115 p.

AMERICAN NATIONAL STANDARDS INSTITUTE (1980). *Development of a probability based load criterion for American National Standard A58*.

AMERICAN SOCIETY OF CIVIL ENGINEERS (2000). ASCE 10-97 *Design of Latticed Steel Transmission Structures*, Reston, Virginia.71 p.

BARKER, R.M. et al. (1991). *Manual For The Design Of Bridge Foundations*. National Cooperative Highway Research Program, Report 343, Transportation Research Board, Washington, D.C., 308 p.

BEAULIEU, PICARD et al. (2003). *Calcul des charpentes d'acier – Tome 1,* 1e édition, Institut canadien de la construction en acier, Toronto, Ontario, Canada, 794 p.

BECKER, D.E. (1996, (a)). *Eighteenth Canadian Geotechnical Colluquium: Limit States Design For Foundations, Part I,* An overview of the foundations design process. Canadian Geotechnical Journal, 33: 956-983.

BECKER, D.E. (1996, (b)). *Eighteenth Canadian Geotechnical Colloquium: Limit States Design For Foundations, Part 2*, Development for the National Building Code of Canada. Canadian Geotechnical Journal, 33: 984-1007.

BEEN, K., CLARK, J.I., and LIVINGSTON, W.R., (1993). *Verification and calibration studies for the new CAN/CSA-S472 foundations of offshore structures,* Canadian Geotechnical Journal., 30, No.3, page 515-525

BJERRUM, L. (1973). *Problem of soil mechanics and construction on soft clays,* Proc. 8th Int. Conf. on soil Mechanics and Foundations Engineering, Moscow, State of the Art Report, Vol. 3, pp. 111-159.

BOWLES, J.E. (1988). *Foundation Analysis and Design*, 4th Edition, McGraw-Hill, 1175 p.

BOWLES, J.E. (1996). *Foundation Analysis and Design*, 5th Edition, McGraw-Hill, 1175 p.

BRAHAMA, Chandra S. (1992). Computer-aided Design of Shallow Foundation, International Conference on Geotechnical Engineering, Johor Bahru, Malaysia.

BRINCH HANSEN, J. (1970). *A Revised and Extended Formula for Bearing Capacity*, Danish Geotechnical Institute, Copenhagen, Bul. No. 28, 21 pp.

BRINCH HANSEN, J. (1956). *Limit Design and Safety Factors in Soil Mechanics,* The Danish Technical Press, Copenhagen, Bulletin No. 1.

BRINCH HANSEN, J. (1953). *Earth pressure calculations,* The Danish Technical Press, Copenhagen.

CANADIAN GEOTECHNICAL SOCIETY (1992). *Canadian Foundation Engineering Manual,* 3rd Ed., BiTech Publishers, Vancouver, British Columbia.

CANADIAN GEOTECHNICAL SOCIETY (1985). *Canadian Foundation Engineering Manual*, 2nd Ed., BiTech Publishers, Vancouver, British Columbia.

CANADIAN INSTITUTE OF STEEL CONSTRUCTION (2004), *Handbook of Steel Construction*, (CSA S16-01 Standard), 8th Ed., Toronto, Ontario, Canada.

CANADIAN PORTLAND CEMENT ASSOCIATION (1998). *Concrete Design Handbook* (CSA A23.3-94 Standard), 2nd Ed., Ottawa, Ontario, Canada.

CANADIAN STANDARDS ASSOCIATION (1992b). *Foundations, CSA*, Rexdale, Ont. Publication No. CAN/CSA-S472-92.

CANADIAN STANDARDS ASSOCIATION (2001). *Antennas, Towers, and Antenna-Supporting Structures* S37-01, Toronto, Ont. Canada. 118 p.

CANADIAN STANDARDS ASSOCIATION (2001). *Calcul aux états limites des charpentes en bois.* Norme nationale du Canada CAN/CSA-O86.1, Canada.

CONSEIL CANADIEN DU BOIS (2001). *Manuel de calcul des charpentes en bois,* Ottawa, Ontario, Canada.

CONSEIL CANADIEN DU BOIS (1996). *Introduction au calcul des charpentes en bois – Guide d'étude complémentaire au Manuel de calcul des charpentes en bois,* Canada.

CSA INTERNATIONAL (2000). *Code canadien sur le calcul des ponts routiers,* Norme nationale du Canada CAN/CSA-S6-00, Toronto, Ont. Canada.

CHIEN, RITCHIE (1984). *Design and Construction of Composite Floor Systems*, Canadian Institute of Steel Construction, Markham, Ontario, 323 pp.

CHRISTIAN, J. T., Carrier,W. D. et al. (1978). *Janbu, Bjerrum and Kjaernsli's chart reinterpreted*, Canadian Geotechnical Journal, Vol. 15, pp. 123-128.

EUROPEAN COMMITEE FOR STANDARDIZATION (CEN), (1993). *Geotechnical Design, General Rules*, Eurocode. Danish Geotechnical Institute, Copenhagen.

CONCRETE REINFORCING STEEL INSTITUTE (1997*). Placing Reinforcing Bars – Recommended Practices*, 7th Edition.

DEPARTEMENT OF THE NAVY (1982). *Foundation and Earth Structures - Design Manual 7.2*, NAVFAC DM-7.2.

DS 415 (1984). *Danish Code of Practice for Foundation Engineering*, 3rd Edition, Danish Technical Press, Copenhagen.

EUROPEAN COMMITEE FOR STANDARDIZATION, (1994). *ENV 1997-1: Eurocode 7, Geotechnical Design, part 1 General Rules.*

FAVRE, R., JACCOUD, J.-P., BURD, O. ET CHARIF, H. (1997). *Dimensionnement des structures en béton – Aptitude au service et éléments de structure*, Traité de Génie Civil de l'École polytechnique fédérale de Lauzanne – Volume 8, Presses polytechniques et universitaires romandes, Lauzanne, 591 p.

