Volume

2

CIVILDESIGN INC.

VisualDesign Software– Version 5.9

April 2006

Complete User's Manual

Disclaimer

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

DYNAMIC ANALYSIS

TABLE OF CONTENTS

Chapter 7 Dynamic Analysis

General

Dynamic Analysis Module

The Dynamic analysis module includes the Modal, Spectral, Linear Time History, Non-Linear Time History analyses and the General Dynamic analysis.

See also

Distinction between Modal, Spectral and Time History analysis

Seismic Analysis Modeling to Satisfy Building Codes

Effects of earthquakes according to the CNBC 1995

Calibration of Spectral, Linear and Non-linear Time History Analysis

Participating Mass

CQC Method for Modal Combination

Distinction between Modal, Spectral, and Time History Analysis

A Modal analysis involves computation of the structure's natural frequencies and the corresponding mode shapes.

According to modes shapes obtained from the Modal analysis, the Spectral analysis will compute, for each seismic direction and each mode shape, the maximum forces and displacements by using a spectrum and scaling factors given by the code.

The Time History analysis computes displacements in time.

See also

Static Analysis

Modal Analysis

Spectral Analysis

Time History Analysis

Effects of Earthquakes According to CNBC 1995

Excerpts from: CANADIAN NATIONAL BUILDING CODE (1995), User's Guide – Structural Commentaries (Part 4), Commentary J – Effects of Earthquakes

Direction of Earthquake Motions

"For normal buildings […] independent design about each of the horizontal axes together with the associated torsional forces id considered to provide adequate resistance against earthquake motions applied in any direction. This simplification forms the basis for the earthquake requirements in the NBC. Particular attention should be paid, however, to the effect of the combined stresses at the external and re-entrant corners, which are especially vulnerable to the effect of concurrent translational and torsional motions."

Vertical accelerations

[…] In certain special structures, these accelerations may lead to instability or unusual reductions in the factors of safety. Cantilevered structures or cantilevered building components are also sensitive to vertical accelerations. When this becomes a governing design consideration, dynamic analysis should be employed."

Structural Response to Ground Motion

"The elastic response of a single-degree-of-freedom system to ground motion depends on the fundamental period and the damping characteristics of the system, and on the frequency content and amplitude of the ground motion. The base shear, which can be used as a measure of this response, is expressed as the product of the mass of the system and the spectral acceleration as given by the response spectrum.

[…] For usual buildings of low or moderate height, the principal earthquake response is due to the fundamental mode of vibration. For taller structures, some allowance for contributions of the higher modes is made in the base shear calculations in most building codes, including the NBC."

Force Modification Factor, R

[…] In choosing the structural system for a building, large dissimilarities in the stiffness and ductility characteristics of framing systems in the orthogonal directions should be avoided."

Foundation Factor, F

[…]The seismic design procedures outlined in the NBC are based on the assumption that the structures are founded on a rigid base moving with the ground surface motion. […] For most buildings covered by the NBC, neglecting soil-structure interaction will give a conservative design."

Lateral Force Distribution

"For translational vibrations, the Code formula assumes that the building response is primarily due to its fundamental mode and the mode shape is assumed to be linear."

Dynamic Analysis

"[NBC] code states that the distribution of forces in the building may also be determined by a dynamic analysis. This would apply especially to buildings with significant irregularities either in plan or in elevation, and buildings with setbacks or major discontinuities in stiffness or mass. Performing a dynamic analysis will lead to a better representation of modal contribution in tall buildings."

[…]The dynamic analysis in [the NBC] is based on the linear spectral modal technique. If the results of such calculations yield a lower base shear, they have to be calibrated to the static base shear as required [by the Code]. The reason is that Code values represent the minimum values commensurate with an acceptable level of public safety."

Modal Combination

[…] The square-root-sum-square (SRSS) combination rule may be used when the periods are well separated. For buildings having closely spaced periods, a more refined rule such as the complete quadratic combination (CQC) rule should be used. Enough modes should be included that at least 90% of the participating mass of the structure is included in the calculation of response for each principal horizontal direction."

[…] When a two dimensional mathematical model is used, torsional effects need to be considered statically […] and combined with the design parameters of interest […] to arrive at the final design values.

When a three dimensional mathematical model is used, the accidental torsional effects need to be considered statically […] and combined with the design parameters […] to arrive at the final design values."

[…] For cases where more detailed knowledge of the building motion or forces near the ultimate limit state is required, a non-linear step-by-step approach is indicated, with consideration of a number of appropriately selected ground motions."

Overturning Moments

[…] A more refined method of accounting for the maximum overturning moments is through dynamic analysis."

Torsional Moments

"The inertia forces induced in the structure by earthquake ground motions act through the centre of gravity of the masses. If the centre of mass and the centre of rigidity do not coincide because of asymmetrical arrangement of structural elements or uneven mass distributions, torsional moments will arise."

[…] Accidental torsion moments are intended to account for the possible additional torsion arising from variations in the estimates of dead and live loads at the floor levels, addition of wall panels and partitions after completion of the building, variation of the stiffness with time, and inelastic or plastic action. The effects of possible torsional motion of the ground should also be considered."

[…] In buildings with complete diaphragms, such as complete reinforced concrete floor slabs, all elements interconnected by such members can be counted on to resist torsional forces."

In core-type buildings, where all stiffening elements are located in a central core away from the periphery, accidental torsion and torsional ground motion are particularly significant."

Setbacks

"A setback is a sudden change in plan dimension or a sudden change in stiffness along the height of a building. […] The effects of major changes in stiffness or geometry are best investigated by dynamic methods."

Deflection

"Deflection refers to the lateral deflection at any point in the structure relative to the ground. Incremental deflection or Interstory deflection refers to the lateral deflection of a story relative to the one just below it. The calculations of deflections are intended to be based on accepted practice and should include such items as P-Delta effects, foundation rotations, and the effects of cracked concrete sections, when these have an important effect on the structure."

P- Delta Effects

"When a flexible building is subjected to lateral seismic forces, the gravity loads acting through the lateral displacements lead to additional moments throughout the structure. These additional P-Delta moments reduce the capacity of the structure to resist story shears."

[…] The procedure recommended to allow for P-Delta effects is equivalent to proportioning the structure at each level x to resist an increased seismic shear force. […] With the seismic shear capacities at each story increased to allow for P-Delta effects, the ability of the strengthened structure to absorb inelastic energy during an earthquake is also increased."

Special Provisions

"Unreinforced masonry buildings have fared badly when subjected to earthquakes. The presence of reinforcing embedded in mortar or grout increases ductility and reduces the likelihood of brittle failure."

[…] Floor systems that act as diaphragms should be studied to ensure that they are capable of distributing the loads to the various elements."

[…] Special mechanical protection systems such as base isolation or controlled friction damping devices can significantly alter the seismic response of buildings."

Seismic Analysis Modeling to Satisfy Building Codes

Excerpts from:

Dr Edward L. Wilson, *Articles on Numerical Techniques used in SAP2000, ETABS and SAFE*, Chap. 12, 15 and 17. University of California, Berkeley.

Introduction

[…] The major advantage of using the forces obtained from a dynamic analysis as the basis for a structural design is that the vertical distribution of forces may be significantly different from the forces obtained from an equivalent static load analysis. Consequently, the use of dynamic analysis will produce structural designs that are more earthquake resistant than structures designed using static loads.

"For many years, approximate two-dimensional static load analysis is acceptable as the basis for seismic design in many geographical areas and for most types of structural systems. During the past twenty years, due to the increasing availability of modern digital computers, most engineers have experience with the static load analysis of three-dimensional structures. However, few engineers […] have had experience with the three-dimensional dynamic response analysis. Therefore, the interpretation of the dynamic analysis requirement […] represents a new challenge to most structural engineers.

[…] codes allow the results obtained from a dynamic analysis to be normalized so that the maximum dynamic base shear is equal to the base shear obtained from a simple two-dimensional static load analysis. Most members of the profession realize that there is no theoretical foundation for this approach. However, for selecting the magnitude of the dynamic loading that will satisfy codes requirements, this approach can be accepted, in a modified form, until a more rational method is adopted.

"The calculation of the design base shear is simple and the variables are defined in codes. It is of interest to note, however, that the basic magnitude of the seismic loads has not changed significantly from previous codes. The major change is that dynamic methods of analysis must be used in the principal directions of the structure. The codes do not state how to define the principal directions for a three dimensional structure of arbitrary geometric shape. Since the design base shear can be different in each direction, the scaled spectra approach can produce a different input motion for each direction, for both regular and irregular structures. Therefore, the codes dynamic analysis approach can result in a structural design that is relatively weak in one direction. The method of dynamic analysis proposed in this chapter results in a structural design that has equal resistance in all directions.

"In addition, the maximum possible design base shear […] is approximately 35 percent of the mass of the structure. For many structures, it is less than 10 percent. It is generally recognized that this force level is small when compared to measured earthquake forces. Therefore, the use of this design base shear requires that a substantial ductility be designed into the structure.

"The definition of an irregular structure, the scaling of the dynamic base shears to the static base shears for each direction, the application of accidental torsional loads and the treatment of orthogonal loading effects are areas which are not clearly defined in building codes. […] the method of three-dimensional seismic analysis [that will be presented] is a method based on the response spectral shapes defined in codes and previously published and accepted computational procedures […] and will satisfy the Lateral Force Requirements of building codes.

Three Dimensional Computer Model

"Real and accidental torsional effects must be considered for all structures. Therefore, all structures must be treated as three-dimensional systems. Structures with irregular plans, vertical setbacks or soft stories will cause no additional problems if a realistic three-dimensional computer model is created. This model should be developed in the very early stages of design since it can be used for static wind and vertical loads, as well as dynamic seismic loads.

"Only structural elements with significant stiffness and ductility should be modeled. […] The rigid, in-plane approximation of floor systems has been shown to be acceptable for most buildings. For the purpose of elastic dynamic analysis, gross concrete sections, neglecting the stiffness of the steel, are normally used. A cracked section mode should be used to check the final design.

"The P-delta effects should be included in all structural models. […] The effects of including P-Delta displacements in a dynamic analysis results in a small increase in the period of all modes. In addition to being more accurate, an additional advantage of automatically including P-Delta effects is that the moment magnification factor for all members can be taken as unity in all subsequent stress checks.

"The mass of the structure can be estimated with a high degree of accuracy. The major assumption required is to estimate the amount of live load to be included as added mass. For certain types of structures, it may be necessary to conduct several analyses with different values for mass. The lumped mass approximation has proven to be accurate. In the case of the rigid diaphragm approximation, the rotational mass moment of inertia must be calculated.

"The stiffness of the foundation region of most structures can be modeled by massless structural elements. It is particularly important to model the stiffness of piles and the rotational stiffness at the base of shear walls.

"The computer model for static loads only should be executed prior to conducting a dynamic analysis. Equilibrium can be checked and various modeling approximations can be verified with simple static load patterns. The results of a dynamic analysis are generally very complex and the forces obtained from a response spectra analysis are always positive. Therefore, dynamic equilibrium is almost impossible to check. However, it is relatively simple to check energy balances in both linear and non-linear analysis.

Three Dimensional Mode Shapes and Frequencies

"The first step in the dynamic analysis of a structural model is the calculation of the three dimensional mode shapes and natural frequencies of vibration. [For example, consider an] irregular, eight story concrete building with several hundred degrees-of-freedom. […] the three components of mass are lumped at each of the eight floor levels. Therefore, only 24 three-dimensional shapes are possible.

"Each three dimensional mode shape of a structure may have displacement components in all directions. For the special case of a symmetrical structure, the mode shapes are uncoupled and will have displacement in one direction only. Since each mode can be considered a deflection due to a set of static loads, six base reaction forces can be calculated for each mode shape. […] The magnitude of the forces and moments has no meaning since the amplitude of a mode shape can be normalized to any value. However, the relative values of the different components of the shears and moments associated with each mode are of considerable value.

"The [codes] define an irregular structure as one which has a certain geometric shape or in which stiffness and mass discontinuities exist. A far more rational definition is that a regular structure is one in which there is a minimum coupling between the lateral displacements and the torsional rotations for the mode shapes associated with the lower frequencies of the system. Therefore, if the model is modified and tuned by studying the three dimensional mode shapes during the preliminary design phase, it may be possible to convert a geometrically irregular structure to a dynamically regular structure from an earthquake-resistant design standpoint.

[…] it is of interest to note that the mode shapes, which tend to have directions that are 90 degrees apart, have almost the same value for their period. This is typical of three dimensional mode shapes for both regular and irregular buildings. For regular symmetric structures, which have equal stiffness in all directions, the periods associated with the lateral displacements will results in pairs of identical periods. However, the directions associated with the pair of three dimensional mode shapes are not mathematically unique.

[…] For a response spectrum analysis, [the codes] state that at least 90 percent of the participating mass of the structure must be included in the calculation of response for each principal direction. Therefore, the number of modes to be evaluated must satisfy this requirement."

Three Dimensional Dynamic Analysis

"It is possible to conduct a dynamic, time-history, response analysis by either the mode superposition or step-by-step methods of analysis. However, a standard time-history ground motion, for the purpose of design, has not been defined. Therefore, most engineers use the response spectrum method of analysis as the basic approach. The engineer may choose many types of spectra defined in the codes and scale them with the acceleration factor of the zone. (The El Centro spectrum is always used.) Furthermore, the engineer must not forget that an accelerogram changes according to the layers of soil underneath and the depth of the foundation. Each site is different so we can only approximate the response spectrum. […] The first step in a response spectrum analysis is the calculation of the three dimensional mode shapes and frequencies of the structure.

Definition of Principal Directions

"A weakness [in the codes] is the lack of definition of the principal horizontal directions for a general three dimensional structure. If each engineer is allowed to select an arbitrary reference system, the dynamic base shear will not be unique and each reference system could result in a different design. One solution to this problem, that will result in a unique design base shear, is to use the direction of the base shear associated with the fundamental mode of vibration as the definition of the major principal direction for the structure. The minor principal direction will be, by definition, ninety degrees from the major axis."

Directional and Orthogonal Effects

"The required design seismic forces may come from any horizontal direction and, for the purpose of design, they may be assumed to act non-concurrently in the direction of each principal axis of the structure. In addition, for the purpose of member design, the effects of seismic loading in two orthogonal directions may be combined using the SRSS (Square-Root-of-the-Sum-of-Squares) method or the CQC (Complete Quadratic Combination) method."

Basic Method of Seismic Analysis

"In order to satisfy the [codes] requirements, it is necessary to conduct two separate spectrum analyses in the major and minor principal directions. […]The spectra used in both of these analyses can be obtained directly from the codes."

Scaling of Results

"Each of these analyses will produce a base shear in the major principal direction. A single value for the dynamic base shear is calculated by the SRSS or CQC method. Also, a dynamic base shear can be calculated in the minor principal direction. The next step is to scale the previously used spectra shapes by the ratio of design base shear to the minimum value of the dynamic base shear. This approach is more conservative than proposed by the current requirements, since only the scaling factor that produces the largest response is used. However, this approach is far more rational since it results in the same design earthquake in all directions."

Dynamic Displacements and Member Forces

"The displacement and force distribution are calculated using the basic SRSS method or CQC method to combine the results from 100 percent of the scaled spectra applied in each direction. If two analyses are conducted in any two orthogonal directions, in which the CQC [(Complete Quadratic Combination)] method is used to combine the modal maximums for each analysis, and the results are combined by the SRSS method, exactly the same results will be obtained regardless of the orientation of the orthogonal reference system. Therefore, the direction of the base shear of the first mode defines a reference system for the building.

If site-specific spectra are given, for which scaling is not required, any orthogonal reference system can be used. In either case, only one computer run is necessary to calculate all member forces to be used for design."

Torsional Effects

"Possible torsional ground motion, the unpredictable distribution of live load mass and the variations of structural properties are three reasons why both regular and irregular structures must be designed for accidental torsional loads. Also, for regular structure, lateral loads do not excite torsional modes.

[…] The codes allow the use of pure static torsional loads to predict the additional design forces caused by accidental torsion. The basic vertical distribution of lateral static loads is given by the codes equations. The static torsional moment at any level is calculated by the multiplication of the static load at that level by 5 percent of the maximum dimension at that level. […] it is recommended that these pure torsional static loads, applied at the centre of mass at each level, be used as the basic approach to account for accidental torsional loads. This static torsional load is treated as a separate load condition so that it can be appropriately combined with the other static and dynamic loads."

Summary of Dynamic Analysis Method

1.   […] a dynamic analysis method is summarized that produces unique design displacements and member forces which will satisfy the codes. It can be used for both regular and irregular structures. The major steps in the approach are as follows:

2.   A three-dimensional computer model must be created in which all significant structural elements are modeled. This model should be used in the early phases of design since it can be used for both static and dynamic loads.

3.   The three-dimensional mode shapes should be repeatedly evaluated during the design of the structure. The directional and torsional properties of the mode shapes can be used to improve the design. A well-designed structure should have a minimum amount of torsion in the mode shapes associated with the lower frequencies of the structure.

4.   The direction of the base reaction of the mode shape associated with the fundamental frequency of the system is used to define the principal directions of the three dimensional structure.

5.   In VisualDesign, either the CQC or the SRSS methods can be used to calculate the dynamic base shear in each principal direction due to 100 percent of the normalized spectra shapes. Use the minimum value of the base shear in the principal directions to produce one scaled design spectra. The design base shear is based on the longest period obtained from the computer model.

6.   The dynamic displacements and member forces are calculated using the SRSS value of 100 percent of the called design spectra applied non-concurrently in any two orthogonal directions.

7.   A pure torsion static load condition is produced using the suggested vertical lateral load distribution defined in the codes.

The member design forces are calculated using the following load combination rule:

$$F_{DESIGN} = F_{DEAD\ LOAD} \pm [F_{DYNAMIC} + |F_{TORSION}] + F_{OTHER}$$

The dynamic forces are always positive and the accidental torsional forces must always increase the value of force. If vertical dynamic loads are to be considered, a dead load factor can be applied.

Calibration of Analysis in Seismic Design

Analysis	Code	Calibrated Forces	Displacements (1)	Forces not calibrated	Displacements (1)
Spectral	CNB 1995	V/Vdyn	R*V/Vdyn	I*F*U/R	I*F*U
	UBC 1994	V/Vdyn	R*V/Vdyn	I*S/R	I*S
	UBC 1997	V/Vdyn	R*V/Vdyn	I/R	I
	CAN-S6-00	N/A	N/A	I*S/R	I*S
Linear Time History	CNB 1995	V/Vdyn	R*V/Vdyn	I*F*U/R	I*F*U
	UBC 1994	V/Vdyn	R*V/Vdyn	I*S/R	I*S
	UBC 1997	V/Vdyn	R*V/Vdyn	I/R	I
	CAN-S6-00	N/A	N/A	I*S/R	I*S
Non-linear Time History	CNB 1995	I*F	I*F	I*F	I*F
	UBC 1994	I*S	I*S	I*S	I*S
	UBC 1997	I	I	I	I
	CAN-S6-00	I*S	I*S	I*S	I*S

Where:

V: Minimum base shear according to code
Vdyn: Base shear calculated in the spectral analysis, with no calibration.

And according to Canadian and American Building Codes:

I: Importance factor;
R: Force Modification factor (global ductility of the structure or components);
F and S: Foundation or Site factor;
U: Calibration factor.

N.B. There is no calibration for non-linear analysis. Modification factor R and factor U must not have any effect on non-linear time history analysis. If you wish to factor non-linear time history analysis, enter a load factor in the **Load Combinations Definition** spreadsheet.

Note (1): VisualDesign never calibrates displacements.

See also
Static Analysis
Modal Analysis
Spectral Analysis
Project Configuration
Overall Ductility of a Structure: Force Modification Factor R

Inclusion of Non-Linear Effects into Analysis

The user has the possibility to use the stiffness matrix [K] that includes P-Delta effects. This approach allows more realistic and accurate results for structures having a non-linear behaviour such as long spans bridges with great compression in the deck (suspension bridges, guyed bridges…), multi-storey buildings, transmission lines towers with guys or without guys, and all structures with guys (pre-stressed or not).

In general, if non-linear effects are included in the analysis, results will be more accurate.

When a Moving Load analysis or a Modal analysis is run without including non-linear effects, the stiffness matrix [K] is calculated in a linear manner, which excludes P-delta effects. This method is acceptable as long as non-linear effects are not important.

This option may be activated in the **Modal Analysis** dialog box and in the **Moving Loads Analysi**s dialog box for a guyed bridge or a 40-storey building, where non-linear effects are important.

Modal Analysis

By activating the option "Include non-linear effects", the analysis will use the "Mass" load combination to calculate initial non-linear effects. The stiffness matrix that is calculated includes P-Delta effects. The calculation is done before the eigenvalues and eigenvectors are calculated.

Spectral and Time History Analysis

No option is needed. The program uses parameters that had been set for the modal calculation. It will use eigenvalues and eigenvectors that include non-linear effects. Internal stresses for each modal contribution are calculated including P-Delta effects. Accidental torsion effects are considered and P-Δ and P-δ non-linear effects are included. It is the same with the effects due to theta x.

Moving Load Analysis

By activating the option "Include non-linear effects", the analysis will use the " Mass " load combination to calculate initial non-linear effects. (The user must create this load combination to run a modal analysis.) The stiffness matrix is calculated using the non-linear P-Delta effects before the calculation of influence lines is done.

See also
Modal Analysis
Moving Loads Analysis
Inelastic Effects due to an Earthquake

Accidental Torsion Effects

If you wish to add accidental torsion effects in the **Time History** analysis, you must execute the spectral analysis beforehand, because the spectral analysis generates static load cases that cause accidental torsion effects. In addition, even if you do not want to add accidental torsion effects, it is preferable to proceed with a spectral analysis for, to at least, evaluate the percentage of participating weight. This percentage (minimum 90%) is a good indicator for the minimum number of modes to use.

The **Time History** analysis can be relatively long (longer than **Spectral** analysis), so it is important to master the Spectral analysis of the project before doing a Time History analysis.

Note If accidental torsion effects or inelastic effects are required, the analysis function will not be activated unless you perform a spectral analysis.

See also
Loads Definition spreadsheet
Time History Analysis
Spectral Analysis
Accelerograms

Tension-only Bracings

- Go to the **Member** tab.

- Select a tension-only end condition, Axial Fz (<-[ ]->), and select the usage "[Xt] Diagonal" in the Usage list box.

- Increase the number of iterations (up to 10) in the **Analysis** tab (**Project Configuration**).

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.

Convergence problem

In some cases, convergence cannot be reached during a steel design with tension-only bracings. Lateral loads or gravity loads applied directly at the top of braced bays can induce compression in the bracings. The compression causes instability in the structure when VisualDesign remove the compressed elements (bracings) during the release process. In fact, the structure looses the bracings during the process.

To resolve this problem, do as follows:

- In the **Member** tab: Put back tension/compression axial end conditions and assign a "Standard" usage to bracings. Specify a value of 0.5 as effective stiffness for bending, axial and shear.

- In the **Steel Design** tab: Disable the automatic calculation of effective compression lengths (Kx and Ky) and put a 0 value everywhere.

- Launch the design again.

Guys (Tower design)

Axial end conditions are different for guys. Please refer to Modeling and analysing a guyed structure.

See also
Member End Conditions

Project Configuration

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 - CNBC 2005

This table describes the fields included in this tab when the Canadian National Building Code 2005 is selected in the "Building Code" list box.

Parameters	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.
Location Category	Location category (A, B, C, D, E, or F), as specified at paragraph 4.1.8.4.
Total height, hn	Total height of the structure from the base to level N, where the base of the structure is the level at which horizontal earthquake motions are considered to be imparted to the structure.
Number of stories, N	Total number of stories above exterior grade up to level N.

Parameters	Definition
Importance factor, Ie	Seismic importance factor of the structure, as specified in Table 4.1.8.5.
Acceleration factor, Fa	Acceleration factor depending on the location, as specified at paragraph 4.1.8.4. 4).
Velocity factor, Fv	Velocity factor depending on the location, as specified at paragraph 4.1.8.4. 4).
Ie Fa Sa (0.2)	Product of these parameters.

Spectral Accelerations (g)

City	Click on the button and select a city among the drop down list box. Parameters Sa, Fa, Fv, and Sa (0.2) will be automatically initialized.
Sa (0.2)	Spectral acceleration of the response considering a damping of 5% relatively to "g" during a period of 0.2 sec.
Sa (0.5)	Spectral acceleration of the response considering a damping of 5% relatively to "g" during a period of 0.5 sec.
Sa (1.0)	Spectral acceleration of the response considering a damping of 5% relatively to "g" during a period of 1.0 sec.
Sa (2.0)	Spectral acceleration of the response considering a damping of 5% relatively to "g" during a period of 2.0 sec.

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.
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).
SFRS oriented towards orthogonal axes	Activate this option if the building Seismic Force Resisting Systems act towards orthogonal axes.

Parameters	Definition
Regular structure	Activate this box if the structure has a regular geometry and inertia.
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 for applying the accelerogram. 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". Read 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 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**.

Seismic tab - CNBC 95

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.

Parameters	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

Parameters	Definition
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.
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.

Heading	Definition
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**.

See also

Seismic tab - CSA-S6-00

This table describes the fields included in this tab when the *CSA-S6-00* has been selected in the "Building Code" list box.

Parameters	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

Parameters	Definition
Zonal acceleration ratio, A	Acceleration ratio for this zone, as indicated at article 4.4.3).
Importance factor, I	Seismic importance factor of the structure.
Type of soil profile	Select a type of soil profile in the drop-down list box.
Site factor, S	Corresponding site factor.

Spectral Analysis

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.

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.

Parameters	Definition
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**.