FELLENIUS, Bengt H. (1994). *Limit States Design for Deep Foundations*, Proceedings of the International conference on Design and Construction of Deep Foundations, Volume II, Sessions 1 through 4: 415-426.

FOX, Steven R., SCHUSTER, Reinhold M. (1996). *Design in Cold Formed Steel – Based on CSA-S136-94 – A One Day Seminar*, Presented by Canadian Sheet Steel Building Institute and the Department of Civil Engineering, University of Waterloo.

GALAMBOS, Theodore V. (1998). *Guide to Stability Design criteria for Metal Structures*, 5th Edition, Wiley, 911 pp.

HOFFMAN, Edward S., GUSTAFSON, David P., GOUWENS, Albert J. (1998). *Structural Design Guide to the ACI Building Code*, 4th Edition, Kluwer Academic Publishers, 466 pp.

HUMAR, J.L. (1990). *Dynamics of Structures*, Prentice-Hall, Englewood Cliffs, New Jersey, 780 pp.

INSTITUT D'ACIER D'ARMATURE DU CANADA (1996), *Acier d'armature – Manuel des normes recommandées*, 3e édition, Montréal, Québec, 96 pp.

LABONTÉ, Laurent (1982). *Dessin de charpente en béton*, Modulo éditeur, Outremont, Québec, 440 pp.

LIAO, S.S.C., and WHITMAN, R.V. (1986). *Overburden correction factors for the SPT in sand*, ASCE Journal Geot. Division Vol. 112, No. 3, pp. 373-377.

LIN, Yude. (1996). *Limit State Design with Load Dependence in Offshore Foundations*, Proceedings of the Sixth - International Offshore and Polar Engineering Conference (1996): 562-569.

MEYERHOF, G. G. (1953). *The Bearing Capacity of Foundation under Eccentric and Inclined Loads*, 3rd Proceedings of International Conference on Soil Mechanics and Foundation Engineering, vol. 1, pp. 440-445.

MEYERHOF, G. G. (1982). *Soil-Structure Interaction and Foundations, State-of-the Art Report*, Sixth Pan-American Conference on Soil Mechanics and Foundation Engineering, Vol.1, pp. 109-139. Paper no. 100 in Tech-Press, Technical University of Nova Scotia, Halifax, N.S.

MEYERHOF, G. G. (1995). *Development of Geotechnical Limit State Design*, Canadian Geotechnical Journal, 32: 128-136.

MINISTRY OF TRANSPORTATION OF ONTARIO, MTO, (1991). *Ontario Highway Bridge Design Code and Commentary (two volumes)*, MTO, Quality and Standards Division, Toronto, Ontario, 713 p.

NAWY, E. G., (2000). *Prestressed Concrete – A Fundamental Approach*, 3rd Ed., Prentice Hall International series in civil engineering and engineering mechanics, New Jersey, 938 p.

NBC (1995). *National building Code of Canada*, National Research Council of Canada, Ottawa.

OVENSEN, N.K., and ORR, T. (1991). *Limit States Design-The European Perspective, Proceedings of Geotechnical Engineering Congress*, American Society of Civil Engineers, Special Publication, No. 27, Vol. II,pp.1341-1352.

PECK, R.B., HANSON, W.E., and THORNBURN, T.H. (1974). *Foundation Engineering*, J. Wiley & sons, 514 pp.

PICARD, A. (1983). *Béton Précontraint Tome 1 "Principes fondamentaux et dimensionnement"*, Gaëtan Morin Éditeur, Chicoutimi, Québec, Canada, 355p.

PICARD, BEAULIEU (1991). *Calcul des charpentes d'acier*, Institut Canadien de la Construction en Acier, Canada, 862p.

POPOV, E.P. (1978). *SI Version – Mechanics of Materials*, 2nd Edition, Prentice-Hall, Englewood Cliffs, New Jersey, 590 pp.

SKEMPTON, A.W. (1951). *The Bearing Capacity of Clays*, Proc. Building Research Congress, vol. 1, pp.180-189.

TAYLOR, D.W. (1948). *Fundamentals of Soil Mechanics*, J. Wiley & Sons, New York, 700 pp.

TERZAGHI & PECK. (1967). *Soil Mechanics in Engineering Practice*, 2nd ed., John Wiley & Sons, New York, 729 pp.

TERZAGHI, K. (1943). *Theoretical Soil Mechanics*, John Wiley & Sons, New York, 510 pp.

TIMOSHENKO, S., WOINOWSKY-KRIEGER, S. (1959). *Theory of Plates and Shells*, 2^{nd} Edition, Engineering Societies Monographs Committee and Mc Graw-Hill Book Company, 580 pp.

U.S. ARMY CORPS OF ENGINEERS (1994) . *Bearing Capacity of Soils*, Adapted from the Technical Engineering and Design Guide no.7, published by the American Society of Civil Engineers, 142 pp.

VESIC, A. S. (1973). *Analysis of Ultimate Loads of Shallow Foundations*, Journal of Soil Mechanics and Foundation Division, American Society of Civil Engineering Division, vol. 96, SM 2, March, pp. 561-584.

WALTER, R., MIEHLBRADT, M. (1990). *Dimensionnement des structures en béton – Bases et technologies*, Traité de Génie Civil de l'École polytechnique fédérale de Lauzanne – Volume 7, Presses polytechniques et universitaires romandes, Lauzanne, 388 p.

YU, Wei-Wen (1991). *Cold-Formed Steel Design*, 2^{nd} Edition, A Wiley-Interscience Publication, John Wiley & Sons, Inc., 631 pp.