Seismic tab - UBC 94

This table describes the fields included in this tab when the *UBC 94* has been selected in the "Building Code" list box.

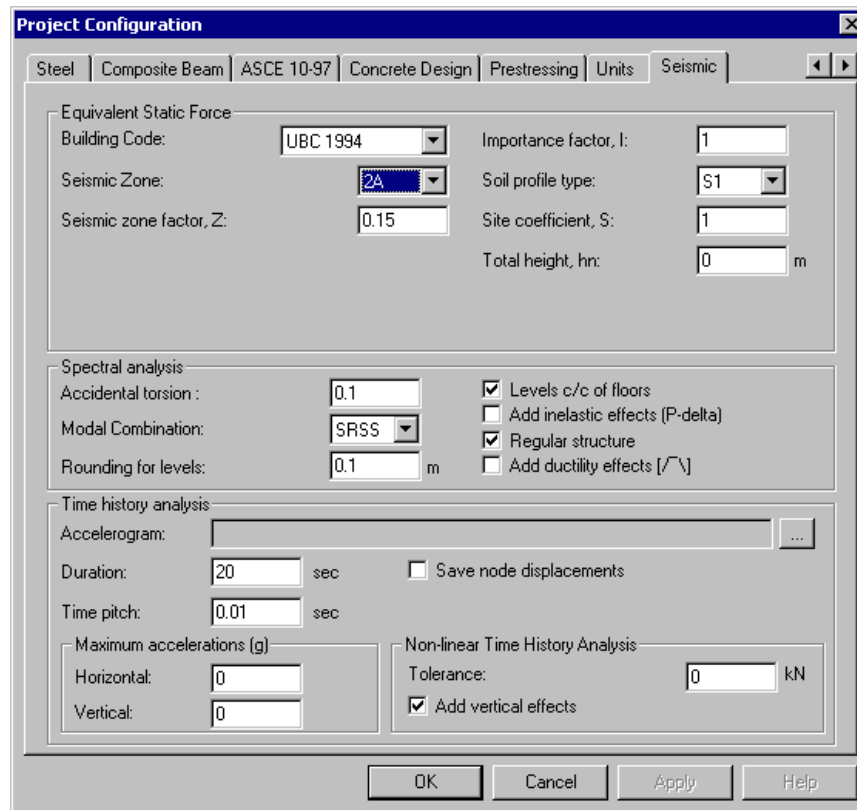| Parameters	Definition
Equivalent Static Force	
Building Code	Each building code uses a normalized spectrum according to the occurrence probability of each country. The selection of a building code automatically fixes the spectrum to be used.
Seismic zone	Acceleration-related seismic zone, as described at clause 1627.2 and figure 16-2, p. 2-41.
Seismic zone factor, Z	Seismic zone factor calculated from the acceleration-related seismic zone, as described at clause 1627.2 and Table 16-I, p. 2-34.
Importance factor, I	Seismic importance factor of the structure. Refer to Table 16-K.
Type of Soil Profile	Select a type of soil profile among the list box. Refer to Table 16-J.
Site coefficient, S	Site coefficient dependent upon the soil profile.
Total height, hn	Total height of the structure from the base, where the base of the structure is the level at which horizontal earthquake motions are considered to be imparted to the structure.
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.
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). Inelastic effects are calculated according to article 4.1.9.1. 28) of NBC 1995.
Regular structure	Check this option if the building has a regular geometry.

Parameters	Definition
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**.

See also

Static Analysis

Dynamic Analysis

Project Configuration

Overall Ductility of a Structure: Force Modification Factor R

Calibration of Spectral, linear and non-linear Time History analysis

Seismic tab - UBC 97
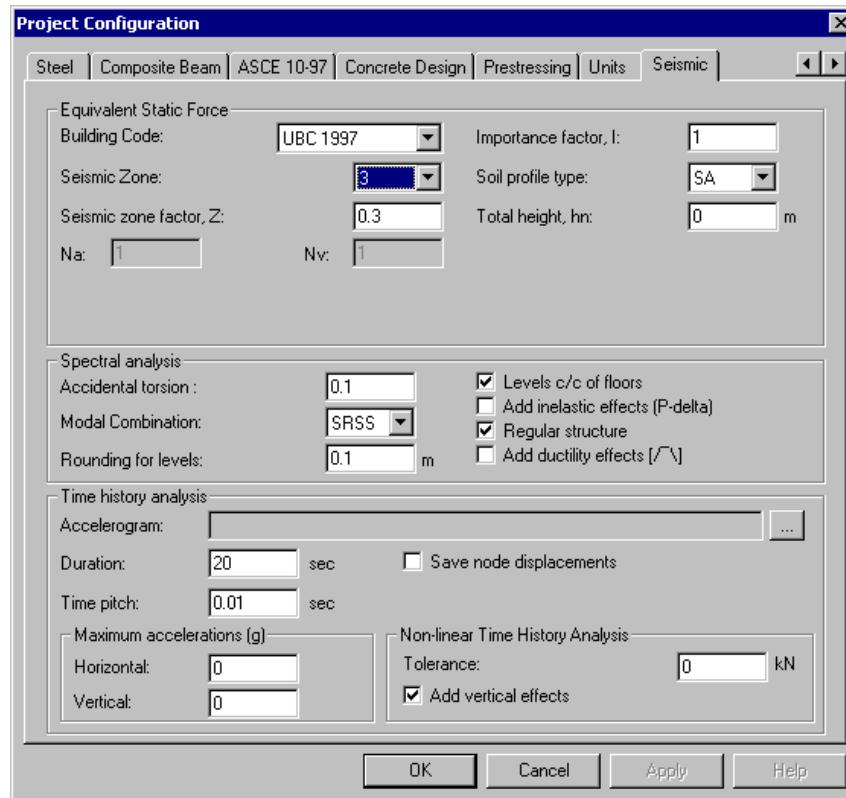
This table describes the fields included in this tab when the *UBC 97* has been selected in the "Building Code" list box.

Parameters	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.
Seismic zone	Select the seismic zone among the list box, according to article 1629.4.1 and figure 16-2.

Parameters	Definition
Seismic zone factor, Z	Seismic zone factor calculated from the acceleration-related seismic zone.
Na	Near source factor affecting Ca, based on Table 16-S.
Nv	Near source factor affecting Cv, based on Table 16-T.
Importance factor, I	Seismic importance factor of the structure, according to Table 16-K.
Type of Soil Profile	Select a type of soil profile among the list box. Refer to Table 16-J.
Total height, hn	Total height of the structure from the base, where the base of the structure is the level at which horizontal earthquake motions are considered to be imparted to the structure.

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.
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). Inelastic effects are calculated according to article 4.1.9.1. 28) of NBC 1995.
Regular structure	Check this option if the building has a regular geometry.
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.

Parameters	Definition
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**.

See also
Static Analysis
Dynamic Analysis
Project Configuration
Overall Ductility of a Structure: Force Modification Factor R
Calibration of Spectral, Linear and Non-linear Time History analysis

---

Modal Analysis

General

The modal analysis calculates the values and vectors (frequencies and mode shapes) specific to the structure, with the *Subspace Iteration Method*. During a modal analysis, VisualDesign calculates the modal contribution for each mode and for the three main directions, which are needed for the spectral analysis. These modal contributions are written in the **Frequencies and Vibration Modes** spreadsheet. Consequently, when the spectral analysis is launched, the software can automatically evaluate the vibration mode that contributes the most, for a given seismic direction.

Before running a modal analysis, the engineer has to define a *Mass* load combination according to the proper building code. This mass load combination shall be selected in the **Modal Analysis** dialog box, which appears on the screen when the Modal analysis is launched. The user shall specify the desired number of modes. The numerical model will then compute the *n* first required modes and classify them in an orderly manner. Refer to Number of Calculated Modes.

Excerpts from:

Dr Edward L. Wilson, *Articles on Numerical Techniques used in SAP2000, ETABS and SAFE*, Chap. 12, 15 and 17. University of California, Berkeley.

"All real physical structures, when subjected to loads or displacements, behave dynamically. The additional inertia forces, from Newton's second law, are equal to the mass times the acceleration. If the loads or displacements are applied very slowly then the inertia forces can be neglected and a static load analysis can be justified. Hence, dynamic analysis is a simple extension of static analysis.

"In addition, all real structures potentially have an infinite number of displacements. Therefore, the most critical phase of a structural analysis is to create a computer model, with a finite number of massless members and a finite number of node (joint) displacements, which will simulate the behaviour of the real structure. The mass of a structural system, which can be accurately estimated, is lumped at the nodes. Also, for linear elastic structures the stiffness properties of the members, with the aid of experimental data, can be approximated with a high degree of confidence. However, the dynamic loading, energy dissipation properties and boundary (foundation) conditions for many structures are difficult to estimate. This is always true for the cases of seismic input or wind loads.

"To reduce the errors that may be caused by the approximations summarized in the previous paragraph, it is necessary to conduct many different dynamic analyses using different computer models, loading and boundary conditions."

[…] It is important for engineers to realize that the displacements, which are normally printed by a computer program, are relative displacements and that the fundamental loading on the structure is foundation displacements and not externally applied loads at the joints of the structure.

See also

Vibration Modes Spreadsheet

Non-Linear Seismic Directions Spreadsheet

Information on Levels according to Seismic Direction

Spectral Analysis

Modal Analysis Dialog Box

| ω |

The "Modal Analysis" icon of Tools toolbar

Open the **Modal Analysis** dialog box by clicking the icon on Tools toolbar or select Modal Analysis in the Analysis menu of VisualDesign main window. Complete the required parameters and press the "Analyse" button to launch the analysis.

Description of this dialog box:

Parameters	Description
Structure	
Name	Name of the project, as specified in the General tab of project Configuration dialog box.
Elements	Number of elements composing the structural model.
Calculation of eigenvalues and eigenvectors	
Weight load combination	Select the "Weight" load combination that will be used to calculate the natural frequency. It must include 100% of the structure dead load plus 25% of snow load, if applicable.
Number of requested modes	It represents the number of specific values and vectors. For a spectral analysis in keeping with the *National Building Code – Canada 1995*, you must select 2 to 3 modes per story.
Number of calculated modes	Number of modes to be calculated in order to obtain a quicker convergence of the analysis. A default value is suggested, depending on the number of desired modes. Refer to topic Number of Calculated Modes.
Consider horizontal DDF only	Activate this option if you want to get horizontal vibration modes only. Gravity vibration modes will not be considered.
Include non linear effects	Activate this option if there are cables in the structure or if the building is very tall (40 stories).
Number of iterations	Number of iterations for modal analysis. Default value is 500.
Tolerance	Tolerance for modal analysis convergence. Default value is 1E-10 and is usually satisfying.
Results window	This window shows the modal calculations in progress. If you get a message that is warning you about the Jacobi matrix, it means that the structure is not stable.

DATE AND TIME

VisualDesign now displays the date of analysis in the **Modal 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.

Number of Calculated Modes

The **Number of Calculated Modes**, which is part of the **Modal Analysis** dialog box, is automatically initialized by VisualDesign and is equal to: Nos. of requested modes + 8, according to the numerical method *Subspace Iteration Method.* In general, this number is sufficient. However, when a project is composed of many nodes and where the number of requested modes is high, such as 200, VisualDesign will calculate the parameter *Number of calculated modes* as follows:

Nos. of requested modes + max [8 and 0.01 [3 x (nos. nodes-1)]]

A higher number of calculated modes reduces the number of iterations that is needed to find the number of requested modes, because the "Space" is much larger. Consequently, the modal analysis is much faster. For example, if the number of requested modes is 200, the number or calculated modes will be around 225 or 230 depending on the number of nodes, instead of 208.

The user can modify this number of calculated modes.

Modal Contribution of each Mode

During the modal analysis, VisualDesign automatically calculates the modal contribution for each vibration mode for the three x, y and z directions. These contributions are written in the *Frequencies and Vibration Modes* spreadsheet and results are saved. During the spectral analysis, VisualDesign can evaluate the vibration mode that contributes the most to the participating mass and then, determine the proper vibration mode to use for the direction to be analysed.

You will find the modal contributions γx, γy and γz in the *Frequencies and Vibration Modes* spreadsheet once that the modal analysis is completed. From these values, it is easy to calculate the modal contribution for each mode by the scalar product of γ and the normalized seismic direction.

The participating modal mass for the ith mode will be:

$$m_i = (\vec{\gamma} \cdot \vec{d})^2 M_{gi}$$

where:

d: direction of earthquake;

Mgi: Generalized mass of ith mode;

Therefore, the fraction of participating mass for this mode will be:

$$f_i = \frac{m_i}{M_{total}} = \frac{(\vec{\gamma} \cdot \vec{d})^2 M_g}{M_{total}}$$

During a spectral analysis, the user can verify that the mode identified by VisualDesign, corresponds exactly to the mode that contributes the most to the participating mass.

Code S6-00

For a spectral analysis using Code S6-00, the fundamental vibration mode that is identified by the program now allows the exception of fundamental mode according to clause 4.4.7.2 b).

Cyclic Design

Finally, during a cyclic design, frequencies and vibration modes can vary consistently. From now on, VisualDesign will automatically readjust the selection of vibration modes and seismic directions because they will be validated at each cycle of design.

See also

The Frequencies and Vibration Modes spreadsheet
The Seismic Directions spreadsheet
Spectral Analysis
Procedures for Spectral Analysis

Damping (Modal)

The damping percentage of 5%, indicated in the **Frequencies and Vibration Modes** spreadsheet, is in accordance with most Canadian and American building codes. However, this value is editable. The modified damping will be saved when the modal analysis will be launched again, and during the design process.

Modal Analysis Procedures

- Specify the seismic analysis parameters in the **Seismic** tab of **Project Configuration**. The spectral accelerations are determined according to the selected code.

- Create a "Mass" load combination that will include all structure dead load plus 25% of snow load, if applicable.

- Open the **Modal Analysis** dialog box by clicking the icon ω .

- Select the Mass load combination and activate the option "Consider horizontal degrees of freedom" if you want to get horizontal vibration modes only. Complete other parameters.

- Launch the modal analysis by clicking the "Analyse" button.

Jacobi matrix

If you get a warning message about the Jacobi matrix, it means that the structure is not stable. Look at member end conditions. Refer to topic "Null Pivot" in chapter 5 (Static analysis).

- Select a vibration mode on Activation toolbar and look at displacements. Use the "Increase amplitude" function (Diagram toolbar). Do that for the first 10 vibration modes because it can help you realizing that the structure doesn't behave properly and locating modeling errors.

- Look at results in the **Frequencies and Vibration modes** spreadsheet. A default damping of 5% is specified in this spreadsheet because it is recommended by building codes. However, you are allowed to specify another value for the damping but you will have to launch the modal analysis again.

- Activate a vibration mode on Activation toolbar and consult node displacements in **Results / Modal/Spectral**.

See also
Spectral Analysis

Modal Analysis Results

Frequencies and Vibration Modes Spreadsheet

Group: Dynamic Analysis Results

Column	Description	Editing
Mode	Vibration mode number.	No
w	Frequency	No
f	Natural frequency	No
T	Period	No
ξ	Percentage of modal damping (%), generally equal to 5% for most building codes. This value can be modified. However, a new analysis must be launched..	No
Shape	Type of deformation	No
x-dir.	Mode component – global x direction	No
y-dir.	Mode component – global y direction	No
z-dir.	Mode component – global z direction	No
γx	Modal contribution in the x-direction.	No
γy	Modal contribution in the y-direction.	No
γz	Modal contribution in the z-direction.	No
Sar	Spectral acceleration for this mode	No
Sdr	Spectral displacement for this mode	No
Svr	Spectral speed for this mode	No
Mg	Generalized mass for this mode	No

CivilDesign Inc. Engineering Software

Node Displacements for a Vibration Mode

Node displacements are available for a selected vibration mode. Activate the Vibration mode activation mode on Activation toolbar and select one. Then go to **Results** / **Modal/Spectral / Nodes displacements**.

Group: Dynamic Results: (title)

Column	Description	Editing
Number	12 alphanumeric characters	No
Displ. x	Displacement in the global x-axis system	No
Displ. y	Displacement in the global y-axis system	No
Displ. Z	Displacement in the global z-axis 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

See also
Modal Analysis

Spectral Analysis

General

This **Spectral** Analysis is based on the article [NBC 4.1.9.1. 13b)] of the User's Manual – **NBC 1995** *Comments on calculating structures* (Part 4), prepared by the NRC.

A modal analysis must be executed before performing a spectral analysis.

According to the modes shapes obtained from the Modal analysis, the spectral analysis will compute, for each seismic direction and each mode shape, the maximum forces and displacements by using a spectrum and scaling factors given by the code.

First, we determine the minimum static base shear that will be applied to the structure according to the desired seismic direction and considering the empirical period given by the code and the corresponding period calculated from the eigenvectors and eigenvalues. Then, we run the spectral analysis, which will add up all the structure internal forces and deflections, supports reactions and base shear, for each mode. When all mode shapes have been processed, the program calculates the resultant based on the SRSS method (*Square Root of Sum of Squares*) or the CQC method (*Complete Quadratic Combination*) and we calibrate the forces according to the minimum base shear. Then, we compare with the fundamental period and the base shear calculated by VisualDesign™. According to the chosen code, we then find the minimum base shear to use for the design.

This value (V) allows us to calibrate static forces applied to the model's nodes to simulate seismic forces in a static analysis. Accidental torsional effects are statically added to the previous results. It is done also for inelastic effects.

The spectral analysis generates results envelopes (**E**) that can be included in static analysis load combinations. The results spreadsheets for diagrams and reactions provide the maximum and minimum values, for a given load combination that includes this spectral envelope and other load cases.

The spectral analysis using **UBC 1994** is based on the division III-EARTHQUAKE DESIGN, sections 1624 to 1629.

The spectral analysis using **UBC 1997** is based on the division IV-EARTHQUAKE DESIGN, sections 1626 to 1635.

See also

Approximations and Limitations of the Response Spectrum Method
Modal Analysis
Participating Mass
Spectral Analysis Procedures

Approximations and Limitations of the Response Spectrum Method

Excerpts from:

Dr Edward L. Wilson, Articles on Numerical Techniques used in SAP2000, ETABS and SAFE, Chap. 12, 15 and 17. University of California, Berkeley.

 […]"This method cannot be used to approximate the non-linear response of a complex three-dimensional structural system.

The recent increase in the speed of computers has made it practical to run many time history analysis in a short period of time, which produces superior results, since each member is not designed for maximum peak values as required by the response spectrum method."

Design spectra

"Design spectra are not uneven curves since they are intended to be the average of many earthquakes. […] Many codes have defined specific equations for each range of the spectrum curve for different soil types. For major structures it is now common practice to develop a site-dependant design spectrum which includes the effect of local soil conditions and distance to the nearest faults."

Orthogonal effects in spectral analysis

"It is reasonable to assume that motions that take place during an earthquake have one principal direction. Or, during a finite period of time when maximum ground acceleration occurs, a principal direction exists. For most structures this direction is not known and, for most geographical locations, cannot be estimated. Therefore, the only rational earthquake design criterion is that the structure must resist an earthquake of a given magnitude from any possible direction. In addition to the motion in the principal direction, a probability exists that motions normal to that direction will occur simultaneously. In addition, because of the complex nature of three dimensional wave propagation, it is valid to assume that these normal motions are statistically independent."

Recommendations on Orthogonal Effects

"For three dimensional response spectra analysis, it has been shown that the design of elements for 100 percent of the prescribed seismic forces in one direction plus 30 or 40 percent of the prescribed forces applied in the perpendicular direction is dependent on the user's selection of the reference system. These commonly used percentage combination rules are empirical and can underestimate the design forces in certain members and produce a member design that is relatively weak in one direction. It has been shown that the alternate building code approved method, in which an SRSS combination of two 100 percent spectra analyses with respect to any user defined orthogonal axes, will produce design forces that are not a function of the reference system. Therefore, the resulting structural design has equal resistance to seismic motions from all directions."

The CQC3 method uses two spectra in orthogonal directions separated by an arbitrary angle theta. The minor spectrum is assumed as a fraction of the major spectrum. This fraction is called a. If the two spectra are equal, meaning that a=1, the CQC3 method reduces to the SRSS method. "This method […] should be used if a value of a less than 1.0 can be justified. It will produce realistic results that are not a function of the user selected reference system. "

Limitations of the Response Spectrum Method

[…] The response spectrum method is an approximate method used to estimate maximum peak values of displacements and forces and has significant limitations. It is restricted to linear elastic analysis in which the damping properties can only be estimated with a low degree of confidence. Non-linear spectra, which are commonly used, have very little theoretical background. They should not be used for the analysis of complex three-dimensional structures. For such structures, true non-linear time-history response should be used."

Participating Mass for Spectral Analysis

To get an accurate spectral analysis, building codes generally require that the mass participation to displacement of a structure in a given direction, be at least 90% of the structure dead weight plus 25% of the snow load.

To get this percentage, the number of calculated modes can be increased in the modal analysis or the dead weight that is distributed to supports can be modified in such a way to minimize this weight and to maximize the weight acting above supports.

The weight of a member is equally distributed at end nodes. Typically, half of column weight is distributed to support node in the first story of a building, so this weight is lost and is not considered in the participating mass of the structure. To reduce the distributed weight that goes to supports, split the columns at a level of about 0.5m above supports.

Shear Wall

The same principle applies to a shear wall that is composed of plates and bearing on support nodes. It would be interesting to specify the first row of nodes above supports as "Level Nodes" for the design of such wall. Refer to Node Characteristics dialog box.

See also

Modal Analysis

Spectral Analysis Procedures

Accidental Torsion Effects

Seismic Analysis Modeling to Satisfy Building Codes

CQC Method
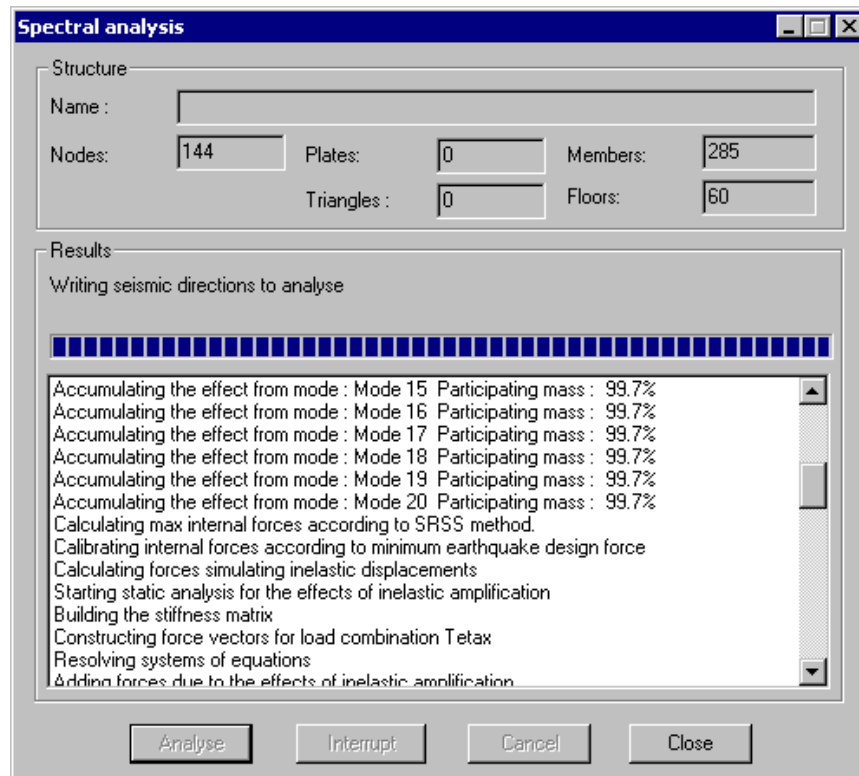
Effects of earthquakes according to CNBC 1995

Spectral Analysis Dialog Box

The "Spectral Analysis" icon of Tools toolbar

A modal analysis must be executed before performing a spectral analysis.

- Open the **Spectral Analysis** dialog box by clicking this icon on Tools toolbar or select **Spectral Analysis** in **Analysis** menu of VisualDesign main window.

- Click the "Analyse" button to launch the analysis.

The spectral analysis generates spectral envelopes (**E01, E02** or **E03**) that can be included in static analysis load combinations.

Date and Time

VisualDesign displays the date of analysis in the **Spectral 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.

Seismic Analysis - CQC Method (Complete Quadratic Combination)

Excerpt from:

Earthquake Engineering And Structural Dynamics, Vol.9, 187-194 (1981) *Short Communications – A Replacement for the SRSS Method in Seismic analysis*, John Wiley & Sons, Ltd, p. 187-192.

"It is well-known that the application of the Square-Root-of- Sum-of-Squares (SRSS) method in seismic analysis for combining modal maxima can cause significant errors. Nevertheless, this method continues to be used by the profession for significant buildings. The purpose of this note is to present an improved technique to be used in place of the SRSS method in seismic analysis.

A Complete Quadratic Combination (CQC) method is proposed which reduces errors in modal combination in all examples studied. The CQC method degenerates into the SRSS method for systems with well-spaced natural frequencies. Since the CQC method only involves a small increase in numerical effort, it is recommended that the new approach be used as a replacement for the SRSS method in all response spectrum calculations.

[…] The CQC method requires that all modal response terms be combined by the application of the following equations:

For a typical displacement component, uk:

$$u\mathrm{k} = \sqrt{\left(\sum_i \sum_j u_{ki} \cdot \rho_{ij} \cdot u_{kj} \right)}$$

(12a)

For a typical force component, fk:

$$f\mathrm{k} = \sqrt{\left(\sum_i \sum_j f_{ki} \cdot \rho_{ij} \cdot f_{kj} \right)}$$

(12b)

where $u$ki is a typical component of the modal displacement response vector Uf max, and $f$ki is a typical force component that is produced by the modal displacement vector Uf max. Note that this combination formula is of a complete quadratic form including all cross-modal terms, hence, the reason for the name Complete Quadratic Combination. It is also important to note that the cross-modal terms in equations (12) may assume positive or negative values depending on whether the corresponding modal responses have the same or opposite signs. The signs of the modal responses are, therefore, an important key to the accuracy of the CQC method.

In general, the cross-modal coefficients ϱij are functions of the duration and frequency content of the loading and of the modal frequencies and damping ratios of the structure. If the duration of earthquake is long compared to the periods of the structure, and if the earthquake spectrum is smooth over a wide range of frequencies, then, it is possible to approximate these coefficients by:

$$\varrho ij = \frac{8 \cdot \sqrt{\left(\zeta_i \cdot \zeta_j\right)}\left(\zeta_i + \zeta_j\right) r^{3/2}}{\left(1 - r^2\right)^2 + 4 \cdot \zeta_i \zeta_j \cdot r \cdot \left(1 + r^2\right) + 4 \cdot \left(\zeta_i^2 + \zeta_j^2\right) \cdot r^2}$$

where r = ωi / ωj. For constant modal damping, ζ, this expression reduces to:

$$\varrho ij = \frac{8 \cdot \zeta^2 \cdot \left(1 + r\right) r^{3/2}}{\left(1 - r^2\right)^2 + 4 \cdot \zeta^2 \cdot r \cdot \left(1 + r\right)^2} = \frac{8 \cdot \zeta^2 \cdot r^{3/2}}{\left(\left(1 - r\right)^2 + 4 \cdot \zeta^2 \cdot r\right) \cdot \left(1 + r\right)}$$

Note that for equal damping and r = 1, ϱij = 1.0

Final Remarks

[…] It should also be pointed out that the SRSS method gives good results for some structures subjected to two-directional seismic input, even when the modal frequencies are closely spaced. It can also be shown that this is due to cancelling of the cross-modal terms corresponding to the two directions of input. This phenomenon, however, is not generally true. For example, when the two components of input are of different intensities, or when the three-dimensional structure is highly asymmetric, the cross-modal terms would still be significant and, therefore, the SRSS method will lead to erroneous results.

Based on the preceding numerical example and the above discussion, it is strongly recommended that the use of the SRSS method for seismic response analysis of structures be immediately discontinued. Continued use of the SRSS technique may dramatically overestimate the required design forces in some structural elements or it may significantly underestimate the forces in other elements. The proposed CQC method is based on fundamental theories of random vibration. It consistently yields accurate results when compared to time-history analyses.

See also
Effects of earthquakes according to the CNBC 1995
Participating Mass

Spectral Analysis Procedures

Modal Analysis

- Specify the seismic analysis parameters in the **Seismic** tab of **Project Configuration**. The spectral accelerations are determined according to the selected code.

- Create a "Mass" load combination that will include all structure dead load plus 25% of snow load.

- Run a modal analysis to obtain the structure's natural frequencies. The **Modal Analysis** dialog box will appear on your screen. Complete the required parameters and click the Analyse button.

Spectral Analysis

- Open the **Linear Seismic Directions** spreadsheet and insert two lines. Give a name to each seismic direction and enter the direction vectors, usually 1.0, in appropriate columns. VisualDesign will automatically initialize main vibration modes, according to maximum modal contributions obtained in the **Frequencies and Vibration modes** spreadsheet.

- Open the **Spectral** analysis dialog box and click the Analyse button.

At least 90% of participating weight must be attained for the two main directions. To be sure, open the **Linear Seismic Directions** spreadsheet again and consult the percentages obtained at the far right of the spreadsheet.

The spectral analysis provides seismic envelopes and generates static torsion loads induced by accidental torsion effects. VisualDesign transforms torsion loads into equivalent static loads. See column "Auto generated Torsion" in the **Loads Definition** spreadsheet. Seismic envelopes "E01, E02, E03" can be included in load combinations, using the **Load Combination Generator**.

Linear Seismic Directions

Linear Seismic Directions Spreadsheet

The **Linear Seismic Directions** spreadsheet must be completed before launching a spectral analysis. If the spreadsheet is empty, insert the required number of lines, usually two, because you need to analyse at least two orthogonal seismic directions.

Maximum Modal Contributions

VisualDesign automatically initializes appropriate seismic directions in the **Linear Seismic Directions** spreadsheet according to maximum modal contributions obtained in the **Frequencies and Vibration modes** spreadsheet, for each mode and main directions.

Vectors Dir. x, Dir. y, and Dir. z

Users must enter components in the **Linear Seismic Directions** spreadsheet. They are not automatically initialized. Take care to note these components (Dir. x, Dir. y, Dir. z), corresponding to maximum modal contributions, as they are supplied in the **Frequencies and Vibration modes** spreadsheet.

If main directions act at an angle with global axis system, you can obtain components such as 0.86 for dir. x and 0.35 for dir. z. You must enter these orthogonal components in the **Linear Seismic Directions** spreadsheet. Do the same for the second seismic direction, which is supposed to act at 90 degrees.

Spectral Envelopes

The force modification factors R and corresponding envelope numbers must be specified for each linear seismic direction. The type of Seismic Force Resisting System (SFRS) shall also be specified. Refer to the selected code in the **Seismic** tab of **Project Configuration** dialog box (NBC-95, NBC-2005, CAN S6-00, UBC-94, and UBC-97).

Each code possesses its specific **Linear Seismic Directions** spreadsheet:

Linear Seismic Directions - NBC 95
Linear Seismic Directions - NBC 2005
Linear Seismic Directions - CAN-S6-00
Linear Seismic Directions - UBC-94
Linear Seismic Directions - UBC-97

Linear Seismic Directions - CNBC 2005

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Name or number for this seismic direction (12 alphanumeric characters).	Single click
Dir. x	Global x component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. y	Global y component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. z	Global z component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Envelope	Select an envelope that will include this seismic direction, if desired. Usually, an envelope is assigned per seismic direction. But it could also include more than one seismic direction. Refer to the Load Combination Generator to include these envelopes into the generation.	Double-click
Tdyn (Mode)	This vibration mode has the biggest modal contribution for this direction. VisualDesign automatically initializes it. This parameter is used to calculate the value of S dyn.	Single click
Calibration	To calibrate seismic forces, activate this option [x].	Double-click or Space bar
Torsion	Activate this option [x] to include accidental torsion effects into seismic calculations. The torsion moment will be supplied at column "Torsion M" when the spectral analysis will be completed.	Double-click or Space bar
Structure	Choose the type of frame that will be used for the calculation of empirical period. See the note below.	Double-click
Seismic Force Resistance System	Select the type of SFRS that will resist seismic forces for this direction: Double click to open the selection tree that is composed of a list of SFRS available for each standard CSA- S16-01, CSA-A23.3, CSA-O86, and CSA-S304.1.	Double-click

Column	Description	Editing
Rd	Ductility-related force modification factor, indicated in Table 4.1.8.9.	Single click
Ro	Overstrength-related force modification factor, indicated in Table 4.1.8.9.	Single click
T emp	Empirically calculated period, as per code	No
T dyn	Period based on dynamic analysis.	No
T selected	Selected period.	No
S (Ta)	Seismic response coefficient calculated using the selected period.	No
Mv	Factor that accounts for the effect due to superior mode over the base shear, according to paragraph 4.1.8.11 5).	No
W	Total weight of the structure, acting towards this direction.	No
V	Minimal lateral seismic force at the base of the structure.	No
Ve	Equivalent lateral force exerted at the base of the structure and representing the elastic response, as per code.	No
Vd	Lateral seismic force exerted at the base of the structure, based on the dynamic analysis. (This value includes factors Ro, Rd, and F). The result is available when the spectral analysis is completed.	No
V selected	Lateral seismic force that is selected.	No
Torsion M	Torsional moment at the base of the structure.	No
Modal M / M	Percentage of modal mass used. This result is available when the spectral analysis is completed.	No

Explanatory note for the selection of a type of frame used to calculate the empirical period, Ta, according to CNB-2005:

Braced Frame $T = 0.025 \, hn$

MRF General where $T = 0.1N$ (General moment resisting frame);

MRF Steel where $T = 0.085 \, (hn)^{3/4}$ (Steel moment resisting frame);

MRF Concrete where T = 0.075 (hn) ¾   (Concrete moment resisting frame);

Other where T = 0.05 (hn)¾   (Shear wall or other structures).

See also
Force modification factors Rd and Ro

Force Modification Factors Ro and Rd (NBC-2005)

Table 4.1.8.9. Ductility-related force modification factor, Rd, for SFRS, overstrength-related force modification factor, Ro, and general restrictions (1)

Cases A to E are used to lighten the table below:

Case A: If IE *Fa * Sa (0,2) < 0,2

Case B: If IE *Fa * Sa (0,2) ≥0,2 and <0,35

Case C: If IE *Fa * Sa (0,2) ≥0,35 and ≤0,75

Case D: If IE *Fa * Sa (0,2) > 0,75

Case E: If IE*Fv*Sa (1,0) > 0,3

Type of SFRS	Rd	RO	Restrictions (2)				
			A	B	C	D	E
Steel structures in accordance with CSA-S16 standard							
Ductile moment-resisting frames	5,0	1,5	NL	NL	NL	NL	NL
Moderately ductile moment-resisting frames	3,5	1,5	NL	NL	NL	NL	NL
Limited-ductility moment-resisting frames	2,0	1,3	NL	NL	60	NA	NA
Moderately ductile concentrically braced frames:							
• Bracings without K elements	3,0	1,3	NL	NL	40	40	40
• K-bracings	3,0	1,3	NL	NL	40	40	40
• Tension-only bracings	3,0	1,3	NL	NL	20	20	20
Limited-ductility concentrically braced frames:							
• Bracings without K elements	2,0	1,3	NL	NL	60	60	60
• K-bracings	2,0	1,3	NL	NL	60	60	60
• Tension-only bracings	2,0	1,3	NL	NL	40	40	40

Type of SFRS	Rd	RO	Restrictions (2)				
Ductile eccentrically braced frames	4,0	1,5	NL	NL	NL	NL	NL
			A	B	C	D	E
Ductile plate walls	5,0	1,6	NL	NL	NL	NL	NL
Moderately ductile plate walls	2,0	1,5	NL	NL	60	60	60
Conventional construction (shear wall, braced frame or moment-resisting frame).	1,5	1,3	NL	NL	15	15	15
Other steel SFRSs.	1,0	1,0	15	15	NA	NA	NA
Concrete structures in accordance with CSA-A23.3 standard							
Ductile moment-resisting frames	4,0	1,7	NL	NL	NL	NL	NL
Moderately ductile moment-resisting frames	2,5	1,4	NL	NL	60	40	40
Ductile coupled walls	4,0	1,7	NL	NL	NL	NL	NL
Ductile partially coupled walls	3,5	1,7	NL	NL	NL	NL	NL
Ductile shear walls	3,5	1,6	NL	NL	NL	NL	NL
Moderately ductile shear walls	2,0	1,4	NL	NL	NL	60	60
Conventional construction:							
• Moment-resisting frames	1,5	1,3	NL	NL	15	NA	NA
• Shear walls	1,5	1,3	NL	NL	40	30	30
Other concrete SFRSs.	1,0	1,0	15	15	NA	NA	NA
Timber structures in accordance with CSA-086 standard							
Nailed shear walls with wood-based panels	3,0	1,7	NL	NL	30	20	20
Shear walls with wood-based and gypsum panels in combination	2,0	1,7	NL	NL	20	20	20
Moderately ductile braced frames or moment-resisting frames	2,0	1,5	NL	NL	20	20	20
Limited-ductility braced frames or moment-resisting frames	1,5	1,5	NL	NL	15	15	15

Type of SFRS	Rd	RO	Restrictions (2)				
Other wood- or gypsum-based SFRSs.	1,0	1,0	15	15	NA	NA	NA
			A	B	C	D	E
Masonry structures in accordance with CSA-S304.1 standard							
Moderately ductile shear walls	2,0	1,5	NL	NL	60	40	40
Limited-ductility shear walls	1,5	1,5	NL	NL	40	30	30
Conventional construction:							
• Shear walls	1,5	1,5	NL	60	30	15	15
• Moment-resisting frames	1,5	1,5	NL	30	NA	NA	NA
Unreinforced masonry	1,0	1,0	30	15	NA	NA	NA
Other masonry SFRSs.	1,0	1,0	15	15	NA	NA	NA

Refer to article 4.1.8.10 *Other Restrictions*.

Precisions relatively to restrictions:

- NA means "Not allowed".

- Numbers represent limits for maximum heights, in mm.

- NL means "Not limited", i.e. this system is permitted and no limit applies to the height of the SFRS. However, the height may be restricted in other part of the NBC.

- The most rigorous provisions prevail.

Linear Seismic Directions - CNBC 1995

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Name or number for this seismic direction (12 alphanumeric characters)	Single click
Dir. x	Global x-component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. y	Global y component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. z	Global z component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Envelope	Select an envelope that will include this seismic direction, if desired. Usually, an envelope is assigned per seismic direction. But it could also include more than one seismic direction. Refer to the Load Combination Generator to include these envelopes into the generation.	Double-click
Tdyn (Mode)	This vibration mode has the biggest modal contribution for this direction. VisualDesign automatically initializes it. This parameter is used to calculate the value of S dyn.	Single click
Calibration	To calibrate seismic forces, activate this option [x].	Double-click or Space bar
R	Enter the force modification factor that represents the ductility and over-strength of a structure (or one of its components), and the capability to dissipate energy through inelastic behaviour. See topic "Overall ductility of a structure: factor R"	Single click
Torsion	Activate this option [x] to include accidental torsion effects into seismic calculations. The torsion moment will be supplied at column "Torsion M" when the spectral analysis will be completed.	Double-click or Space bar

CivilDesign Inc. Engineering Software

Column	Description	Editing
Structure	Choose the type of frame that will be used for the calculation of empirical period. See the note below.	Double-click
Ds	2D: Dimension of wall or braced framing which constitutes the principal lateral resisting system against earthquake forces. 3D: sum of wall or braced frames dimensions. Rigid frame: Total width. Refer to NBC-95, Clause 4.1.9.1.7b)	Single click
T emp	The empirically calculated period, as per code.	No
T dyn	Period based on dynamic analysis.	No
S emp	Seismic response coefficient, empirically calculated as per code.	No
S dyn	Seismic response coefficient calculated using the appropriate period of dynamic analysis.	No
S	Selected seismic response coefficient.	No
Ve	Equivalent lateral force exerted at the base of the structure and representing the elastic response	No
W	Total mass of the structure acting in this direction.	No
V	Minimum lateral seismic force at the base of the structure.	No
Vdyn	Lateral seismic force exerted at the base of the structure, based on the dynamic analysis. (This value does not include factors R and F.) The result is available when the spectral analysis is completed.	No
Torsion M	Torsion moment at the base, created by accidental torsion effects.	No
Modal M/M	Percentage of modal mass used. This result is available when the spectral analysis is completed.	No

Explanatory note for the selection of a type of frame used to calculate the empirical period according to CNB-95:

According to the type of frame, the user must select one of the following options (the formula is written to avoid any confusion):

RF General where T = 0.1N   (General moment resisting frame);

RF Steel where T = 0.085 $(hn)^{3/4}$ (Steel moment resisting frame);

RF Concrete where T = 0.075 $(hn)^{3/4}$ (Concrete moment resisting frame);

Other where T = 0.09 $hn/(Ds)^{1/2}$ (Braced structure).

Force modification factor, R, CNBC-95

National Building Code of Canada 1995 – Seismic Calculation

A force modification factor, R, must be selected in the **Linear Seismic** spreadsheet. This factor represents the overall ductility of the structure or its components. The force modification factor expresses the structural capacity to dissipate energy through inelastic behaviour during an earthquake.

The table below (CNBC 1995) gives a description of bracing systems that can resist lateral forces in a building and the corresponding factor **R**.

Table 4.1.9.1.B   Force Modification Factors, R

CASE	TYPE OF LATERAL - FORCE – RESISTING - SYSTEM	R
	Steel Structures Designed and Detailed According to CAN/CSA-S16.1	
1	Ductile moment-resisting frame	4,0
2	Ductile eccentrically braced frame	4,0
3	Ductile steel plate shear wall	4,0
4	Ductile braced frame	3,0
5	Moment-resisting frame with nominal ductility	3,0
6	Nominally ductile steel plate shear wall	3,0
7	Braced frame with nominal ductility	2,0
8	Ordinary steel plate shear wall	2,0
9	Other lateral-force-resisting systems not defined in Cases 1 to 8.	1,5
	Reinforced Concrete Structures Designed and Detailed According to CSA A23.3	
10	Ductile moment-resisting frame	4,0
11	Ductile coupled wall	4,0
12	Other ductile wall systems	3,5

CASE	TYPE OF LATERAL - FORCE – RESISTING - SYSTEM	R
13	Moment-resisting frame with nominal ductility	2,0
14	Wall with nominal ductility	2,0
15	Other lateral-force-resisting systems not defined in Cases 10 to 14	1,5
	Timber Structures Designed and Detailed According to CSA-O86.1	
16	Nailed shear panel with plywood, wafer board or OSB.	3,0
17	Concentrically braced heavy timber frame with ductile connections	2,0
18	Moment-resisting wood frame with ductile connections	2,0
19	Other systems not included in Cases 16 to 18	1,5
	Masonry Structures Designed and Detailed According to CSA-S304.1	
20	Reinforced masonry wall with nominal ductility	2,0
21	Reinforced masonry	1,5
22	Unreinforced masonry	1,0
23	**Other lateral-force-resisting Systems not Defined in Cases 1 to 22**	1,0

Refer to clause 4.1.9.3. *Special Provisions*, paragraph 1), 2) and 3).

Read the Commentary J – Effects of Earthquakes, User's Guide – NBC 1995 Structural Commentaries (Part 4)

See also
Local Ductility for the Design of a Lateral-Forces-Resisting-System
Seismic tab of Project Configuration
Dynamic, Spectral and Time History Analysis
Effects of Earthquakes - NBC 1995

Linear Seismic Directions - CAN/CSA-S6-00

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Name or number for this seismic direction (12 alphanumeric characters).	Single click
Dir. x	Global x component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. y	Global y component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. z	Global z component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Envelope	Select an envelope that will include this seismic direction, if desired. Usually, an envelope is assigned per seismic direction. But it could also include more than one seismic direction. Refer to the Load Combination Generator to include these envelopes into the generation.	Double-click
W	Total weight of the structure, acting towards this direction.	No
R	Enter the factor that represents the ductility and over-strength of a structure (or one of its components), and the capability to dissipate energy through inelastic behaviour. See topic "Force Modification Factor, R"	Single click
Vdyn	Lateral seismic force exerted at the base of the structure, based on the dynamic analysis. This result is available when the spectral analysis is completed.	No
Modal M / M	Percentage of modal mass used. This result is available when the spectral analysis is completed.	No

See also

Force Modification Factors R - NBC-95
Force Modification Factors Rd and Ro – NBC-2005

CivilDesign Inc. Engineering Software

Linear Seismic Directions - UBC-94

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Name or number for this seismic direction (12 alphanumeric characters).	Single click
Dir. x	Global x component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. y	Global y component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. z	Global z component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Envelope	Select an envelope that will include this seismic direction, if desired. Usually, an envelope is assigned per seismic direction. But it could also include more than one seismic direction. Refer to the Load Combination Generator to include these envelopes into the generation.	Double-click
Tdyn (Mode)	This vibration mode has the biggest modal contribution for this direction. VisualDesign automatically initializes it.	Single click
Calibration	To calibrate seismic forces, activate this option [x].	Double-click or Space bar
R	Enter the factor that represents the ductility and over-strength of a structure (or one of its components), and the capability to dissipate energy through inelastic behaviour.	Single click
Torsion	Activate this option [x] to include accidental torsion effects into seismic calculations. The torsion moment will be supplied at column "Torsion M" when the spectral analysis will be completed.	Double-click or Space bar
Structure	Choose the type of frame that will be used for the calculation of empirical period. Available options are: *SMRF*: Steel moment-resisting frame *CMRF*: Concrete moment-resisting frame *Other*.	Double-click

Column	Description	Editing
T Method A	Empirically period calculated according to art. 1630.2.2, Equation (30-8)	No
T Method B	Fundamental period calculated according to art. 1630.2.2, Equation (30-8).	No
C Method A	Seismic coefficient indicated at Table 16-Q.	No
C Method B	Seismic coefficient indicated at Table 16-R.	No
W	Total weight of the structure, acting towards this direction.	No
V Method A	Design base shear exerted at the base of the structure, according to art. 1630.2.1, Equation (30-4).	No
V Method B		No
V	Minimal lateral seismic force exerted at the base of the structure, according to art. 1630.2.1, Equation (30-6).	No
Vdyn	Lateral seismic force exerted at the base of the structure and obtained from the dynamic analysis. This result is available when the spectral analysis is completed.	No
Torsion M	Torsional moment at the base of the structure.	No
Modal M / M	Percentage of modal mass used. This result is available when the spectral analysis is completed.	No

Linear Seismic Directions - UBC-97

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Name or number for this seismic direction (12 alphanumeric characters).	Single click
Dir. x	Global x component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. y	Global y component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Dir. z	Global z component of seismic direction, as calculated in the Frequencies and Vibration Modes spreadsheet.	Single click
Envelope	Select an envelope that will include this seismic direction, if desired. Usually, an envelope is assigned per seismic direction. But it could also include more than one seismic direction. Refer to the Load Combination Generator to include these envelopes into the generation.	Double-click
Tdyn (Mode)	This vibration mode has the biggest modal contribution for this direction. VisualDesign automatically initializes it.	Single click
Calibration	To calibrate seismic forces, activate this option [x].	Double-click or Space bar
R	Enter the factor that represents the ductility and over-strength of a structure (or one of its components), and the capability to dissipate energy through inelastic behaviour.	Single click
Torsion	Activate this option [x] to include accidental torsion effects into seismic calculations. The torsion moment will be supplied at column "Torsion M" when the spectral analysis will be completed.	Double-click or Space bar
Structure	Choose the type of frame that will be used for the calculation of empirical period. Available options are: *SMRF*: Steel moment-resisting frame *CMRF*: Concrete moment-resisting frame *Other*.	Double-click

Column	Description	Editing
T Method A	Empirically period calculated according to art. 1630.2.2, Equation (30-8)	No
T Method B	Fundamental period calculated according to art. 1630.2.2, Equation (30-8).	No
Ca	Seismic coefficient indicated at Table 16-Q.	No
Cv	Seismic coefficient indicated at Table 16-R.	No
W	Total weight of the structure, acting towards this direction.	No
V Method A	Design base shear exerted at the base of the structure, according to art. 1630.2.1, Equation (30-4).	No
V Method B		No
V	Minimal lateral seismic force exerted at the base of the structure, according to art. 1630.2.1, Equation (30-6).	No
Vdyn	Lateral seismic force exerted at the base of the structure and obtained from the dynamic analysis. This result is available when the spectral analysis is completed.	No
Torsion M	Torsion moment at the base, created by accidental torsion effects.	No
Modal M / M	Percentage of modal mass used. This result is available when the spectral analysis is completed.	No

Spectral Analysis Results

Information on Levels According to Seismic Direction

When the spectral analysis is completed, and 90% was obtained as participating mass for the two main directions, open this spreadsheet (**Results / Spectral/Modal / Levels**. Make sure that interstorey drift does not exceeds the allowable limit, as per code.

Group: Spectral Analysis Results

Column	Description	Editing
Sis. Direction	Seismic direction applied to the level.	No
Height	Level height	No
Width	Width of level used for calculation of accidental torsion effects at this level	No
F	Lateral forces acting on this level	No
V	Mean shear stress at this level	No
W	Mass for this level	No
δ ave	Average displacement of all nodes in a level for a given spectral direction (the sum of node displacements at a level is divided by the number of nodes at this level). Displacements are not calibrated. Refer to topic Calibration.	No
δ max	Maximum displacement at ends of the structure, created by equivalent static forces acting at a distance of +/- 0.10Dnx from centre of gravity of each floor.	No
Bx	Ratio δ max/ δ ave at level x, which describes the building sensitiveness to torsion. This ratio also indicates the regularity in the geometry of the building	No
Δ mx	Corrected interstory drift, considering structural ductility.	No
hs	Height between this level and the one below	No
θ x	Stability factor	No

Steel Design Results - Seismic Design

CAN/CSA-S16-01

This spreadsheet includes calculated parameters and additional seismic results based on clause 27 of standard S16-01 for seismic design, for a chosen load combination. Therefore, load combinations that include a seismic envelope can be consulted.

Following a steel design with standard CAN/CSA-S16-01 with seismic loads, activate a load combination and go to **Results / Structure Design / Steel – Seismic Design**.

AISC-LRFD-95

Results are also available for American Standard AISC-LRFD-95.

Group: Load Combination Results

Column	Description	Editing
Number	Member number	No
Section	Steel shape assigned to this member.	No
Group	Group in which belongs this member.	No
Usage	Member usage	No
Ag	Gross area of this member.	No
Aw	Web area of this section.	No
Aw / Ag	Web area divided by the gross area.	No
Zx	Plastic section moment on strong axis	No
Ry	Factor applied to Fy for evaluating the probable yield strength of this member	No
Fy	Yield strength of steel	No
Class Bending Mx	Calculated class for bending on strong axis, Mx.	No
Class Bending My	Calculated class for bending on weak axis, My.	No
Class Web	Class considered in the calculation of web buckling resistance.	No
Class Compression	Class considered in the calculation of compression strength.	No

Column	Description	Editing
e	Length of the link beam.	No
Pf	Factored axial force in the link beam, according to selected load combination.	No
Vf	Factored shear force in the link beam, according to selected load combination.	No
Vp	0.55wdFy.	No
V'p	Value V'p based on clause 27.7.2 "Link beam Resistance", according to selected load combination.	No
Mp	Resisting plastic moment = Z Fy.	No
M'p	Value M'p based on clause 27.7.2 "Link beam Resistance", according to selected load combination.	No
ΦV'p 2ΦM'p/e	Shear resistance of the link beam. This resistance represents the lesser value of ΦV'p and 2ΦM'p.	No
1.1 RyMp	Probable moment for developing a plastic hinge.	No
AgRyFy	Probable tension to attain plasticity of the section.	No
1.2 Cpr	Probable compression, calculated with 1.1 fy.	No
0.2 AgRyFy	This value must be used along with 1.2 Cpr.	No
γ	Link rotation, according to clause 27.7.4.	No
γ max	Maximum link rotation, according to clause 27.7.4.	No
e min	Minimum length of the link beam.	No
e max	Maximum length of the link beam.	No
1.15 Ry Vn	Probable shear force to attain plasticity of the section.	No
1.30 Ry Vn	Probable shear force to attain plasticity of the section	No

See also
Steel Design Results (General)

Linear Time History Analysis

Linear Time History Analysis

The "Linear Time History Analysis" icon of Tools toolbar

The **Time History** Analysis calculates the stresses in a structure, according to a real accelerogram and a given direction.

Note: A modal analysis must have been carried out before executing a Time History Analysis.

The maximum time (in sec), the number of seismic directions and the number of subdivisions for the members are parameters that greatly influence the calculation duration of the **Time History** analysis.

Selection of Nodes

Before launching the analysis, a personalized selection must be created for nodes to be analysed. This selection must be selected in the **Time History** dialog box, as shown below. To learn more about personalized selection, please refer to topic Personalized selection of elements.

Configuration of parameters

All the parameters used to calibrate the Spectral analysis are also used to calibrate the **Time History** analysis.

The **Time History** generates a results envelope (Et) that you can include in your load combinations. The diagrams results spreadsheet and the reactions for a load combination that include this envelope (Seismic: Et) and other load cases give the combined Max and Min values.

Date and Time

VisualDesign now displays the date of analysis in the **Time History 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.
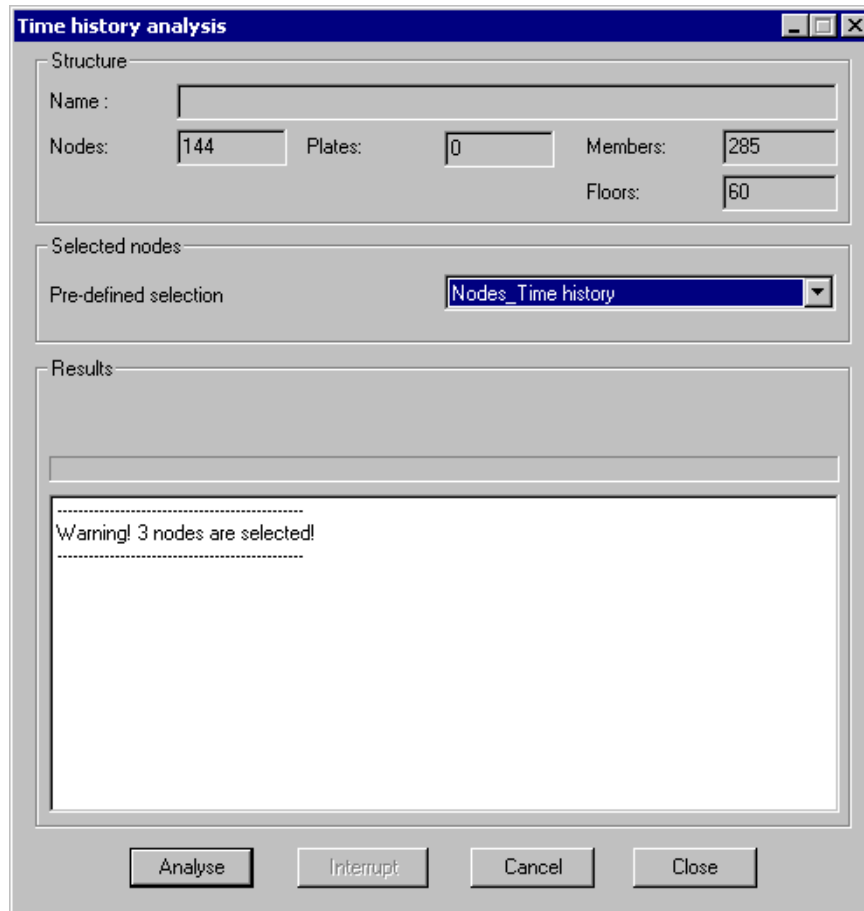
See also

Accelerograms

All accelerograms are stored in the *seismes.mdb* database. These accelerograms are normalized for 1.0g.

VisualDesign™ calibrates the chosen accelerogram according to the maximum horizontal acceleration or the maximum vertical acceleration, depending on seismic direction: vertical or horizontal.

Users can add their own accelerograms using Microsoft Access 97 software.

Selection of an Accelerogram

To select an accelerogram, do as follows:

- Select the "Seismic" tab of **Project Configuration** under the **File** menu;

- Click the […] button located next to the "Accelerogram" field;

- Finally, choose an accelerogram in the expanding tree. When the accelerogram is chosen, the data is read (a few seconds) and the accelerogram is displayed in the graphic. It can be printed through the toolbar posted above the accelerogram.
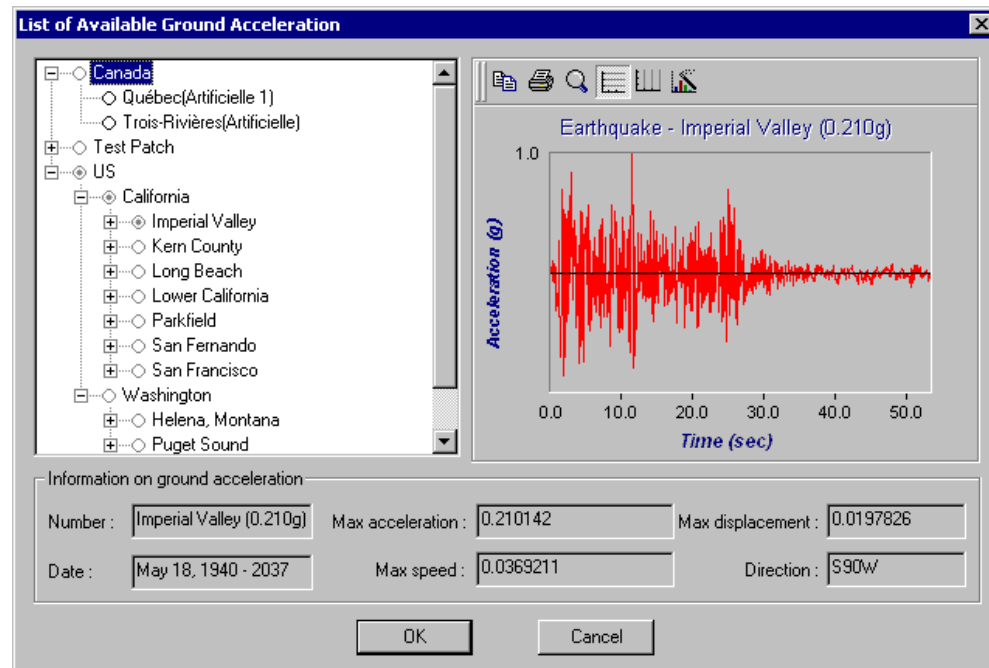
See also
Project Configuration
Spectral Analysis

Calculation of Stresses and Displacements

Displacements are calculated using the Duhamel integrals for coupled systems where the damping is modal (see Mario Paz page 69, page 205). The displacements and corresponding stresses are evaluated for each *Time pitch* chosen by the user, or for each accelerogram steps defined in the database (generally 0.02sec). After the stress calculation is done for each period, results are compiled to build the envelope.

See also
Time History Analysis
Spectral Analysis
Accelerograms

Linear Time History Analysis Procedure

- Select the **Analysis tab** of the **Project Configuration** and activate the radio button corresponding to a linear analysis.

- Select the **Seismic tab** of the **Project Configuration** and complete parameters for the spectral analysis because these parameters are also used to calibrate a linear time history analysis.

  - Choose an accelerogram that will be used for the linear time history analysis by clicking the button […] located next to the "Accelerogram" field. Enter appropriate values for *Duration* and *Time pitch* and check box *Save node displacements*.

- Generate load combinations using the **Load Combination Generator** and specify the inclusion of the linear time history envelope **Etl**. Ask for the generation of the "Mass" load combination. It should be selected in the **Modal Analysis** dialog box.

- Run a modal analysis.

- Open the **Linear Seismic Directions** spreadsheet and insert two lines. Give a name to each direction and specify direction vectors in columns dir. x, dir. y or dir. z.

  - Corresponding vibration modes are automatically initialized in the **Linear Seismic Directions** spreadsheet, according to the maximum modal contributions calculated by VisualDesign, and supplied in the **Frequencies and Vibration Modes** spreadsheet.

- Run a spectral analysis. Open the **Linear Seismic Directions** spreadsheet and make sure that you obtained at least 90% of participating mass for each seismic direction.

- Select the **Information on Levels** spreadsheet and look at interstory drift. Make sure that they do not exceed the limit permitted by the Code.

- Create a personalized selection for nodes that must be analysed. To do so, select nodes and go to **Edit** menu / **Select** / **Create a selection**. Give a name to this selection.

- Open the **Time History Analysis** dialog box. Select the name of the node selection in the "Pre-defined selection" list box and click the "Analyse" button.

- When analysis is completed, envelope Et will be automatically selected on Activation toolbar.

- Look at results by selecting the **Time History** heading under **Results** menu. Study the nodes displacements in time.

- Run a static analysis.

- Look at load combination and envelope results.

Non-Linear Time History Analysis

Pall Friction Dampers

Canadian researchers Pall and Marsh, developed the friction damping system in 1982. The systems main goal was to eliminate the problems caused by the degradation of the hysteretic loops of steel bracing. The system comprises of a simple mechanism containing mechanical brakes. These brakes are designed to not slip at service loads or during minor earthquakes. However, during an important earthquake, the brake slips at a predetermined load before a plastic deformation occurs in the structural element. The slipping of the brake absorbs the seismic energy in a mechanical way without plastic deformation of the bracing or other structural members.
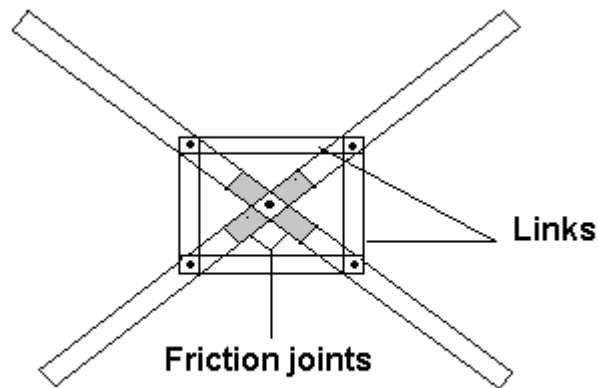
Friction Damping Systems

Many options are offered to the user regarding the use of friction dampers. The most common uses are illustrated in the figures below.

Friction Damper applied to a Bracing

CivilDesign Inc. Engineering Software
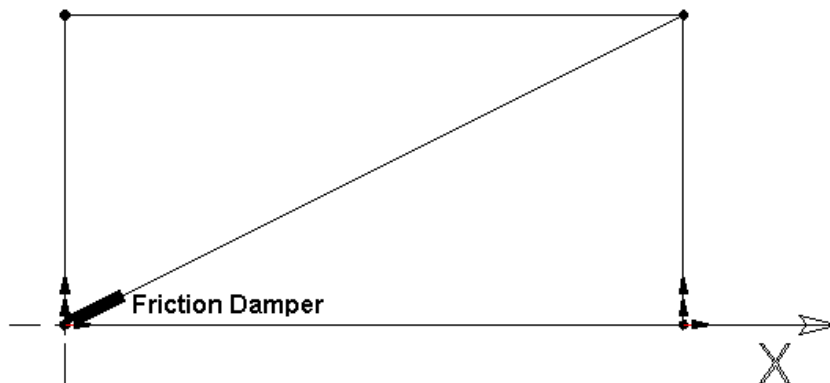
Friction Dampers applied to X-Bracings
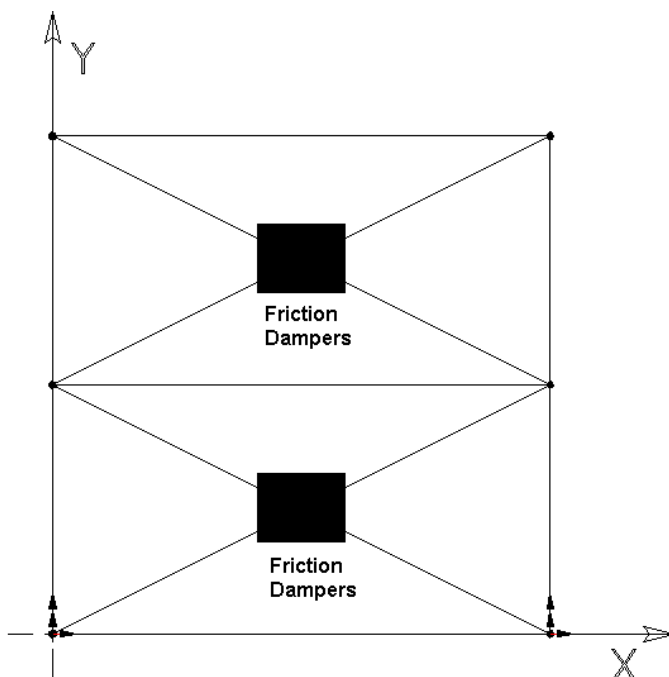
Friction Damper applied to a Chevron Bracing
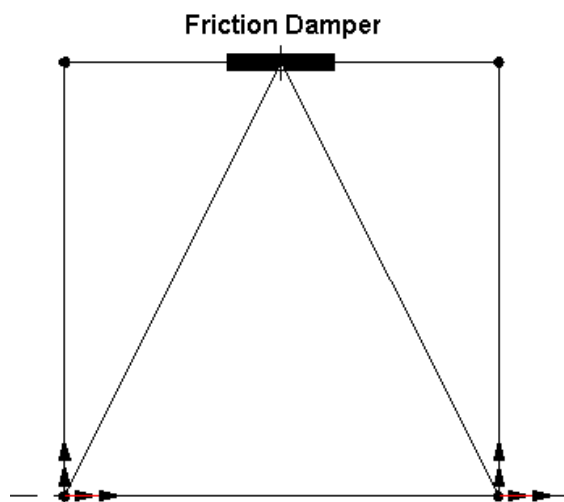
See also

Time History Analysis

Elastoplastic Properties

Non-linear Time History Analysis Procedure

Time History Results

Non-Linear Time History Analysis

The "Non-linear Time History Analysis" icon of Tools toolbar

The **Non-linear Time History** analysis allows modeling friction dampers and elastoplastic hinges for selected members. VisualDesign™ considers these members as full axially elastoplastic members.

Date and Time

VisualDesign now displays the date of analysis in the **Non-linear Time History 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
Pall Friction Dampers

Non-Linear Seismic Directions Spreadsheet

Go to **Loads / Seismic Directions / Non-linear** and insert lines. Give a name to each direction and specify the direction vectors.

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Dir. x	Global x component of seismic direction.	Single click
Dir. y	Global y component of seismic direction.	Single click
Dir. z	Global z component of seismic direction.	Single click
Vdyn	Lateral seismic stress exerted at the base of the structure, based on the dynamic analysis.	No
M modal / M	Percentage of modal mass that was considered.	No

Non-Linear Time History Analysis Procedure

- Select the **Analysis tab** of the **Project Configuration** and activate a non-linear static analysis. Specify a number of 200 iterations to speed up convergence. Complete parameters required for a non-linear analysis.

- Select the **Seismic tab** of the **Project Configuration** and complete the parameters required for a spectral analysis. Choose the base acceleration that will be used for the non-linear time history analysis. Specify a tolerance of 1 kN to speed up convergence.

- Run a modal and spectral analysis of the building and study the bracing behaviour (Maximum compression and tension.). Choose the location of elastoplastic members (PALL).

- Activate the "Structure" mode, select these members and open the **Member Characteristics** dialog box.

    - In the **Member** tab, specify these members as elastoplastic in the "Behaviour" field.

    - Then, select the **Behaviour** tab and complete the required parameters.

- Generate load combinations using the **Load Combination Generator** and specify the inclusion of the non-linear time history envelope **Etnl.**

- Define non-linear seismic directions in the **Non-linear Seismic Directions** spreadsheet.

- Run a non-linear static analysis.

- Select nodes from which you wish to obtain displacements in time.

- Run the non-linear time history analysis.

- Activate the "Load Combination" mode and select a load combination.

- Look at results by selecting the **Time History** heading under **Results** menu.

See also

Nodes Displacement in Time
Forces in Time
Reactions in Time
Forces and Displacement - Hysteretic loops
The Behaviour tab of Member dialog box
Non-linear Time History Analysis
Dynamic Analysis
Static Analysis

Time History Analysis Results

General

Study the displacements of nodes or the differential displacement between nodes by selecting **Time History/Nodes displacement in Time** under the **Results** menu. These values are calibrated according to V/Vdynamic ratio. Values do not take into account accidental torsional effects nor amplification caused by inelastic effects.

The stress diagrams for members and plates and support reactions are calibrated according to the V/Vdynamic ratio. Results include accidental torsion effects and the amplification caused by inelastic effects is considered.

See also

Accidental Torsion Effects

Stresses and Displacements Calculation

Nodes Displacement in Time

Reactions in Time

Forces in Time

Forces and Displacement - Hysteretic loops

Nodes Displacement in Time

The **Nodes Displacement in Time** function allows visualising the node displacement in time or the differential between two nodes for the **Time History** analysis. Simply specify the node(s) for which you need to know the displacement and press the read button to obtain the displacement or differential displacement graphically and numerically.

Field	Description
Selection of Nodes	
First	Indicate the node number to consult its displacement in time.
Second	Enter the second node number to obtain the differential displacement between nodes.
Direction	Enter a seismic direction
Results	
X, Y, Z	The min and the max node displacements are indicated, for each axis.

Field	Description
dx, dy, dz	The differential displacement between nodes is indicated for each axis.
Read button	When node numbers are specified, press the read button to obtain numerical and graphic results.
Spreadsheet button	Press the **Spreadsheet** button to obtain the results in a spreadsheet format.

Reactions in Time

This function allows visualising reactions present in a member versus the time. To access this function, select the "Time History" heading under **Results** Menu. Then, select the member in the combo box for which you want to see the **Reactions in Time** diagram and press the "Reading" button at the bottom of the page.

Forces in Time

This function allows visualising the forces present in a member versus time. Select the "Time History" heading under **Results** Menu. Then, select the member for which you want to see the **Forces in Time** diagram in the combo box. Press the "Reading" button at the bottom of the page. The diagram is then displayed.

Forces and Displacement - Hysteresis Loops

This function allows visualising hysteretic loops for a selected member. Select the "Time History" heading under **Results** Menu. Then, select the member number in the combo box. Press the "Reading" button at the bottom of the page. Calculated hysteretic loops, forces and displacement are then displayed.

Hysteresis Loops

A hysteretic loop is a force-displacement or moment-rotation cyclic curve that defines an element or structural system behaviour in the elastic range. (Ref. *Éléments de génie parasismique et de calcul dynamique des structures*, André Filiatrault, 1996)

CivilDesign Inc. Engineering Software
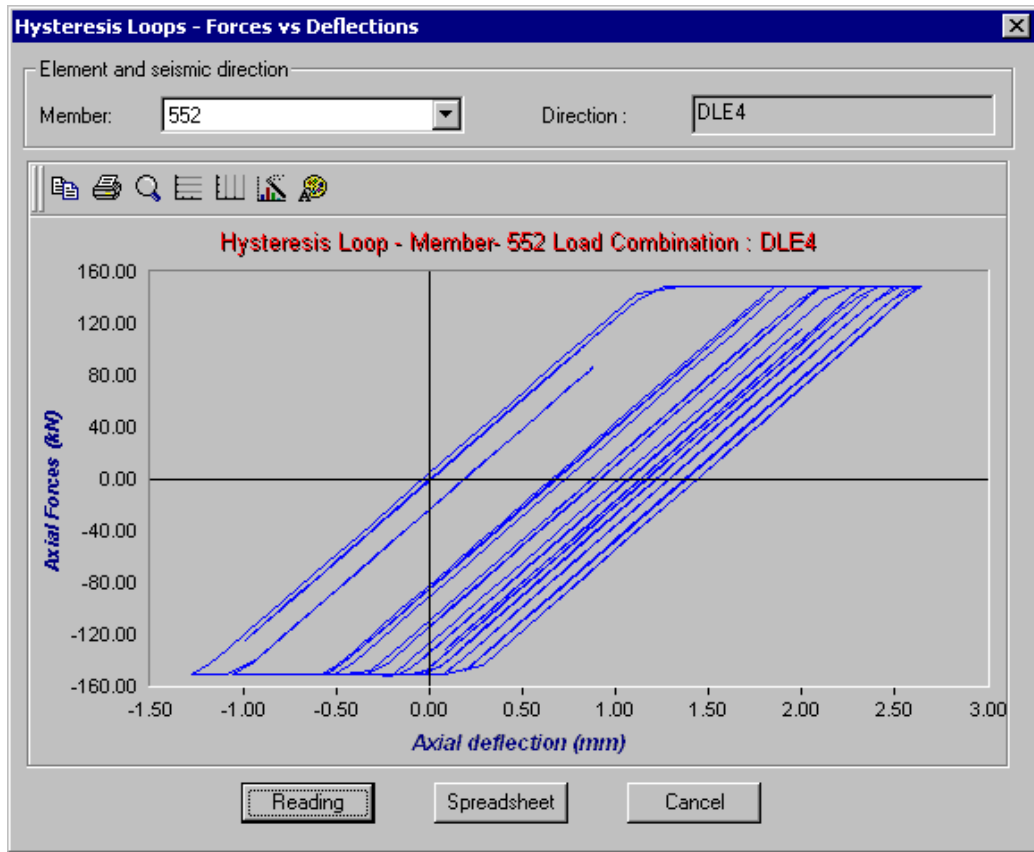
General Dynamic Analysis

General

A general dynamic analysis represent any dynamic analysis that is created with loads other than those created from a ground motion such as an earthquake. This type of analysis may be transient or harmonic, for example.

VisualDesign™ has several types of loadings that can be chosen by the user, namely:

- Sine

- Cosine

- Half-sine + step:

- Triangle + step:

- Rectangle + step:

- Trapezoidal + step:

After having chosen the accelerogram that will fit with the type of vibration, the engineer will indicate the number of cycles that will be applied to the structure and also the time intervals dt1, dt2, etc. through a spreadsheet. The accelerogram starting time may be specified if, for example, the analysis is carried out using two accelerograms (one in each direction) that will be applied one after the other. The engineer can also modify the damping in the **Frequencies** spreadsheet that is available after the dynamic analysis is over.

The accelerogram that is defined in a spreadsheet has a unitary value. The user will define the amplitude through the appropriate Loads dialog box. The accelerogram may be applied to nodes (concentrated load) or floors made of plates (distributed load).

When viewing the results through the **Node Displacement in Time** (time history analysis results), the engineer must verify the deflection created by this dynamic load. Actually, VisualDesign™ gives the node displacement for the dynamic load only. The deflection due to static loads such as dead load will be added in a later version.

See the topic "Procedure to Run a General Dynamic Analysis" to get systematic procedure.

See also
Loads Definition Spreadsheet
General Dynamic Load Diagrams
Procedure to Run a General Dynamic Analysis

Dynamic Load Diagrams

Before running a general dynamic analysis, the user must choose an accelerogram in the **General Dynamic Loads Diagrams** spreadsheet that is available under the **Loads** menu. Accelerograms have a unitary value. In the spreadsheet, the engineer must specify the time intervals dt1, dt2, etc. and the number of cycles that will be applied on the structure.

Afterwards, the accelerogram will be selected in the **Loads Definition** spreadsheet. The accelerogram starting time may also be specified in this spreadsheet.

Note **Time pitch** must be specified in the **Analysis** tab of **Project Configuration** dialog box. dt1, dt2, etc. values must all be greater than the time pitch in order to obtain interesting results.

Available Dynamic Loadings

SINE:
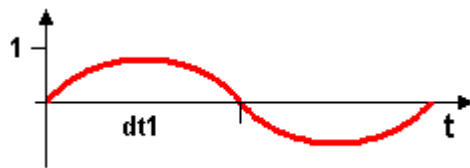
COSINE:
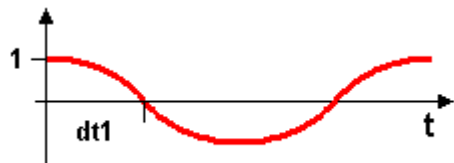
HALF-SINE + STEP:
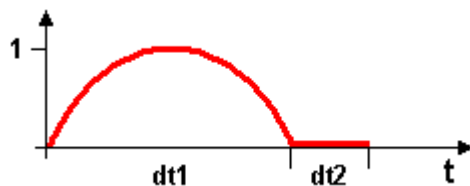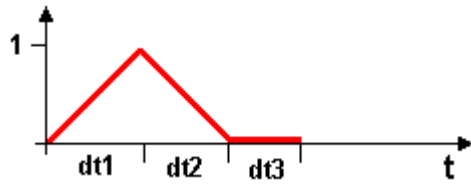
TRIANGLE + STEP:

RECTANGLE + STEP:
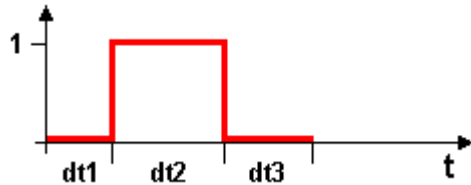
TRAPEZOIDAL + STEP:
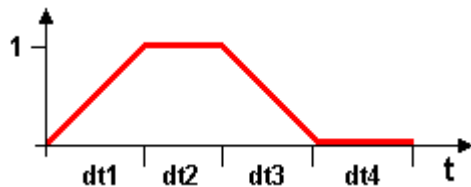
Group: General Dynamic Analysis

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Type	Among the drop-down list box, choose the type of accelerogram that will be applied to the structure	Double click
Number of cycles	Indicate the number of cycles that you wish to apply.	Single click
dt1	Specify the time interval dt1.	Single click
dt2	Specify the time interval dt2.	Single click
dt3	Specify the time interval dt3.	Single click
dt4	Specify the time interval dt4.	Single click

Note dt1, dt2, etc. values must all be greater than the time pitch in order to obtain interesting results.

See also

Load Definitions

Open the **Load Definitions** spreadsheet and specify a "Dynamic" type of load. Then, go to the **Dynamic** tab and fill in required parameters. If more than one dynamic load is applied, specify the start time for each.

There is no limit to numbers of dynamic loads.

Use the **Load Combination Generator** to generate load combinations that will include this dynamic load.

Applying the Dynamic Load to the Structure

- Activate the "Load Case" mode on the Activation toolbar.

- Choose the name of the general dynamic load among the drop-down list box on the same toolbar.

- Activate the element icon on Elements toolbar.

- Double-click the element to call up the **Loads** dialog box (or click the element once and click the **Properties** icon). Enter the numerical value for the concentrated or distributed load. Click OK.

Procedure to Run a General Dynamic Analysis

- Define the *Duration* and *Time pitch* variables in the **Analysis** tab of **Project Configuration** dialog box. Tick off the "Save Node Displacements" box to save the displacement results.

- Open the **General Dynamic Load Diagrams** spreadsheet (**Loads** menu). Insert a line and select a type of accelerogram among the list box. Give it a name and specify the number of cycles that will be applied to the structure. Finally, enter the time intervals dt1, dt2, etc. Please note that time intervals must all be greater than the time pitch.

- Select the **Loads Definition** spreadsheet under the **Loads/Load Cases** menu:

  - In the **Load Cases** tab, select a *Dynamic* type of load in the "Type" column.

  - Select the **Dynamic** tab and select the general dynamic loading that you defined. In the column "Starting Time", enter the accelerogram starting time, if there is more than one dynamic load to be applied.

CivilDesign Inc. Engineering Software

- To apply the general dynamic loads on your structure:

    - Activate the "Load Case" mode on the Activation toolbar.

    - Choose the name of the general dynamic load among the drop-down list box on the same toolbar.

    - Activate the element you want to load, on Elements toolbar.

    - Double-click the element to call up the Loads dialog box (or click on the element and press the "Properties" icon). Enter the numerical value for the concentrated or distributed load. Click OK.

- Use the **Load Combination Generator** to generate load combinations as per selected code. Ask for the "Mass" load combination, which includes all dead loads plus 25% of snow load. This load combination should be selected in the Modal Analysis dialog box.

- Open the Modal Analysis dialog box by pressing icon $\boxed{\omega}$. Select the "Mass" load combination and click the "Analyse" button.

- If you wish to modify the damping, select the **Frequencies and Vibration modes** spreadsheet in the **Results** menu under **Modal/Spectral** heading.

- Run a static analysis. You may select only the nodes that you wish to study before launching the static analysis in order to reduce the time of analysis. Results (**Node Displacement in Time**) will include selected nodes only.

- To look at the results, select the heading **Node Displacement in Time** under the menu **Results**/**Time History**. Enter the node number in the appropriate field in the appearing dialog box and look at the displacement diagram. Press the "Spreadsheet" button to access the results spreadsheet.

Note   The deflection that is shown on the diagram, for a given load combination, is calculated for the dynamic load only. For the moment, it does not include the deflection created by static loads such as the structure dead load. Verify the allowable deflection permit by the code.

See also
General Dynamic Analysis
Loads Definition spreadsheet
General Dynamic Load Diagrams
Time History Analysis Results

MOVING LOAD ANALYSIS

TABLE OF CONTENTS

Chapter 8 Moving Load Analysis

General

Moving Load Analysis

The **Moving Load** analysis module computes the internal forces generated by moving loads such as trucks, trains or crane runways. The module was developed in such a way that it allows the application of customized trucks and design codes. In fact, engineers can use standardized loadings, as per codes, or they can define different scenarios with custom moving load arrangements.

Three mobile axes are available if you want to study forces and displacements induced by trucks that are moving on different axes.

Up to 10 envelopes of results can be generated in a single analysis. In addition, you can select the concomitant values (Mx, Vy, Nz, etc.) that you want to include in the results spreadsheet, for each results envelope. These concomitant values are selected in the **Moving Load Analysis** dialog box, which appear on screen once that you launch the analysis.

See also
Available Trucks
Nomenclature of Trucks
Axles/Wheels Tab
Definition of Moving Loads
Moving Load Cases Generator
Moving Load Cases Spreadsheet
Imbalance Factor
2D Axle Factors
Bridge Spans Factors Tab
The Moving Load Cases Components Tab
The Moving Load Analysis Dialog Box
Moving Load Analysis Procedures
Moving Load Axes
Definition of Moving Load Envelopes
Moving Load Analysis and Culvert Design
Copying a Moving Load Case along with Components

Moving Load Axes

Three moving load axes can be defined in VisualDesign within a single project. Axes can be located on members, along plates (one side), and along floors (one side). The moving load axis must be selected in each of these elements dialog box. The side of the floor or plate must also be specified as the location of the moving load axis.

Then, when moving load cases are defined (**Loads / Moving Loads / Moving Loads Cases,**) a moving load axis must be selected for each moving load case. If you do not want to include one particular case in a moving load envelope, simply select the option *Not required* in the column titled "Moving Load Axis".

Display moving load axes through the **Attributes** tab of **View Options** dialog box.

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.
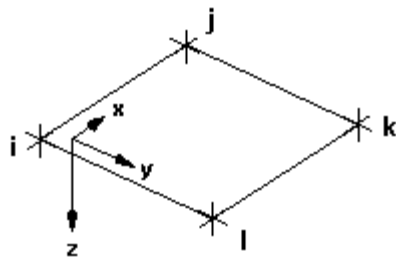
Plate Local Axis System
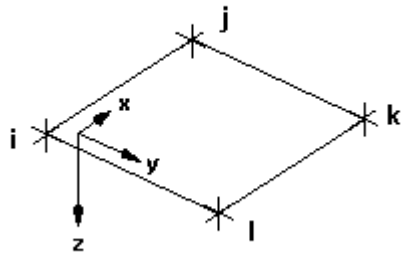
Point (0,0,0) corresponds to the plate's Node *i*.

Floor Local Axis System

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

Floor Local Axes System

Local axes are displayed inside the floor, near node I, which is corresponding to point (0,0,0).

Trucks' Nomenclature

Mobiles that are included in the **Trucks** tab, are described as follows:

[2D] – QS-660: 2D structural model using a QS-660 truck.

[3D] – CL1-625 – 0.30: 3D structural model using a CL-625 truck, Level 1, and DLA factor of 0.3.

[3D] – CL123-625 – 0.30: 3D structural model using a CL-625 truck, Level 3, and DLA factor of 0.3.

For a 2D or 3D project, you must include all moving load cases that correspond to a given truck (Ex. CL-W) with different DLA factors so that all cases are covered in the analysis, as per clause 3.8.4.5 of S6-00 standard. To quickly generate all required moving load cases, use the **Moving Load Case Generator**.

See also
The Trucks Tab
Moving Load Case Generation Wizard

Position of Truck Axles and Wheels

Location of wheels on mobile local axes
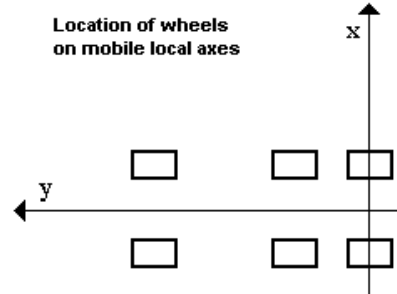
Location of Axles

Ze = Location of front axle on the mobile Z axis

E = Location of no i axle on the mobile Z axis

$y0 = 0$

$y2$

$y1$

$E2 = Ze - y2$

$E1 = Ze - y1$

$E0 = Ze - y0$

Z
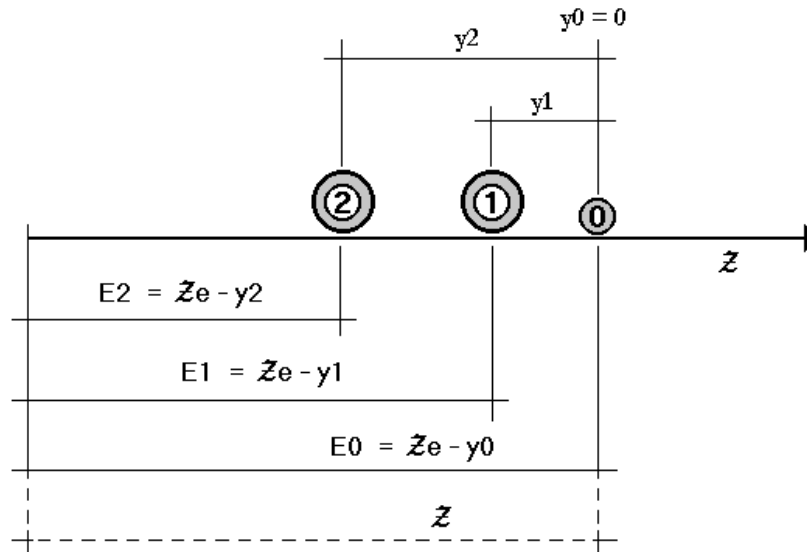
See also

The Trucks Tab
The Axles/Wheels Tab

Imbalance Factor

Ref: "Centrifugal Force", clause 3.8.5 of CAN/CSA-S6-00 Standard.

The Imbalance factor is dependent of the truck speed and the curve radius. According to Code S6-00, this factor is equal to:

Imbalance factor:
$$\frac{V^2}{127 \cdot r}$$

Where V id the truck speed in the curve (km/hour) and r, the curve radius (m).

From this imbalance factor, VisualDesign will calculate the imbalance loads on each side of the truck, due to the centrifugal force. It will multiply this factor by H/L where H is equal to 2.0 and represents the distance from ground to the centre of gravity of truck, and L, the distance centre-to-centre of right and left wheels (Usually L is 1.8 m). The value of load imbalance is usually between zero and 0.5. Its default value is zero.

Sign Convention:
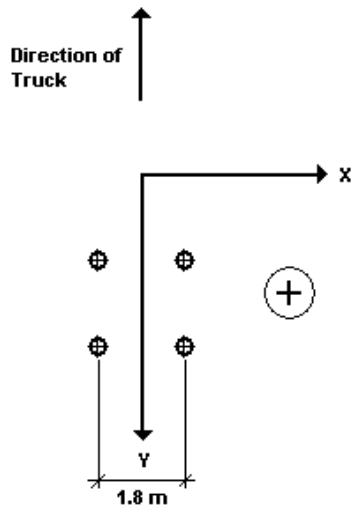
If the truck turns left:
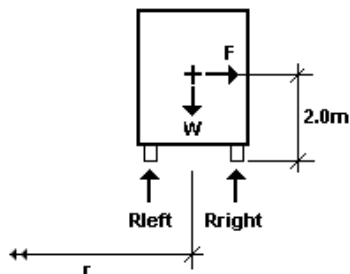
When the truck moves on a straight road, Rleft and Rright are equal to W/2.

Centrifugal Force is equal to:

$$F = \alpha \cdot W$$

The sum of moments relatively to Rleft (positive counter clockwise) is equal to zero:

$$Rd \cdot 1.8 - W \cdot \frac{1.8}{2.0} - F \cdot 2 = 0$$

$$Rd = \frac{W}{2} + \frac{2 \cdot F}{1.8}$$

$$Rd = W \cdot \left(\frac{1}{2} + \frac{2 \cdot \alpha}{1.8} \right)$$

$$Rd = \frac{W}{2} + \frac{2 \cdot \alpha \cdot W}{1.8} \qquad \text{and} \qquad Rg = W - Rd$$

$\dfrac{2 \cdot \alpha}{1.8}$ represents the load imbalance due to centrifugal force. Alpha is the Imbalance factor and is equal to $\dfrac{V^2}{127 \cdot r}$.

Therefore, the load imbalance is = $\dfrac{H}{L} \cdot \dfrac{V^2}{127 \cdot r}$

See also

The Moving Load Cases tab

Pre-Defined Trucks

Definition of Moving Loads

Select **Trucks** under **Common** menu to have access to the **Moving Loads** dialog box. It is composed of two spreadsheets: **Trucks** and **Axles/Wheels**. The Trucks spreadsheet includes a list of predefined trucks (2D & 3D) that are described in several standards (CAN/CSA-S6-88, S6-00 and AASHTO LRFD-04). To know the weight and the number of axles that are considered for each truck listed in the spreadsheet, select the **Axles/Wheels** tab.

Trucks that are used in Ontario and Saskatchewan are also included for S6-00 standard. For more details, refer to:

Moving Loads - CAN/CSA-S6-00
Moving Loads - AASHTO-LRFD-04

Each truck represents a different moving load case according to position of axles and the dynamic load allowance (DLA) factor. For example: refer to clause 3.8.4.5 of S6-00 standard.

Do not modify data in this spreadsheet. To add new data, insert a line at the end of the spreadsheet and define parameters for the new truck.

See also
The Trucks Tab
The Axles/Wheels Tab

Moving Loads - CAN/CSA-S6-00

The following tables include a list of predefined trucks based on standard CAN/CSA-S6-00. They are listed in the **Trucks** spreadsheet, which is available in the **Common** menu under **Trucks**.

Use the **Moving Load Case Generation Wizard** to quickly create all the cases that you need according to a selected code or standard and according to a selected layout of trucks over the bridge.

Design – Ultimate Limits States (Envelope Lm01)
(With lane overload: factor α = 80% and variable DLA)

2D Trucks	3D Trucks
2D Axle factors with multiple lanes	
[2D]-CL1-625-25	[3D]-CL1-625-25
[2D]-CL123-625-30a	[3D]-CL123-625-30a
[2D]-CL1-625-30b	[3D]-CL1-625-30b
[2D]-CL12-625-30c	[3D]-CL12-625-30c
[2D]-CL12-625-40	[3D]-CL12-625-40

Bridge Evaluation: Normal Level and Level #1 (Envelope Lm01)
(With lane overload: factor α = 80% and variable DLA)

2D Trucks	3D Trucks
2D Axle factors with multiple lanes	
[2D]-CL1-625-25	[3D]-CL1-625-25
[2D]-CL123-625-30a	[3D]-CL123-625-30a
[2D]-CL1-625-30b	[3D]-CL1-625-30b
[2D]-CL12-625-30c	[3D]-CL12-625-30c
[2D]-CL12-625-40	[3D]-CL12-625-40

Bridge Evaluation: Levels 2 & 3 (Envelope Lm01)
(With lane overload: factor α = 80% and variable DLA)

Level 2

2D Trucks	3D Trucks
2D Axle factors with multiple lanes	
[2D]-CL2-625-25	[3D]-CL2-625-25
[2D]-CL123-625-30a	[3D]-CL123-625-30a
[2D]-CL12-625-30c	[3D]-CL12-625-30c
[2D]-CL12-625-40	[3D]-CL12-625-40

Level 3

[2D]-CL123-625-30a	[3D]-CL123-625-30a
[2D]-CL3-625-40	[3D]-CL3-625-40

Fatigue and Deflection (Envelope Lm02)
(No lane overload and variable DLA)

2D Trucks	3D Trucks
2D Axle factors with a single lane	
Use one of the above lists according to the selected level for bridge design or evaluation.	Use one of the above lists according to the selected level for bridge design or evaluation.

See also

Moving Load Case Generation Wizard
Moving Load Cases Spreadsheet
Load Combinations and Moving Load Envelopes
Moving Load Envelopes spreadsheet

CivilDesign inc. Engineering Software

Moving Loads - AASHTO-LRFD-04

The following tables include a list of predefined trucks based on standard AASHTO-LRFD-04. They are listed in the **Trucks** spreadsheet, which is available in the **Common** menu under **Trucks**.

Use the **Moving Load Case Generator** to quickly create all the cases that you need according to a selected code or standard and according to a selected layout of trucks over the bridge.

Ultimate Limits States (Envelope Lm01)
(With lane overload, no factor α, and DLA = 0.33)

2D Trucks	3D Trucks
2D Axle factors with multiple lanes	
Design truck:	Design truck:
[2D]-HL93Tr 43	[3D]-HL93Tr 43>>
[2D]-HL93Tr 67	[3D]-HL93Tr 67>>
[2D]-HL93Tr 90	[3D]-HL93Tr 90>>
[2D]-HL93TrTr	[3D]-HL93TrTr >>
Design Tandem:	Design Tandem:
[2D]-HL93Ta	[3D]-HL93Ta

Deflection (Envelope Lm03)
(With lane overload, factor α = 25%, and DLA = 0.33)

2D Trucks	3D Trucks
2D Axle factors with multiple lanes	
[2D]-HL93dTr 43	[3D]-HL93dTr 43

Fatigue (Envelope Lm02)
(No lane overload, and DLA = 0.15)

2D Trucks	3D Trucks
2D Axle factors with a single lane	
[2D]-HL93Tr 90	[3D]-HL93Tr 90

See also
Moving Load Cases Spreadsheet
Moving Load Case Generation Wizard
Load Combinations and Moving Load Envelopes
Moving Load Envelopes spreadsheet

CivilDesign inc. Engineering Software

The Trucks Tab

This spreadsheet, which is accessible through **Common/Trucks**, includes a list of predefined trucks that are described in several standards (CAN/CSA-S6-88, S6-00 and AASHTO LRFD-04). S6-00-trucks that are used in Ontario and Saskatchewan are also included.

Each moving load represents a different moving load case, according to the position of wheels and axles and the dynamic load allowance.

Create New Trucks

In the **Trucks** tab, insert lines at the end of the spreadsheet and enter your own parameters. Then, select the **Wheel/Axle** tab and specify the weight and position of each axle composing this truck.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters describing the type of truck. For more details, go to topic Nomenclature of trucks	Single click
Total W	Total weight of truck	Single click
Dyn. Load All. (Truck)	Dynamic Load Allowance for truckload moving alone, according to clause 3.8.4.5 (S6-00 standard).	Single click
Dyn. Load All. (Truck/Lane)	Dynamic Load Allowance for truckload with a uniformly distributed lane load.	Single click
α	Factor applied to a moving load when superimposed to a uniformly distributed lane load, according to clause 3.8.3.2 (S6-00 standard). See Notes below.	Single click
Overload	Uniformly distributed lane load (Linear load).	Single click
Dyn. Load All. (W Lane)	Dynamic Load Allowance applied to uniformly distributed lane load.	Single click
Lane Width	Lane width	Single click
Remove Axle	If truck axle can be removed to maximize the moving load response, choose option [x]. Refer to clause 3.8.4.1a) of S6-00 standard.	Double-click or Space bar

FACTOR TO APPLY TO AXLE LOAD WHEN A UNIFORMLY DISTRIBUTED LANE LOAD IS CONSIDERED:

VisualDesign will take the worst load case of a) or b):

a) Mobile * (1 + DLA Truck)

b) ((Mobile * α) (1 + DLA Truck/Lane)) + Overload * (1 + DLA W Lane)

For the CAN/CSA-S6-88 Standard: α = 0.6

For the CAN/CSA-S6-00 Standard: α = 0.8

For AASHTO LRFD-98 Standard: α = 1.0

See also
Moving Loads spreadsheet
The Axles/Wheels Tab
Copying a Moving Load along with its Attributes

The Axles/Wheels Tab

This tab is part of the moving load dialog box (**Common/Trucks**). It supplies the weight and position of each axle.

Group: Truck: (title)

Column	Description	Editing
ID	Calculated automatically	No
Coord. X	X-coordinates	Single click
Coord. Y	Y-coordinates	Single click
W / Wtotal	Weight of axle or wheel acting on this location. The value can be expressed as a ratio of total truck weight. To modify units, select the column title, right click and choose the function **Change Units** in contextual menu.	Single click

See also
Moving Load Dialog Box
The Trucks Tab

Copying a Moving Load along with Axles/Wheels

To simultaneously copy a moving load along with its corresponding axles/wheels information, use the **Duplicate** function, available in the spreadsheet's contextual menu. This tool is useful when defining new trucks.

Procedure:

- Open the **Moving Load Definition** spreadsheet.

- Select the line that corresponds to the moving load you want to copy.

- Right click and choose function **Duplicate** in contextual menu.

See also
Duplicate function

Axle Factors for 2D Models

2D Dynamic Load Allowance Factors

You must define axle factors that will be applied to different forces transmitted to spans and supports of a 2D bridge model.

Select **Moving Loads** under **Loads** menu to have access to the **2D Axle Factors** dialog box. It is composed of two tabs: Span and Support.

Calculation of Axle Factors According to S6-00 Standard:

Use the tables included in section 5.7.1 and calculate Fv and Fm. Then, from these values, calculate Vg and Mg along with axle factors.

In VisualDesign, the axle factor, Fa, will multiply the total maximum force (shear and bending moment) as follows:

Vg = Fa *Vt, where Vt is the maximum shear force per lane acting on a section of the studied span.

And

Mg = Fa * Mt where Mt is the maximum bending moment for a sole truck on a section of the studied span.

See also
The Bridge Spans Factors Tab
The Supports Factors Tab

The Bridge Span Tab

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Bridge span number.	Single click
2 Lanes or More		
Mx+	Axle factor for positive moment, on strong axis.	Single click
Mx –	Axle factor for negative moment, on strong axis	Single click
Vy	Axle factor for shear forces, on strong axis	Single click
My, Vx, Nz, Tz	Axle factor for other types of forces such as My, Vx (weak axis), Nz, and Tz.	Single click

Column	Description	Editing
Displacements	Axle factor for strong axis (u) and weak axis (w) displacements.	Single click
Single Lane		
Mx+	Axle factor for positive moment, on strong axis.	Single click
Mx –	Axle factor for negative moment, on strong axis	Single click
Vy	Axle factor for shear forces, on strong axis	Single click
My, Vx, Nz, Tz	Axle factor for other types of forces such as My, Vx (weak axis), Nz, and Tz.	Single click
Displacements	Axle factor for strong axis (u) and weak axis (w) displacements.	Single click

NOTE   When two different load factors are applied to the same node, VisualDesign chooses the biggest one.

See also
Calculation of 2D Axles Factors
Definition of Moving Load Envelopes
The Supports Factors tab
2D Axle Factors Dialog Box

The Supports Tab

Group: Load case Data

Column	Description	Editing
ID	Calculated automatically	No
Number	Support no.	Single click
2 Lanes or More		
Moment	Axle factor applied to Mx, My, and Mz.	Single click
Reaction	Axle factor applied to Rx, Ry, and Rz	Single click
Single Lane		
Moment	Axle factor applied to Mx, My, and Mz.	Single click
Reaction	Axle factor applied to Rx, Ry, and Rz	Single click

NOTE   When two different load factors are applied to the same node, VisualDesign chooses the biggest one.

See also

Calculation of 2D Axles Factors
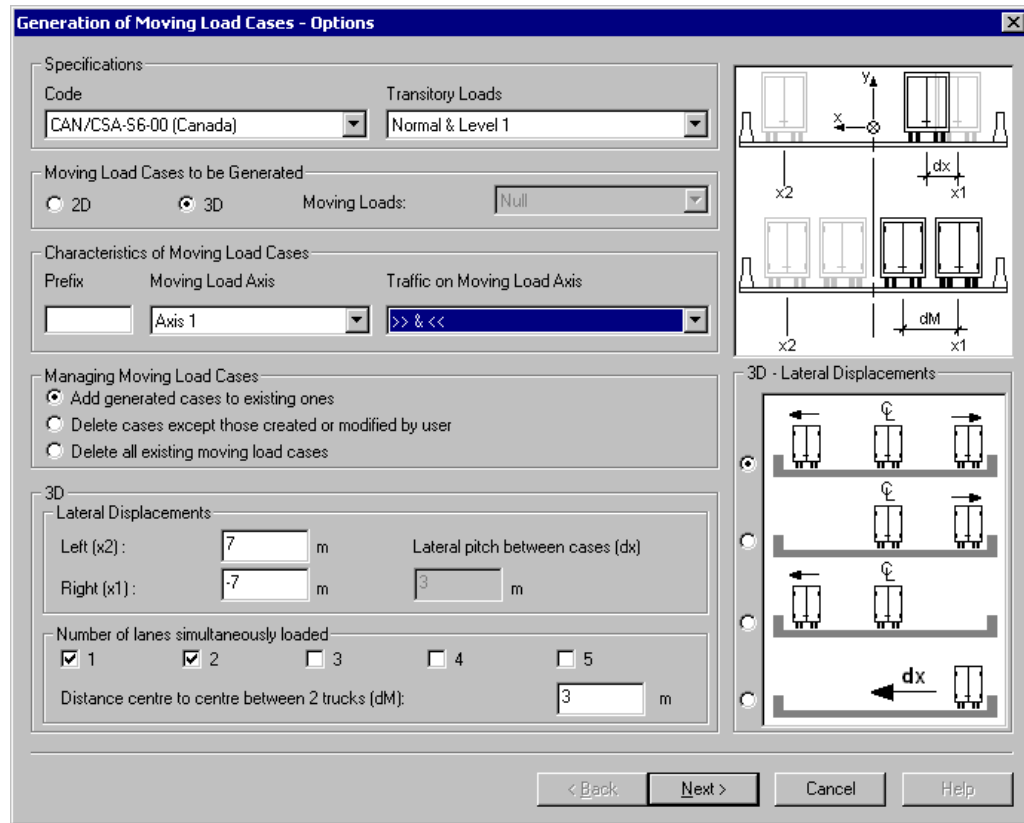Definition of Moving Load Envelopes
2D Axle Factors Dialog Box
The Bridge Spans Factors Tab

Moving Load Cases

Moving Load Case Generator

The generator allows getting all the required moving load cases, in accordance with a selected code and specified lanes and truck offsets over the bridge.

This generator is available through the **Loads** menu under **Moving Load Cases** / **Automatic Generation**. The first page is called **Options**:

CivilDesign inc. Engineering Software

Here is a description of fields included in this page:

| Field	Description
Specifications	
Code	Select the code or standard that will be used for the design and for the generation of required moving load cases.
Transitory Loads	Specify the loads to be used in this project: Normal & Level 1 = Design Level 1, 2, or 3 = Bridge Evaluation Alternative = Select the moving load to be used in the generation in the "Moving Loads" list box below.
Moving Load Cases to be Generated	
2D or 3D	Activate the radio button that represents the modeling of your project. N. B. For a 2D project, you must define axles factors and apply them to spans and supports. Refer to 2D Axles Factors spreadsheet.
Moving Loads	If alternative transitory loads have been specified, select a moving load among the list.
Characteristics of Moving Load Cases	
Prefix	If needed, specify a prefix that will be used for the generated moving load cases.
Moving Load Axis	Select the axis from which moving load cases will be generated. The local axis system of moving load axis is represented by the X- and Y-axis, on the image above.
Traffic on Moving Load Axis	Specify the direction of traffic on this axis: Forward [>>], Backward [<<] or both directions [>> & <<].
Managing Moving Load Cases	**Add generated cases to existing ones**: The generated moving load cases will be added to existing ones in the Moving Load Cases spreadsheet. **Delete cases except those created or modified by users:** Only the cases that were automatically generated will be deleted. **Delete all existing moving load cases**: All cases will be deleted in the Moving Load Cases spreadsheet before adding the new generated moving load cases.

Field	Description
This section applies to 3D project only	
3D – Lateral Displacements	Activate the radio button that corresponds to the generation you want to get. Four layouts are represented in the image.
	Option 1: Cases will be generated from the far left through the far right of the bridge.
	Option 2: Cases will be generated from moving load axis to the far right of the bridge.
	Option 3: Cases will be generated from moving load axis to the far left of the bridge.
	Option 4: Cases will be generated from right to left, using the lateral pitch "dx" as the distance between two cases, instead of the 3m distance between two trucks.
3D	
Lateral Displacements N. B. The distance x2 is positive and x1 is negative, according to the local axis system of moving load axis.	**Left (x2)**: According to the image above, enter the position, x2, of trucks at the far left of moving load axis. VisualDesign will generate cases from this position.
	Right (x1): According to the image above, enter the position, x1, of trucks at the far right of moving load axis. VisualDesign will generate cases with respect to this position.
	Lateral pitch between cases (dx): Specify the lateral pitch that will be used between two cases. VisualDesign will generate intermediate cases considering positions x1 and x2, and this lateral pitch.
Number of lanes simultaneously loaded	Check the number of boxes that correspond to the number of simultaneously loaded lanes.
	Distance c/c between 2 trucks (dM): By default, this distance is 3m. This distance will be added to truck offsets when creating moving load cases.

Press the "Next" button to get to the second page of the Wizard, titled **Truck Offsets and Selection**.

In the left part of the dialog box, you will find truck positions (offsets) for each loaded lanes according to local axis system of moving load axis. In the cases shown below, two lanes are simultaneously loaded.

In the right part of the dialog box, we can see that these offsets will be applied to five moving load cases because they are highlighted. To withdraw a moving load case, click once with the cursor to cancel the selection.
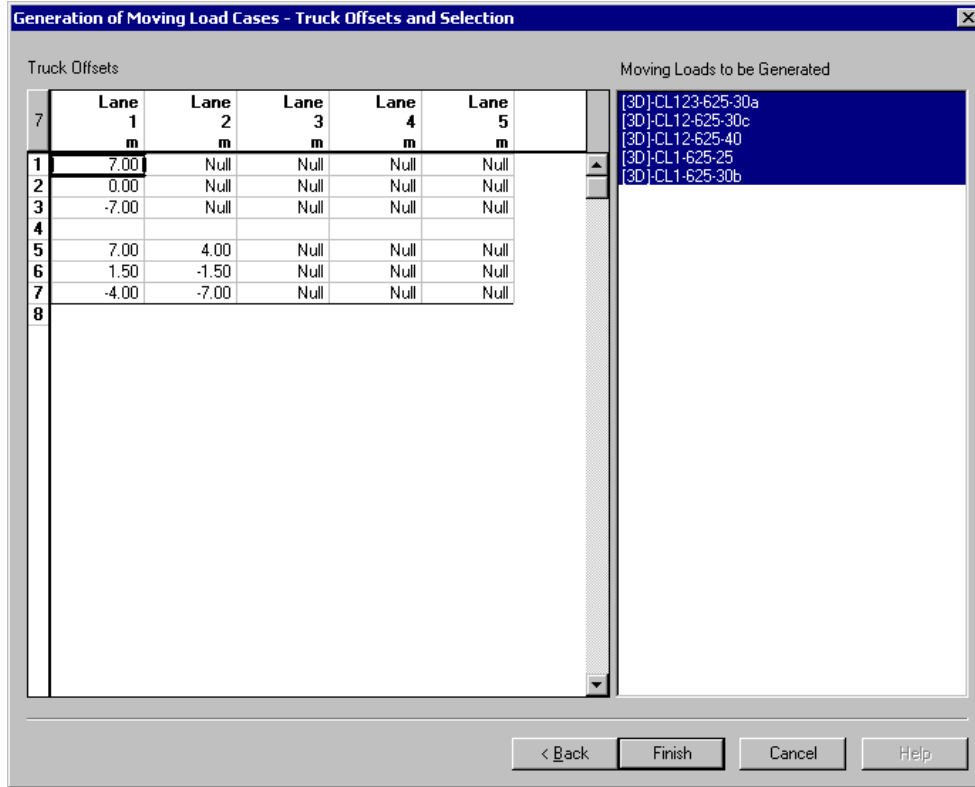
Press the "Finish" button to generate moving load cases.

The **Moving Load Cases** spreadsheet will be open and completed.

See also
The Moving Load Cases Spreadsheet

Moving Load Cases Dialog Box

The dialog box is accessible in the **Loads/ Moving Loads / Moving Load Cases.** It is completed if the **Moving Load Case Generator** was used to generate all required moving load cases.

The dialog box is composed of two tabs: **Moving Load Cases** and **Moving Load Case Components**.

The Moving Load Cases Tab

In this tab, you will find all moving load cases that you need to analyse a 2D or 3D project.

EX. 2D PROJECT WITH S6-00 STANDARD:

Include all [2D] CL-W trucks to cover all cases, as described in clause 3.8.4.5. According to a chosen inspection level, you will have the following cases:

[2D]-CL1-625-25	(All axles)
[2D]-CL123-625-30a	(Axles 1, 2 and 3 with an axle removed)
[2D]-CL1-625-30b	(Axles 4 and 5)
[2D]-CL12-625-30c	(Axles 3 and 4)
[2D]-CL12-625-40	(Axle 4 only)

If you are defining moving load cases "by hand" instead of using the Moving Load Case Generator, include these 10 moving load cases in the **Moving Load Cases** spreadsheet, for envelope *Lm01*, as an example.

For the *Fatigue* envelope *Lm02*, you need to repeat these cases but without considering the overload.

Group: Load case data

Column	Description	Editing
ID	Calculated automatically	No
Number	12 alphanumeric characters	Single click
Mobile	Choose a moving load from the list. (Go to Nomenclature of trucks for more details.)	Double-click
Envelope	Specify which envelope corresponds to this moving load case: Lm1 to Lm10.	Double-click
Moving Load Axis	Specify which axis applies to this moving load case. If there is one axis only, choose *Mobile Axis 1*. See topic Moving Load Axes	Double-click
Traffic on axis	Direction of traffic on this axis: Forward [>>], Back [<<] or both directions [>> and <<].	Double-click
DLA (Mobile)	Dynamic Load Allowance factor applied to the moving load when used alone. For Standard S6-88 you should indicate the appropriate DLA factor generating the first natural frequency mode of the bridge. For the S6-00 standard, do not change the suggested factors.	Single click

Column	Description	Editing
DLA (Mobile/Lane)	Dynamic Load Allowance factor applied to the moving load when using a lane load that is evenly distributed. For Standard S6-88 you should indicate the appropriate DLA factor according to the first natural frequency mode of the bridge. For the S6-00 standard, do not change the suggested factors (the value must be 0.0).	Single click
Imbalance Factor	Based on clause 3.8.5 "Centrifugal Force" of S6-00 Code. Go to topic Imbalance Factor for more details.	Single click
Add Overload	If you want to add overload in this load case, choose option [x].	Double-click or Space bar
Definition	Definition of this moving load case.	Single click
Culvert Design		
Use Elevation	Specify if moving loads be applied over a buried arched culvert or box-culvert.	Double-click
Elevation	Enter the elevation where moving loads will be applied. VisualDesign will transform concentrated moving loads into surface loads according to this elevation.	Single click
Longitudinal Slope (%)	Longitudinal slope of the surface where moving loads will be applied.	Single click

See also

The Imbalance Factor
The Moving Load Cases Dialog Box
The Moving Load Cases Components tab
Copying a Moving Load Case along with Components

Buried Structure (Culverts)

You are allowed to consider trucks passing over a surface that is above your structural model. This can happen if the slab has not been defined in your model or if the structure is buried. VisualDesign will transform concentrated moving loads into distributed loads through the thickness that you will specify in the **Moving Load Case**s spreadsheet.

The Moving Load Case Components Tab

Group: Scenario: (title)

Column	Description	Editing
ID	Calculated automatically	No
Factor	Factor of the moving load which will be applied to the end-bearing loads and the lane overload selected in the scenario spreadsheet. In 3D, this factor will be used as the "Modification factor" when several lanes are loaded. In 2D, this factor can be used as the "Axle Factor", only for Standard S6-88.	Single click
Offset	Value of the lateral offset of the truck with respect to the moving axle on your bridge. To be used only in 3D.	Single click

See also

The Moving Load Cases Dialog Box
The Moving Load Cases tab
Copying a Moving Load Case along with Components

Copying a Moving Load Case along with Components

To simultaneously copy a moving load case along with its attributes (Offset and factor included in the **Components** tab), use the **Duplicate** function, available in the spreadsheet's contextual menu.

PROCEDURE:

- Open the **Moving Load Cases** spreadsheet.

- Select the line that corresponds to the moving load case you want to copy.

- Right click and choose function **Duplicate** in contextual menu.

See also

Duplicate function

Moving Load Envelopes

Definition of Moving Load Envelopes

This spreadsheet, which is available in the **Loads / Moving Loads** menu, allows activating and defining moving load envelopes, according to a selected code. It is also useful to know 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**.

Procedure 1:

- Call up the **Load Combination Generator**. Select Code S6-00 in the first page and click "Next".

- In the second page, called **Specific Options**, include moving load envelopes into the generation and click the button [Mov. Load Envelopes…].
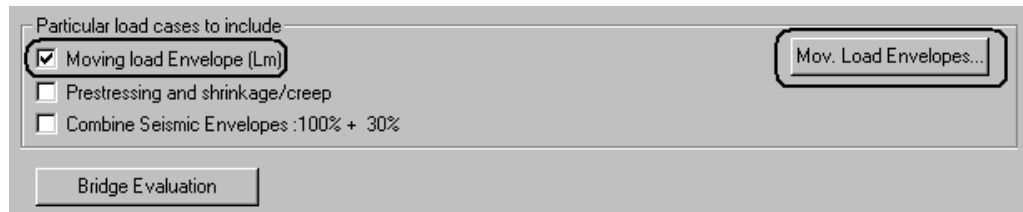
The **Definition of Moving Load Envelopes** spreadsheet will open on screen:

- Click any cell and right click to open the contextual menu. Select the function **Select a code**.

- Select CAN/CSA-S6-00 among the list box and click the button "Reinitialize". The spreadsheet will show you the required load combinations to be generated and corresponding moving load envelope. It also shows the required 2D axle factors for a 2D model.
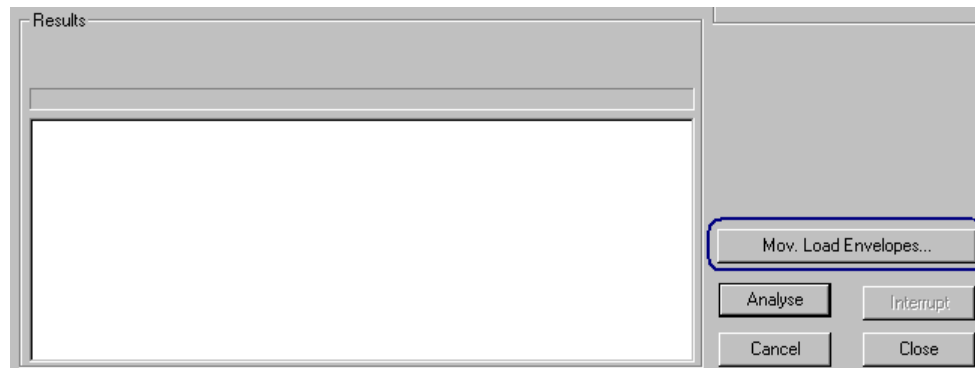

- Go to the column "To be analysed" and activate the moving load envelopes that you want to analyse when the moving load analysis will be launched. Close the dialog box.

- Complete the **Load Combination Generatior**.

Procedure 2:

- If you haven't use the **Load Combination Generator**, open the **Moving Load Analysis** dialog box and click the button [Mov. Load Envelopes…].

- Follow the steps as explained in Procedure 1.

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 Load Combination Generator.	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	VisualDesign shows the required load combinations and moving load envelopes, as per selected code. Disable the option [ ] to withdraw the load combination from the generation.	Double click or Space bar

See also

The Moving Load Analysis Dialog Box
2D Axles Factors spreadsheet
Load Combination Generation Wizard
Moving Load Envelopes and Load Combinations

Moving Load Envelopes and Load Combinations

The following tables describes the required moving load envelopes along with the load combinations that are considered, according to the selected code.

You will find these default values in the **Definition of Moving Load Envelopes** spreadsheet (**Loads / Moving Loads / Envelopes**).

CAN/CSA-S6-2000

Envelope	Load Combinations	Lane Overload	Type of 2D Axle Factor
Lm01	ULS (all of them) ULST no.1	Yes	Multiple lanes
Lm02	ULST no.2 FLS	No	Single lane

AASHTO-LRFD-2004

Envelope	Load Combinations	Lane Overload	Type of 2D Axle Factor
Lm01	Strength (all of them) Extreme event (all of them) Service I Service II Service III	Yes	Multiple lanes
Lm02	Fatigue	Yes	Single lane
Lm03	Service I modified (LL + IM) only	With or without lane load. (Refer to article 3.6.1.3.2)	Multiple lanes

See also

Definition of Moving Load Envelopes
Moving Load Analysis Dialog Box
2D Axles Factors Spreadsheet
Load Combination Generation Wizard

Analysis & Results

Moving Load Analysis Dialog Box

The Moving Load Analysis icon on Tools Toolbar

Open the **Moving Load Analysis** dialog box by clicking this icon and fill in parameters. Specify options to be considered during analysis.

Click the "Analyse" button to launch the analysis.

Section	Description
Structure	This section is shaded and serves only to inform users about the elements composing the structure.
Calculation parameters (For each moving load axis)	**Truck Load Step**: enter the distance between each truck load step. For a skew bridge, specify the distance at the beginning and at the end of the bridge to apply the moving loads on the total length of the bridge deck. **Before bridge**: Specify the truck load step before the bridge. **After bridge**: Specify the truck load step after the bridge.
Structure with cables	Check the box to include non-linear effects and select a *Mass* load combination. See topic Inclusion of non-linear effects into analysis
Corresponding Values (Concomitant)	To obtain concomitant values associated to maximum forces, check appropriate options. Values will be written in results spreadsheets, at column "Value".
Most critical load case for:	Check forces and reactions to obtain the most critical case for this force and/or reaction.
The "Mov. Load Envelopes" Button	Press this button to open the Definition of Moving Load Envelopes spreadsheet and activate the moving load envelopes to calculate.
Analyse button	Press this button to launch the analysis.

Date and Time

VisualDesign now displays the date of analysis in the **Moving Load 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.

Structure with Cables

If cables are included in a structure, such as a bridge, it is sensible to non-linear effects. Activate option "Include no-linear effects". VisualDesign will use the *Mass* load combination to calculate initial non-linear effects.

See also
Moving Load Analysis Procedures
Defining Moving Load
The Moving Load Cases
2D Axle Factors

Results

AFTER A MOVING LOAD ANALYSIS, STATIC ANALYSIS OR DESIGN:

Activate the Envelope mode and select a moving load envelope "Lm" on Activation toolbar. Use the **View Options** and look at diagrams.

Look at load combination, envelope and design results through spreadsheets and **View Options**.

Moving Load Analysis Procedures

User-defined Trucks

- Open the **Moving Load Definition** spreadsheet and, in the **Trucks** tab, check if the truck type to be used for the moving load analysis is listed. A number of pre-defined moving load configurations are used. However, others may be added (example: US codes). For new trucks, go to the **Axle/Wheel** tab and enter the "x & y" coordinates and weight of each axle (expressed as a fraction of the total weight or as an absolute value).

Moving Load Axis

- Define the path of the moving load by selecting a continuous line of elements along the path. These elements are defined by selecting each one and, in the **Properties** box, ticking off **Moving Load Axis** box.

The **Moving Load Axis** could also correspond to a continuous line defined by the edges of floor or plate elements. In this case, select the edge from the **Floor Characteristics** (or Plate) dialog box.

If you have a 2D Model:

Create the axle factor groups using the **2D Dynamic Load Allowance Factors** spreadsheet. Then go to the **Member** and **Support Characteristics** dialog box and associate each member and each support with an axle factor group.

Select the member or support, and then call the **Properties** function. In the **Member Characteristics** dialog box (or that of the **Support Characteristics**), choose a group of axle factors. The scroll list for various groups will appear. It contains the ones you defined in the **2D Dynamic Load Allowance Factors** spreadsheet.

Moving Load Cases

- Call up the **Moving Load Case Generator** (**Loads / Moving Load Cases / Automatic Generation**).

Load Combinations

- Call up the **Load Combination Generator** (**Loads / Load Combinations / Automatic Generation**). In the first page, select code S6-00.

- In the second page of the **Wizard**, include moving load envelopes for the generation of load combinations and click the button "Mov. Load Env." to open the **Definition of Moving Load Envelopes** spreadsheet. Select the code to be used (right click and choose **Selection of Code** in contextual menu). Default values will be initialized in this spreadsheet. Activate the moving load envelopes to be analysed. (For Bridge Evaluation, click this button, in the same page, and complete bridge evaluation parameters.)

Moving Load Analysis

- Open the **Moving Load Analysis** dialog box by clicking the icon . Complete parameters and activate options, if needed:

  - If the bridge is skewed, add the spacing at the beginning and at the end of the bridge to apply the moving loads on the total length of the bridge deck.

  - In the *Corresponding Values*: Activate options to get the maximum forces or reactions and concomitant values.

  - In the *Most Critical Case for*: Activate a type of force or reaction among the list to know the most critical case for this force and/or reaction.

- Click the "Analyse" button to launch the analysis.

The moving load analysis can supply up to 10 Lm envelopes, which can be integrated into static or design load combinations.

TIMBER DESIGN

TABLE OF CONTENTS

Chapter 9 Timber Design

General

The Timber Design Module

This module designs and verifies wood structure and is working like the steel design module. Timber specifications must be specified and timber design groups are available. A **Timber Design** tab appears in the **Member Characteristics** dialog box once that design criteria have been activated.

Materials and timber properties have been classified according to the new CAN/CSA-O86-1 standard. The classification (Visual, MSR, MEL or GLT) that was chosen for members will not be modified during the design, except for Visual C and D. However, for a given classification, VisualDesign will search for available cuts among the database. The section and material compatibility is tested during a design and verification.

Numerical results are in the form of a spreadsheet and a design brief is accessible from this spreadsheet. Graphical results are always available from the **View Options** dialog box or **View Options** toolbar.

See also
Timber Specification
Timber Species and Properties
Timber Nomenclature
Classification
Material and Section Compatibility
Availability of Timber Sections
Shape Designation
Timber Design Spreadsheet - Members
Truss Application
Timber Groups
Timber Design Results Spreadsheet
Design Brief
Print preview of Design Brief
Access to internal forces and deflections

Convention - Forces in members
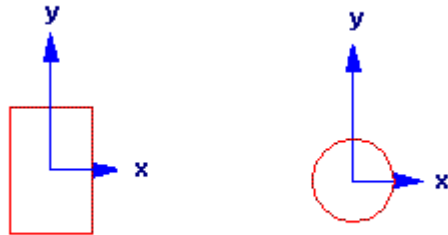
Sections' strong and weak axes:

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). The local z-axis is longitudinal to the member.

Deflection Calculation Method

VisualDesign™ verifies the deflection under service loads. However, users must specify a load combination having an *Instantaneous Deflection* status in the **Load Combinations** spreadsheet before proceeding with a design. The deflection is usually calculated with live loads only. The deflection calculation procedure is shown below.

Length of span between inflexion points (L1 to L5 below)

The calculation of maximum displacement according to an axis passing through two inflexion points is as follows: (v1 to v5 below).

In the design, we assure the L1/v1 ratio stays above the indicated limit in the allowable deflection edit box of each member (**Timber Design** tab of each members). The selection of shapes is based on minimum inertia that satisfies a minimum stiffness with respect to deflection. The strong and the weak axes are controlled in the same way according to the same allowable deflection.
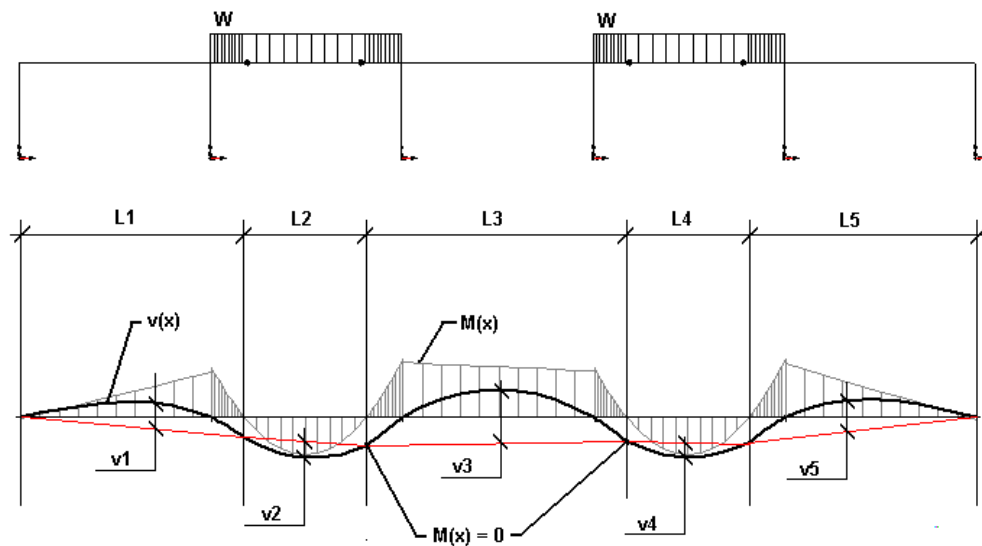
Shear Energy

N. B. The deflection due to shear energy is always added to calculated deflections.

P-Delta Effects

VisualDesign™ automatically uses non-linear analysis for a cyclic design. P-Delta effects are included when calculating internal stresses along members and deflections. It means that internal stress diagrams include the amplification caused by the deflection of the member due to transverse load that may affect the member (point load, triangular or trapezoidal load…).

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.

This table describes the fields in this dialog box:

Field	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

Field	Description
Tributary Area	You must specify if the reduction factor due to tributary area should apply to compression or bending.
	With the drop-down list box, select the code that will be use for the reduction of tributary surface
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 on P axial	The non-linear analysis will terminate when the variation falls below this value.
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 optimization cycles for the design of members.
Number of cycles (corrections)	When the optimized number of cycles is reached, the members that have not been optimized are evaluated in 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.

Materials and Species

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
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

Column	Description	Editing
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

Classification of Timber Members

The following classifications are included in VisualDesign:

Visual: Visual means that they are visually classified. All species are available.

The letter refers to available cuts, as follows:

A: 2x4…4x16

B: 2x2, 2x4, 4x4

C: 6x10, 8x12, 10x14…

D: 6x6, 6x8, 8x8, 8x10…

MSR: Machine Stress-Rated. The shear resistance, fv, is determined with this method. All cuts are available.

MEL: Machine Evaluated Lumber. The shear resistance, fv, is determined with this method. All cuts are available.

GLT: Glue Laminated Timber. Available species are S-P-F, D Fir-L, and Hem-Fir. All cuts are available.

Grade

The grade refers to the quality of the timber section, namely Select structural, No.1, No.2, No.3, Construction and Standard.

Species

Species refer to the tree species. Four species are available: S-P-F, D Fir-L, Hem-Fir or North Species.

See also
Timber Design tab
Timber Species and Properties
Timber Nomenclature

Timber Nomenclature

The name that appears in the *Material Selection* tree and in the **Timber Properties** spreadsheet is explained below.

Description of nomenclature:

The first term represents the classification of material, the second term, its species, and the third, its grade (quality).

Example: V1C_Northern_s:

- V: Visual classification (1C indicates that available cuts are 6x10, 8x12, 10x14, etc.);

- Northern: The species is *North Species*.

- s: The grade is *Select Structural*.

See also
Member Dialog Box
Defining a member section
Timber Properties Spreadsheet

Timber Sections

Shape Designation (Timber)

Sections are listed in the **Rectangular Sections** spreadsheet (**Common / Shapes**) for timber materials. Shapes beginning with this symbol [ ] are sawn timber (standard) sections, as per CAN/CSA-O86.1 Standard. Glue laminated sections begin with letters GL.

Rectangular Shapes Spreadsheet

1056	Metric Designation	Imperial Designation	Material	Canada	USA	Europe	Distribution	d mm	b mm	Composition	Number of pieces in local X-direction	Number of pieces in local Y-direction
486	[]292x445	[]12x18	Timber	[x]	[x]	[ ]	Public	445.00	292.00	Not applicable	1	1
487	[]292x495	[]12x20	Timber	[x]	[x]	[ ]	Public	495.00	292.00	Not applicable	1	1
488	GL80x38[2]	GL80x38[2]	Timber	[x]	[x]	[ ]	Public	76.00	80.00	Glue-laminated	1	2
489	GL80x38[3]	GL80x38[3]	Timber	[x]	[x]	[ ]	Public	114.00	80.00	Glue-laminated	1	3
490	GL80x38[4]	GL80x38[4]	Timber	[x]	[x]	[ ]	Public	152.00	80.00	Glue-laminated	1	4

Glue laminated sections:

A "Glulam" composition is selected for glue-laminated sections. The number of laminations must be specified for local x and y-directions.

The member local x-axis is always corresponding to the strong axis.
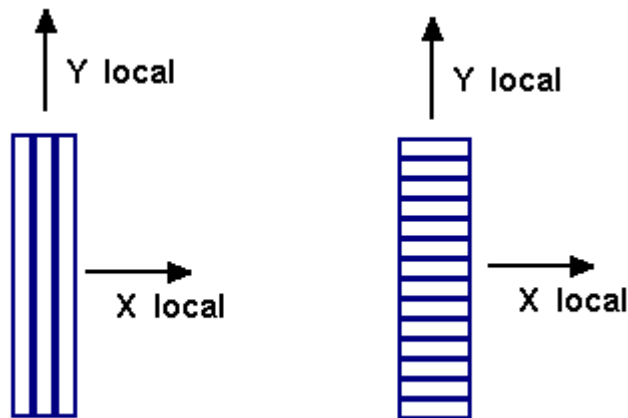
Customized sections

Create your own sections by inserting lines at the end of the spreadsheet. Specify the dimensions b and d only. If it is a glulam section, the columns shown just above must also be filled.

Use the same prefix ([ ] or GL) to localize your customized sections into the *Material* selection tree or use a new prefix. This *Material* selection tree is accessible through the **Member Characteristics** dialog box by pressing the I-Beam Icon.

See also

Rectangular Shapes Spreadsheet

Rectangular section

Identification

Name : GL80x38[10]

Area : 30400 mm²   Perimeter: 920 mm

Dimensions

d : 380 mm

b : 80 mm

x-Axis (strong)

Ix : 365.81 10e6mm4

Sx : 1925.33 10³mm³

rx : 109.7 mm

Zx : 2888 10³mm³

y-Axis (weak)

Iy : 16.21 10e6mm4

Sy : 405.33 10³mm³

ry : 23.09 mm

Zy : 608 10³mm³

Timber

Composition   Number of pieces

Glue-laminated   Direction X: 1

Direction Y: 10

Constant

J : 55.41 10e6mm4   OK

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

Availability of Timber Sections

The availability of timber sections is indicated in the **Rectangular Sections** spreadsheet, at columns "Canada", "USA" and "Europe". The availability of pre-defined sections is not editable. However, it is editable for personalized sections (insert lines at the end of the spreadsheet).

If you didn't create your own group of sections to be used for the design, the default one will be called "Canada". This group will include available sections in Canada, if this country was specified at the installation of VisualDesign. Refer to Group of Sections.

Group of sections

Example: If you added personalized sections in the **Rectangular Sections** spreadsheet and wish to used these sections only during the design, do as follows:

- Create a group of sections: Go to **Structure / Specifications / Sections' Groups.** Insert a line in this spreadsheet, give a name to this group and select personalized rectangular sections.

- Open the Timber Specifications spreadsheet, select a type of shape for the design (rectangular or round, for timber sections) and select the group of sections previously defined. VisualDesign will choose appropriate sections among those included in the group of sections.

Compatibility of Material & Section

Assign preliminary sections but make sure that the classification (material) is compatible with the section dimensions. The section and material compatibility will be tested during the design and verification process.

VisualDesign does not accept some combinations of material and section. For example, a GLT type of material must be assigned to laminated section, as specified in chapter 6 "Glue-Laminated Timber", Tables 6.2.1 and 6.3. As a result, if you assigned an incompatible material to a type of section, you will get a warning and the design will be stopped.

Particular case

There is an exception with Visual C and Visual D classifications. VisualDesign can change the classification C (which was assigned by the user) during the design process, to a classification D, and vice versa. The specified species is not modified.

During the design process and verification, VisualDesign tests the compatibility "material and section" according to the following:

Classification	Compatibility*
Visual A	2x2…2x12, 4x4…4x(All)
Visual B	2x2, 2x3, 2x4, 4x4
Visual C	b>=4½po, d>b+2po (Beam & Stringer)
Visual D	b>=4½po, d<=b+2po (Post & Timber)
MSR	Standard sections with b=38mm and d = all dimensions
MEL	Standard sections with b=38mm and d = all dimensions
GLT	Glue laminated sections only – all dimensions.

* The compatibility is based on the description of materials as defined in the CAN/CSA-O86.1 Standard.

See also
Classification of Timber Elements
Timber Specification
Timber Species and Properties
Timber Nomenclature

Specifications

Sections' Groups

This spreadsheet, available in **Structure / Specifications**, allows creating groups of sections that will be used for the design. A group of sections must be selected afterwards in a steel or timber specification, before launching a design. During the optimization process, VisualDesign will search among the sections that are included in the selected group of sections. Users can include many types of shapes in the same group of sections (W, C, 2L, rectangular, round, etc.) through a selection tree.

4	Number	Selection of sections	
1	Beams	W250x108 W250x37.5 W250x167 W250x14	
2	Columns	W360x1202 W360x1086 W360x990 W360x9	
3	Bracings	L203x152x25 L203x152x22 L203x102x25	
4	Beams_Timber	[]140x445 []140x394 []140x343 []140	
5			

A group of section MUST be selected in the steel specifications spreadsheet or timber specifications spreadsheet.

The default group of sections is "Canada". This group includes sections and shapes that are available in Canada, as indicated in the column "Canada" in the shapes spreadsheets (**Common** menu).

See also
Steel Specifications
Generator of Steel Specifications
Timber Specifications

Timber Specifications Spreadsheet

A timber specification must be assigned to members that will be design or verify. The type of analysis (design or verification) and design code or Standard must be specified in the specification.

Two specifications are already entered in the spreadsheet one for design and the second for verification. A *Glulam* composition must be selected if glue-laminated sections have to be designed. For standard sections, select a *Sawn Timber* composition. Service condition and treatment are also part of the spreadsheet.

Timber Specifications Spreadsheet									
	Number	Code	Type of analysis	Optimization	Group of sections	Interaction	Shape	Maximum height mm	Minimum height mm
1	CSA-086-1-Design	CSA-086-01	Design	Area	Canada	Standard	Solid (Rect.)	5000.00	0.00
2	CSA-086-1-Verif.	CSA-086-01	Verification	Area	Null	Standard	Solid (Rect.)	5000.00	0.00
3									

Specifications used for a design

A group of sections and a type of section (round or rectangular) must be selected in each specification, among other design criteria. Therefore, if different sections are composing the structure, specifications must be added in the spreadsheet because each type of section must possess its specification.

Adding specifications

Insert lines at the end of the spreadsheet (or copy/paste a line) and give a name to each one. Select the type of analysis, design code, a group of sections and a type of section. Then, assign specifications to members. During optimization, VisualDesign will choose sections among those listed in the group of sections.

Specifications used for a verification

The selection of a type of section and a group of section is not required. VisualDesign will verify assigned shapes according to the chosen code or Standard.

During modeling

For each member that will be designed, select either a design group or a specification in the **Timber Design** tab (**Member Characteristics** dialog box). For each member that needs to be verified, select a check specification.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Specification number (16 alphanumerical characters)	Single click
Code	Choose the standard that will be used for the design or verification of timber elements.	Double-click
Type of Analysis	Indicate in which case the specification will be used for: Design or Verification.	Double-click
Optimization	Design: Choose a type of optimization for the design: Area, Inertia or Height of the section. Verification: Not required.	Double-click

Column	Description	Editing
Group of sections	Design: Choose the group of sections that can be used for the design. Verification: Not required.	Double-click
Interaction	Choose an option for the design: Standard or Truss (<610mm). Interaction equations are different.	Double-click
Section	VisualDesign will search for this type among the selected group of sections. Verification: Not required.	Double-click
Max. Height	Design only: Specify the maximum height for optimized sections.	Single click
Min. Height	Design only: Specify the minimum height for optimized sections.	Single click
Max. Width	Design only: Specify the maximum width for optimized sections.	Single click
Min. Width	Design only: Specify the minimum width for optimized sections.	Single click
Maximum Capacity Factor	The design is done with respect to this given percentage of capacity. For example, if a value is inferior to 100%, it means that you are more severe than the chosen code.	Single click
Service Condition	Select the conditions that will be applied to this building: Dry or Wet.	Double-click
Treatment	Indicate the treatment: Treated, unincised, Treated incised (<89mm), Fire Retardant Treated, or Not applicable.	Double-click
Kt Fire-retardant treatment	Specify the Kt factor to be considered, as stipulate in table 5.4.3 of CAN/CSA-O86-01 Standard.	Single click
Kt (E) Fire-retardant treatment	Specify the Kt factor to be used with the Young modulus E, as stipulate in table 5.4.3 of CAN/CSA-O86-01 Standard.	Single click
Description	Describe this specification, if you want to.	Single click

See also

Truss Application
Timber Design Spreadsheet - Members

Timber Design Groups

Grouping Members

Timber specifications must be assigned to members before grouping them.

Use the **Group Members** function (**Structure / Group)** to quickly create design groups for members to get a more practical design.

Procedure:

- Select members that you wish to group together (keep the [Ctrl] key down while selecting members with your mouse);

- Go to **Structure/Groups** / **Group Members** or use the shortcut keys [Ctrl]+G.

- This dialog box will appear. Give a name to this group, choose a timber specification and click OK..
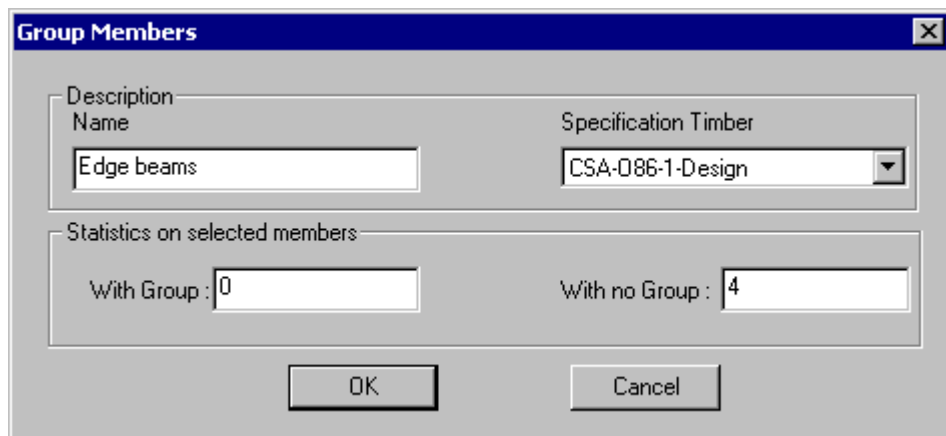
Design groups will be assigned to members in the **Timber Design** tab of **Member Characteristics** dialog box. The name of design groups can be modified in the **Timber Design Groups** spreadsheets.

See also

Displaying Member Design Group
Timber Design Groups spreadsheet
Automatic Grouping of Elements

Timber Design Groups - Members

Open the **Timber Design Group** spreadsheet (**Structure / Groups / Timber**) to modify the name of design groups or to select other specifications.

Group: Structural Data

Column	Description	Editing
ID	ID number for this group. Automatically calculated	No
Number	Name of this design group (16 alphanumerical characters).	Single click
Specification	Assign a timber specification to this design group.	Double-click

Grouping Members Automatically

This tool groups members having a symmetrical layout. Specifications must be assigned to members before grouping them. Members will be grouped together if they are assigned to the same specification.

Procedure:

- Select **Structure/Groups / Auto Group**. This dialog box will appear on screen:

- Members can be grouped according to one of these options:

- Grouping continuous members. These members must be rectilinear and assigned to the same specification.

- Grouping mirror members according to vertical planes. The user can specify the horizontal position of the mirror plane.

- The assigned specification is corresponding to the one that was assigned to the first member composing this group. The group number corresponds to the number of the first member part of this group.

Viewing a Design Group

To look at a design group, do the following:

- Select the **Timber Design Groups** spreadsheet under **Structure/Groups**.

- Highlight the line that corresponds to the group that you want to look at.

- Click OK.

Members that are part of this group will be highlighted on screen.

If you do not like the way members are grouped, delete lines in the **Timber Design Groups** spreadsheet and use the **Group Members** function.

Member

Timber Design tab (Member Dialog Box)

If you activated design criteria and selected a wood material in the **Member** tab of **Member Characteristics** dialog box, the **Timber Design** tab will be activated.

A *Design* specification or design group must be assigned to each member to be designed. Design groups are automatically assigned to members when the **Group** function is used. For members that need to be check only, select the *Verification* specification.

Sawn Timber

The species and classification must be specified through the list boxes.

Glue-Laminated Sections

The species must be specified but the classification is not required.

MSR and MEL Classifications

The value for shear stress, fv, must be specified. This value is obtained with the mechanical tests. Otherwise, results will not be valid because the default value for fv is zero.

Look at this table to learn more about parameters included in the **Timber Design** tab.

Field	Description
Design Parameters	N. B. You must choose between a design group OR a Specification.
Design or Verification	This shaded field informs you about the type of analysis that is going to be carried on, as you specified in the timber specifications spreadsheet.
Specification	Choose a timber specification among the drop-down list box. If it does not exist, press the [...] button next to this field and define one.
Design Group	Select the member design group. If it does not exist, press the [...] button next to this field and create one. (A timber specification must be selected in the Timber Design Groups spreadsheet.)
Bending – Laterally Supported Members	See topic Lateral Support
Top and Bottom Fibre	Specify the member lateral supports, at top and bottom fibre. Tick off the appropriate boxes: Node i, Node j or Continuous lateral support.
Fixed	Activate this box to be allowed to enter a value in the "Lex" field, for the top and bottom of the section. (This option is not available yet.)
Lex	Section effective compression lengths at the top and bottom according to strong axis. (This option is not available yet.)
Bearing Conditions at node i and j (Optional)	
Bearing	Enter the bearing length at node i and j.
Notch	Specify the position of notch at node i and j according to member local axis system.
Depth "dn"	Enter the depth of notch at node i and j for the calculation of Notch factor.
Length "e"	Enter the length of notch at node i and j for the calculation of Notch factor.
General	
Species	Select a species for the design: n/a, D Fir-L, Hem-Fir, S-P-F, or North Species. The species specified in this field takes precedence over the material selected in the Member tab.

Heading	Description
Grade	Select a grade for the design: n/a, Select, No.1, No.2, No.3, Construction, or Standard. The grade specified in this field takes precedence over the material selected in the Member tab.
Shear (MSR/MEL)	Enter a value for the maximum shear stress according to MSR (Machine Stress Rated) or MEL (Machine Evaluated Lumber) methods.
Fastenings	If the member is composite, choose the fastening that is used: Nails, Bolts, Split Ring or n/A.
System Factor (Kh)	Select the case that applies to this member for the calculation of System Factor Kh.
Duration Kd	The default Load Duration factor (Kd) is equal to 1.0. To modify this value, check the Kd box and enter a value in the field next to the box.
Effective Compressive Length	See topic Automatic Calculation of Kx, Ky, Kt and Kz
Kx Factor	Effective compressive length factor relative to strong axis. If *Auto* is checked, Kx coefficient is automatically calculated.
Ky Factor	Effective compressive length factor relative to weak axis. If *Auto* is checked, Ky coefficient is automatically calculated.
Effective Net Area	
Width of Hole	Width of the member hole, if bolts are used as connections.
Allowable Deflection	
Lx/	Enter a deflection criterion relative to strong axis.
Ly/	Enter a deflection criterion relative to weak axis.

See also
Member Dialog Box
Timber Groups
Timber Specifications
Truss Application
Automatic Calculation of Kx, Ky, Kt and Kz

Timber Member Design Spreadsheet

The **Timber Design (Member)** spreadsheet includes the member design criteria and parameters that are needed for a timber design.

Enter data in the **Timber Design** tab (**Member Characteristics** dialog box) and use this spreadsheet to consult or sort data for editing common values.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Member number (16 alphanumerical characters).	Single click
Group	Choose a member design group, if desired. (Specification is included in the Timber Groups spreadsheet.)	Double-click
Specification	If the member is not part of a design group, choose a timber specification in the drop-down list box. Refer to Timber Specifications Spreadsheet	Double-click
Kx computation	Choose an automatic or manual computation of effective compression length factor, relative to strong axis.	Double-click
Kx	Effective compression length factor relative to strong axis. For an "Auto" computation, the Kx coefficient is automatically calculated.	Single click
Ky computation	Choose an automatic or manual computation of effective compression length factor, relative to weak axis.	Double-click
Ky	Effective compression length factor relative to weak axis. For an "Auto" computation, the Ky coefficient is automatically calculated.	Single click
Lux Bottom Noi	If the section bottom flange is laterally supported at node i, choose option [x].	Double-click or Space bar
Lux Bottom Cont	If the section has continuous lateral support at the bottom flange, choose option [x].	Double-click or Space bar
Lux Bottom Noj	If the section bottom flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Lux Top Noi	If the section top flange is laterally supported at node i, choose option [x].	Double-click or Space bar

Column	Description	Editing
Lux Top Cont	If the section has continuous lateral support at the top flange, choose option [x].	Double-click or Space bar
Lux Top Noj	If the section top flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Lx/	Enter a deflection criterion for strong axis.	Single click
Ly/	Enter a deflection criterion for strong axis.	Single click
Fixed "Le" Top	Choose option [x] to fix the value of "Le" (effective compression length according to strong axis) at the top of the section. (This option is not available yet.)	Double-click or Space bar
Lex Top	Enter the effective compression length at the top of the section. (This option is not available yet.)	Single click
Fixed "Le" Bottom	Choose option [x] to fix the value of "Le" (effective compression length according to strong axis) at the bottom of the section. (This option is not available yet.)	Double-click or Space bar
Lex Bottom	Enter the effective compression length at the bottom of the section. (This option is not available yet.)	Single click
Bearing Length Node i	Specify the bearing length at node i.	Single click
Position of Notch Node i	Choose the position of notch at node i among the drop-down list box.	Double-click
Notch Depth "dn" Node i	Enter the depth "dn" of notch at node i.	Single click
Notch Length "e" Node i	Enter the length "e" of notch at node i.	Single click
Bearing Length Node j	Specify the bearing length at node j.	Single click
Position of Notch Node j	Choose the position of notch at node j among the drop-down list box.	Double-click

Column	Description	Editing
Notch Depth "dn" Node j	Enter the depth "dn" of notch at node j.	Single click
Notch Length "e" Node j	Enter the length "e" of notch at node j.	Single click
System Factor (Kh)	In the drop-down list box, choose an option that applies to this member for the calculation of system factor Kh.	Double-click
Fastenings	Choose the type of fastenings used for this member: Nails, Bolts or Split Ring.	Double-click
Fixed Kd	Select option [x] to fix the value of duration factor Kd.	Double-click or Space bar
Duration Kd	Enter the value of duration factor Kd.	Single click
Width of Holes	Enter the width of holes for the calculation of net area.	Single click
Species	Specify the timber species used for this member: D Fir-L, Hem-Fir, S-P-F or North Species.	Double-click
Grade	Specify the section grade: Select, No.1, No.2, No.3, Construction or Standard.	Double-click
Shear MSR or MEL	Enter the maximum shear stress for this member (MSR or MEL Method).	Single click

See also

Timber Design tab- Member Dialog box
Timber Specifications Spreadsheet

Timber Groups Spreadsheet

Select the **Group** function (**Structure / Groups**) or use the short cut keys **[Ctrl]+G** to quickly create design groups for selected members. The created design groups will be listed in the **Design Groups** spreadsheet (**Structure / Groups / Timber**). Open this spreadsheet to edit names, modify the assigned specifications or to select a design group for its display on screen.

Group: Structural Data

Column	Description	Editing
Group ID	Automatically calculated	No
Number	16 alphanumerical characters	Single click
Specification	Select a timber specification among the list. VisualDesign will design this group of members according to this specification.	Double-click

CivilDesign inc. Engineering Software

Truss Application

Sawn Lumber Design for Specific Truss Applications

VisualDesign can design a truss according to clause 5.5.13 (*Sawn Lumber Design for Specific Truss Applications*) of standard CAN/CSA-O86. To specify a member as a truss member, go to the Timber Specification spreadsheet and select option *Truss (<610mm)* at column "Interaction". This specification must be assigned to the members composing the top chord.

Member Usage

Secondary members such as diagonals and verticals must have a usage set to "secondary member". This is required to allow VisualDesign recognizing lateral supports along the top chord and calculating the ratio M1/M2 (clause 5.5.13.5). Ratio M1/M2 are needed to find the appropriate modification factor *KM*

PROCEDURE

- Select all secondary members in the truss and click the **Properties** icon to open the **Member Characteristics** dialog box.

- Open the "Usage" list box in the **Member** tab. Expand the *Timber* root of displayed selection tree, and select "Secondary member" as usage for these members.

Modification Factor KM:

VisualDesign automatically calculates modification factor *KM* per table 5.5.13.5 of standard CAN/CSA-O86.

For load combinations including an envelope (moving load, seismic, or general dynamic), the program considers the third equation for the calculation of modification factor *KM* , namely:

1.67 [Lp/d]exp-1/6 = 1.3

See also
Timber Specifications
The Member tab
Member Usage

Analysis and Results

Procedure

To proceed with a cyclic design or a check of your structure, you must follow these steps:

- Complete the parameters in the **Analysis** tab in the **Project Configuration** dialog box. Two parameters are important to have a correct design, namely: The number of cycles for optimization and the number of cycles for correction.

- Activate design criteria in the **Member** tab of the **Member Characteristics** dialog box.

- Go to **Structure / Specifications** and define your own **Group of Sections** to be used during the design process. (A pre-defined one can also be selected in the **Timber Specifications** spreadsheet.)

- Select the **Timber Specifications** spreadsheet (**Structure** menu). Choose a Design or Verification specification, complete the required parameters, and modify the defaults values, if needed.

- If you wish to have identical sections for continuous members or wish to group members, select them and use the **Group Members** function (or use the short-cut keys [Ctrl] + G) in the **Structure/Group** menu.

- For all members that you wish to design, you must complete the information supplied in the **Timber Design** tab of the **Member Characteristics** dialog box. Select a steel specification OR a design group for the selected members.

- Use the **Load Combination Generator** to generate required load combinations for a selected Code or Standard.

- Click the Analysis and Design icon and start the design process by clicking the "Analyse" button.

- To look at the results, consult the **Timber Design Results** spreadsheet in the **Results/Structure Design** menu. With the **View Options**, you can display the sections number, capacity, and insufficient resistant members. To look at load combination results, activate the "Load Combination" mode and look at results through the **Results/Load Combinations** menu or through **View Options** (**Results** tab).

Timber Design Results Spreadsheet

The spreadsheet includes among others, the name of section, the critical load combination and the code provision that controlled the design, members design load, bending, shear and axial resistances, net area and member's stability.

A message also informs you if the design is OK or not. If not, lines are marked with yellow colour, the invalid result (cell), in red. Colours don't appear at the printing.

As soon as the design is completed, the "Design Results" mode is automatically activated. You can look at results in many ways:

- Go to **Results** menu and select the heading **Structure Design /Timber**. If no members were selected, the spreadsheet will include all members;

- Double-click on a member to access the spreadsheet. Only the selected member results will be included in the spreadsheet;

- Click on many members while pressing down the [Ctrl] key and click on the **Properties** icon . Only the selected members will be part of the results spreadsheet.

In the lower part of this spreadsheet, you will notice three buttons. They give access to member internal forces and to the printing of the selected member design brief. A print preview is also supplied for the design brief.

Group: Design Results

Column	Description	Editing
Number	Section number	No
Group	Timber Group to which the member belongs.	No
Section	Section name	No
Load Combination Mf+Nf	Critical load combination for combined bending and compression forces for this member.	No
Design load Mf+Nf	Member design load for maximum bending and compression forces. A value greater than 100% means that the section capacity is too small	No
Load Combination Shear	Critical load combination for shear force for this member.	No

Column	Description	Editing
Design load Shear	Member design load for maximum shear force. A value greater than 100% means that the section capacity is too small	No
Code Provision	Code provision that controlled the design	No
Type of Mfx	Positive or negative bending moment that controlled the design on strong axis.	No
Mrx Lu>0	Resisting moment (strong axis) when Lu > 0	No
Lux	The member unsupported length on strong axis.	No
w2x	Factor allowing for the increase of the resisting moment of a laterally unsupported segment (strong axis)	No
Type of Mfy	Positive or negative bending moment that controlled the design, on weak axis.	No
Mry Luy=0	Resisting bending moment on weak axis when Lu = 0	No
Type of Nz	Indicate the type of axial force acting on the member: Tension or Compression.	No
Tension Tr	Tensile strength for this section.	No
Cr - Stability	Compressive strength according to KL/rx or KLy/ry.	No
KLx	Effective compression length on strong axis.	No
KLy	Effective compression length on weak axis.	No
Ccx	Slenderness of compressed member on strong axis.	No
Ccy	Slenderness of compressed member on weak axis.	No
Net Area	Net Area of the section: gross area minus bolt holes and affected by the reduction coefficient.	No
Vrx	Shear resistance on weak axis.	No
Vry	Shear resistance on strong axis.	No
Fr yi	The factored lateral resistance of the connection at node i for loads acting parallel to grain.	No

Column	Description	Editing
Fr yj	The factored lateral resistance of the connection at node j for loads acting parallel to grain.	No
Qr yi	The factored lateral resistance of the connection at node i for loads acting perpendicular to grain.	No
Qr yj	The factored lateral resistance of the connection at node j for loads acting perpendicular to grain.	No
Trz	The member torsional resistance.	No
Message	Design result: Ok, Insufficient capacity, etc.	No
Deflection Result	Status of the deflection result: n/a, Ok < Allowable Deflection, Too much deflection.	No
Load Combination Mx	Load combination that governs the deflection criterion according to strong axis.	No
Lx	Calculated span between inflexion points, according to strong axis.	No
Deflection (Mx)	Deflection between two inflexion points, according to strong axis.	No
Lx/	Ration between Lx/Deflection (Mx)	No
Load Combination My	Load combination that governs the deflection criterion on weak axis.	No
Ly	Calculated span between inflexion points, on weak axis.	No
Deflection (My)	Deflection between two inflexion points, on weak axis.	No
Ly/	Ratio between Ly/Deflection (My)	No

Geometric Constants

R	Bend radius to the element axis.	Non
L	Length of element	Non
Z	Volume	Non

Column	Description	Editing
K Factors		
Kd	Duration factor	Non
Khc	System factor for compression loads	Non
Kn	Notch factor for sawn timber.	Non
Kng	Notch factor for glulam	Non
Ksb	Service factor for bending	Non
Ksc	Service factor for compression.	Non
Kse	Service factor for Young modulus	Non
Ksv	Service factor for shear	Non
Kt	Treatment factor	Non
KtE	Treatment factor in relation with the Young modulus.	Non
Kzb	Size factor for bending for sawn timber.	Non
Kzcg	Size factor for compression for glulam	Non
Kzv	Size factor for shear for sawn timber.	Non
K Factors for strong axis and weak axis		
Kc	Slenderness factor for compressed elements	Non
Khb	System factor for bending	Non
Khs	System factor for shear	Non
Kl	Factor for lateral stability of bent elements	Non
Kx	Curvature factor	Non
Kzbg	Size factor for bending, for glulam.	Non
Kzc	Size factor for sawn timber in compression.	Non
Lx ou Ly	Unsupported length between inflexion points, for calculating deflections.	Non
Cv	Coefficient for shear force	Non

Column	Description	Editing
Wf	Total factored load	Non

See also
Print Preview of Design Brief
Internal Forces and Deflection Results

Member Internal Forces from the Design Results spreadsheet

The Internal Forces icon of Design Results spreadsheet

This icon, posted in the lower part of the **Timber Design Results** spreadsheet, allows consulting internal stresses and deformations for the selected member. To do so, select a line in the spreadsheet and press the icon.

See also
Design Brief
Timber Design Results

Print Preview of Design Brief

The "Print Preview" icon for Design Brief

This icon is available in the **Timber Design Results** Spreadsheet. It allows consulting the **Design Brief** before printing.

To activate this function, select a line in the **Timber Design Results** spreadsheet and press this icon. The Design Brief (for the selected member) will appear on screen. If you wish to print it, select the "Print" command that is supplied in the Print Preview window. Notice that results are given for the load combination that controlled the design of this member.

See also
Design Brief
Timber Design Results spreadsheet
Title Bar

Design Brief

The Design Brief icon of Design Results spreadsheet

This icon, posted in the lower part of the **Timber Design Results** spreadsheet, allows printing the design brief for a selected member. To do so, select a line in the spreadsheet and press the icon. Then, select a printer in the "Print" dialog box. Do not forget that results are according to critical load combinations (Combined bending/compression, Shear, and Deflection). Deflection is calculated according to inflexion points and shear energy is included.

The **Design Brief** is divided in three sections:

- The left part shows forces diagrams for critical load combinations, namely, **Bending-Compression** and **Shear** and finally, deflection diagrams for **Deflection** load combinations.

- The middle part shows concentrated and distributed load cases applied to the member, for the given load combination. Example: The loading diagram representing the dead load on the member and the loading diagrams representing the two-way floor triangular distribution of live load transferred on the member. Up to eighteen loading diagrams can be drawn.

- Finally, the third part includes the following verification:

  - Name of section and selected building code;

  - Shape properties;

  - Material properties;

  - Critical load combination for combined bending/compression forces in the member, and forces that are considered in the design.

  - Critical load combination for maximum shear force acting on the member, and corresponding forces;

  - Verification of KL/r max, Mr with and without lateral buckling, verification of code provisions and deflections for *Deflection* load combinations, if any.

  - A list of geometric constants and K factors used for the design or verification.

  - Conclusion.

MESSAGES ABOUT KL/R:

If you get a message telling that the parameter KL/r is exceeding the allowable limit, it means that a load combination created this maximum value for KL/r. It may not correspond to the critical combined forces governing the design.

Internal Forces and Deflections for Design Groups

Activate an envelope, go to **Results / Envelope** and select this spreadsheet to look at maximum and minimum forces for a specific design group.

Group: Envelope Results: (title)

Column	Description	Editing
Number	Design group number (12 alphanumeric characters).	No
Shape	Chosen shape for this design group.	No
Value	Type of results: Max. or Min. An empty field indicates a standard analytical result.	No
Bending Mx	Min. or max. bending moment on strong axis for this design group.	No
Shear Vy	Min. or max. shear force on strong axis for this design group.	No
Bending My	Min. or max. bending moment on weak axis for this design group.	No
Shear Vx	Min. or max. shear force on weak axis for this design group.	No
Axial Nz	Min. or max. axial force for this design group (positive in tension).	No
Torsion Tz	Max or min. torsion force for this design group.	No
u (weak axis)	Max or min displacement on weak axis for this design group.	No
v (strong axis)	Max or min displacement on strong axis for this design group.	No
w (axial)	Max or min axial deformation for this design group (accurate solution only at member ends).	No

STEEL DESIGN
(including Seismic Steel Design)

TABLE OF CONTENTS

Chapter 10 Steel Design

General

Analysis and Cyclic Design Limitations

- Interstory sliding and lateral deflection caused by earthquakes are evaluated at each floor level. The user has to verify these drifts to make sure that they are within the allowable limit.

- The calculation of the unsupported length is not valid for continuous members having different beta angles. (For construction, this case is very rare, almost impossible).

- Lateral buckling of built-up sections is not included in this module.

- Torsion that reduces bending capacity is not included, except for the **Tower Design** Module.

See also
Convention – Forces in Member
P-Delta Effects
Inelastic Effects caused by an Earthquake
Classification of Sections
Code Provisions Considered in the Design
Deflection Calculation Method
Shear Energy
Definition of Bolted Connections
Bolted Connection Design Results
Steel Design Procedure
Steel Design Result
Steel Design Criteria

Convention - Forces in members
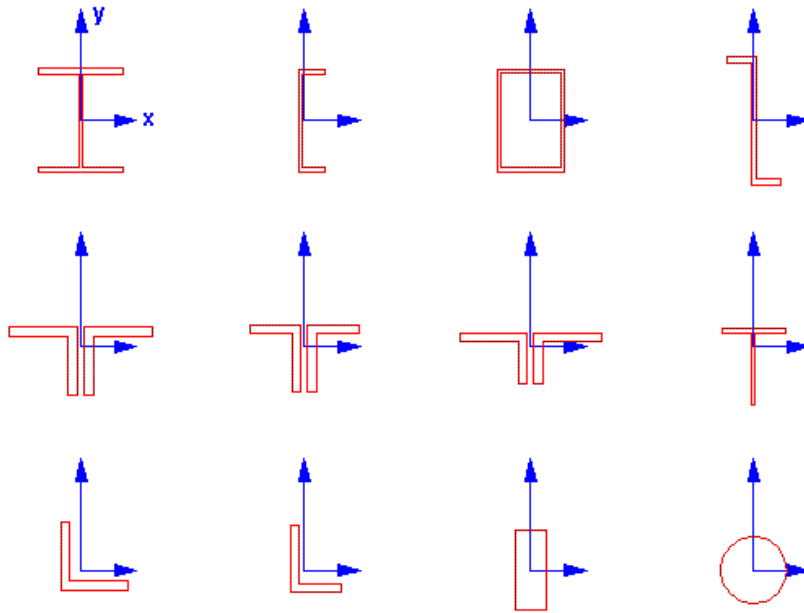
Sections' strong and weak axes:

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). The local z-axis is longitudinal to the member.
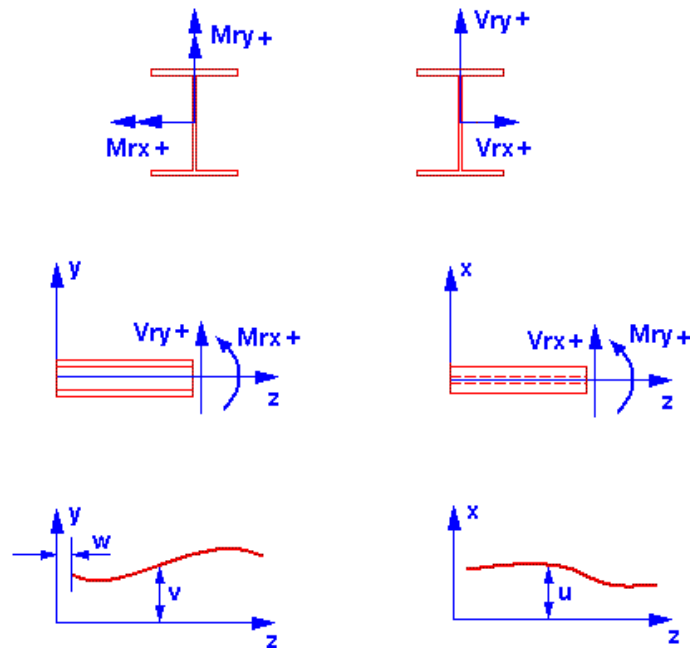
For steel angles, Mw and Mz values are always transformed into the orthogonal axis system once that the design is completed.
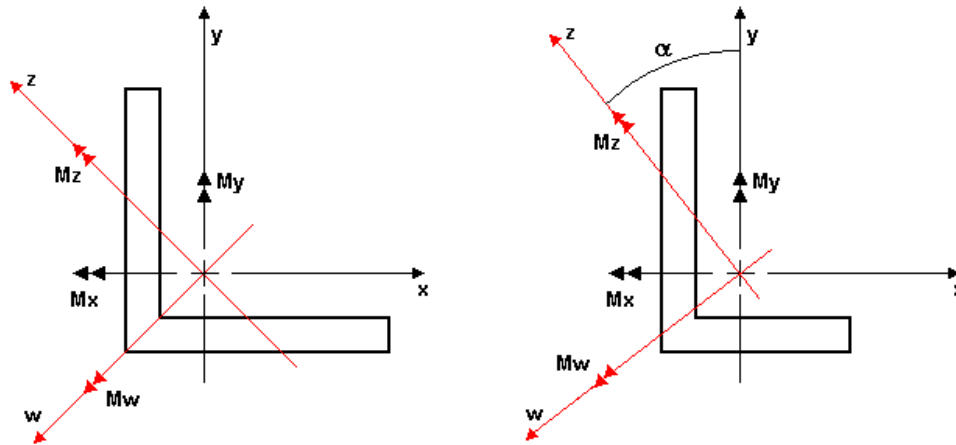
See also

Major/minor and Orthogonal Axis Systems

Tension-only Bracings

- Go to **Analysis** tab (**Project Configuration**) and increase the number of iteration (10 or 12) for non-linear analysis.

- Go to the **Member** tab.

- Select a tension-only axial end condition (<-[ ]->) and select the usage "[Diagonal Xt]" in the Usage list 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.

Convergence problem

In some cases, convergence cannot be reached during a steel design with tension-only bracings. Lateral loads or gravity loads applied directly at the top of braced bays can induce compression in the bracings. The compression causes instability in the structure when VisualDesign remove the compressed elements (bracings) during the release process. In fact, the structure looses the bracings during the process.

To resolve this problem, do as follows:

- In the **Member** tab: Put back tension/compression axial end conditions and assign a "Standard" usage to bracings. Specify a value of 0.5 as effective stiffness for bending, axial and shear.

- In the **Steel Design** tab: Disable the automatic calculation of effective compression lengths (Kx and Ky) and put a 0 value everywhere.

- Launch the design again.

Guys (Tower design)

Axial end conditions are different for guys. Please refer to Modeling and analysing a guyed structure.

See also
Member End Conditions

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.

See also
Member Characteristics Dialog Box
The Member tab

P-Delta Effects

VisualDesign™ automatically uses non-linear analysis when the user runs an analysis and a cyclic design. VisualDesign™ also calculates internal stresses along the members and includes the P-Delta effects. It means that internal stress diagrams include the amplification caused by the deflection of the member due to transverse load that may affect the member (point load, triangular or trapezoidal load…). This unique method allows structural design ignoring correction factors U1 and U2 of CAN/CSA-S16-01 Standard and B1 and B2 factors of American Standard LRFD-1994.

Classification of Sections

Classification of steel sections is required to determine the bending and compressive resistance. Sections are classified in four categories:

- Class 1 (Plastic);

- Class 2 (Compact);

- Class 3 (Non compact);

- Or Class 4 (Slender).

VisualDesign™ evaluates the class of the section according to clause 11.2 (table 1) of the CAN/CSA-S16-01 standard and according to American Code LRFD.

VisualDesign classification is written in the Steel Design Results spreadsheet, in four columns: for bending Mx, My, web and compression. It is based on the steel shape ratio b/t and h/w.

Cold-formed Sections

All cold-formed sections of class 1 to 4 are evaluated according to S136-94 Standard.

L and 2L Shapes

The design and verification for these sections is based on the American AISC-LRFD Standard, 3rd Ed. 2001, which is very complete compared to other standards.

Steel Sections of Class 4, other than cold-formed

For the design: All class 4 sections are evaluated by reducing fy value to re-class them as class 3 sections, as permitted by CAN/CSA-S16-01 Standard. This method is conservative, compared to Standard S136-94.

Verification: VisualDesign will verify the class 4 section by reducing the value of fy, as mentioned above.

HSS and Solid Round Sections:

N.B. HSS shapes and Solid Round Sections can be of class C (cold formed without stress-relieved) or class H (hot formed with stress-relieved). The user must indicate if the section is stress-relieved or not by ticking the appropriate box in the **Member** dialog box.

Elastic Lateral-Torsional Buckling

This topic describes equations that VisualDesign when verifying the bending strength of steel members considering torsional buckling.

Reference: *Guide to Stability Design Criteria for Metal Structures*, John Wiley & Sons, 5e Éd. Theodore V. Galambos, p.194-197.

Simply supported doubly symmetric beams of constant sections

Non-uniform bending:

Mcr = CbM0cr   Equation (5.3)

M0cr : Critical buckling moment

Cb: Equivalent uniform moment factor

M0cr is equal to:

$$M_{0cr} = \frac{\pi}{L}\sqrt{EI_y\,GJ}\cdot\sqrt{1+W^2}$$

Equation (5.1)

Where L is the span length, E and G are the elastic and shear moduli, respectively, and Iy, J and Cw are, respectively, the minor axis moment of inertia, the St-Venant torsion constant, and the warping constant.

$$W = \frac{\pi}{L}\cdot\sqrt{\frac{EC_w}{GJ}}$$

Equation (5.2)

$$C_b = \frac{12.5M\max}{2.5M\max + 3M_A + 4M_B + 3M_C}$$

Equation (5.4c)

Where Cb applies to any shape of the moment diagram curve between the brace points.

M max = absolute value of the unbraced in the unbraced segment;

MA, MB and MC are the absolute value of the moments at the quarter point, centre, and three-quarter point, respectively, in the unbraced segment.

See also
Steel Design Results Spreadsheet

Code Provisions for Steel Design

In the evaluation of member capacity, VisualDesign™ verifies each load combination according to CAN/CSA-S16-01, at each subdivision of every member. It is very important that engineers create adequate load combinations and specify enough member subdivisions (Refer to **Analysis** tab in **Project Configuration**) to get a good design.

VisualDesign verifies the following clauses:

13.2	Axial Tension
13.3.1.	Flexural Buckling
13.3.3b)	Class 4 members in Compression
13.4.1.1	Elastic Analysis (Shear)
13.4.2	Webs of Flexural Members not having Two Flanges
13.5	Bending – Laterally Supported Members
13.6	Bending – Laterally Unsupported Members
13.8.2	Member Strength and Stability – Class 1 and Class 2 Sections of I-Shapes Members
13.8.3	Member Strength and Stability – All Classes of Sections Except Class 1 and Class 2 Sections of I-Shapes Members
13.9	Axial Tension and Bending

Clause13.8.4 is not considered in VisualDesign™ because of the method used in the calculation of stresses in the members.

Deflection Calculation Method

VisualDesign™ verifies the deflection under service loads. However, users must specify a load combination having an *Instantaneous Deflection* status in the **Load Combinations** spreadsheet before proceeding with a design. The deflection is usually calculated with live loads only. The deflection calculation procedure is shown below.

Length of span between inflexion points (L1 to L5 below)

The calculation of maximum displacement according to an axis passing through two inflexion points is as follows: (v1 to v5 below).

In the design, we assure the L1/v1 ratio stays above the indicated limit in the allowable deflection edit box of each member (**Steel Design** tab of each members). The selection of shapes is based on minimum inertia that satisfies a minimum stiffness with respect to deflection. The strong and the weak axes are controlled in the same way according to the same allowable deflection.
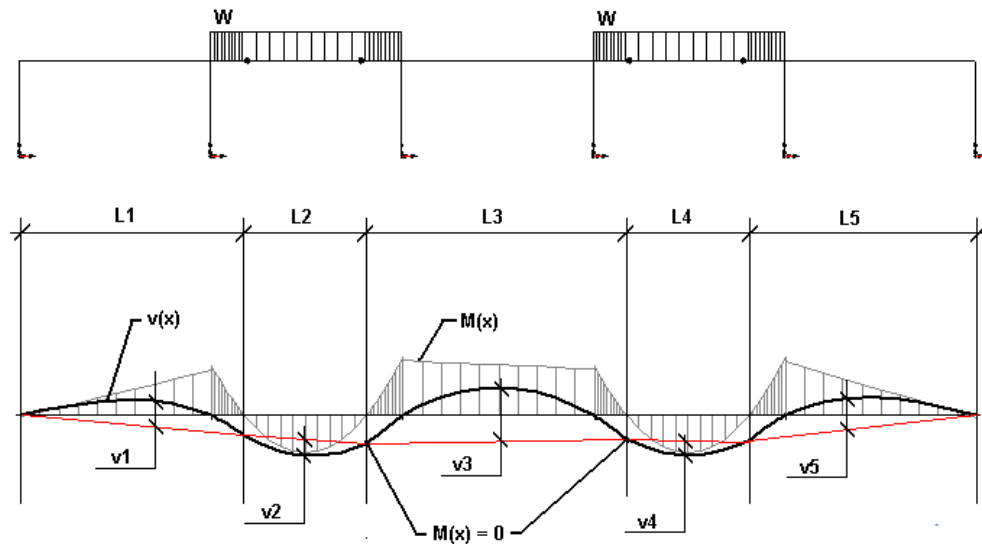
Inelastic Effects caused by an Earthquake

When the user specifies a level of ductility R (*Seismic* tab in **Project Configuration**) for the structure, it means that he accepts a certain deflection during the earthquake. These deflections cause additional stresses in the structure, which can be evaluated in two ways: the first, by elastoplastic analysis, and the second, by an approximate method as defined in the *National Building Code 1995*.

The program considers each seismic direction, all the necessary parameters, and corrects the spectral analysis by including inelastic effects. The **Levels** spreadsheet contains all the parameters used by VisualDesign™ for the computation of *Theta x* that is used to amplify the stresses.

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.

This table describes the fields in this dialog box:

Field	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

Field	Description
Tributary Area	You must specify if the reduction factor due to tributary area should apply to compression or bending.
	With the drop-down list box, select the code that will be use for the reduction of tributary surface
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 on P axial	The non-linear analysis will terminate when the variation falls below this value.
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 optimization cycles for the design of members.
Number of cycles (corrections)	When the optimized number of cycles is reached, the members that have not been optimized are evaluated in 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.

Note: Save Node Displacements

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

Steel tab

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".

Field	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).
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

Automatic Calculation of Kx, Ky, Kt and Kz

Bolts

Bolt Steel Grades Spreadsheet

Define the bolt steel grades that you will be using in the design or verification of tower member 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

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
Hole Diameter	Required hole diameter for this bolt.	Single click

Column	Description	Editing
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

See also
Bolt Steel Grades

Bolted Connection Models

Bolts Layout

The bolts layout can be of three kinds: In line, staggered A or staggered B. Look at the images below.

Bolts Layout: In line
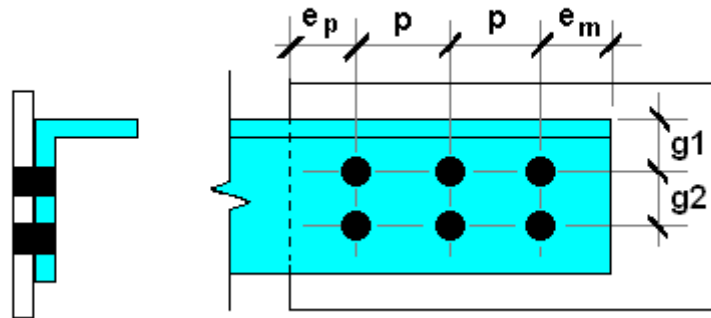
Bolts Layout: Staggered - Type A
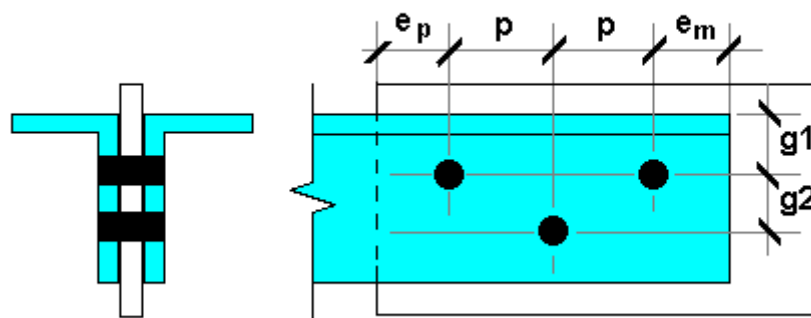
Bolts Layout: Staggered - Type B
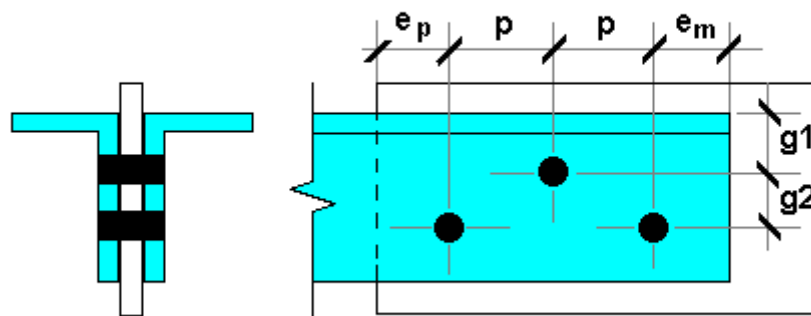
Bolted Connection Models

Eleven models are available in the **Bolted Connection Definition** Spreadsheet (Structure menu). Bolted connections are assigned to members through the **Member Bolted Connection** tab.

Long Leg This Side — **Type 1**

Type 2

Long Leg This Side — **Type 3**

Type 4

Type 5
Steel angles
Equal Legs Only

Type 6
Steel angles
Equal Legs Only
Spliced Sections

Short Leg This Side — **Type 7**

Short Leg This Side — **Type 8**

Type 9
Steel angles
Equal Legs Only
Sections ends on

Type 10
Steel angles
Equal Legs Only

Type 11
V Sections only
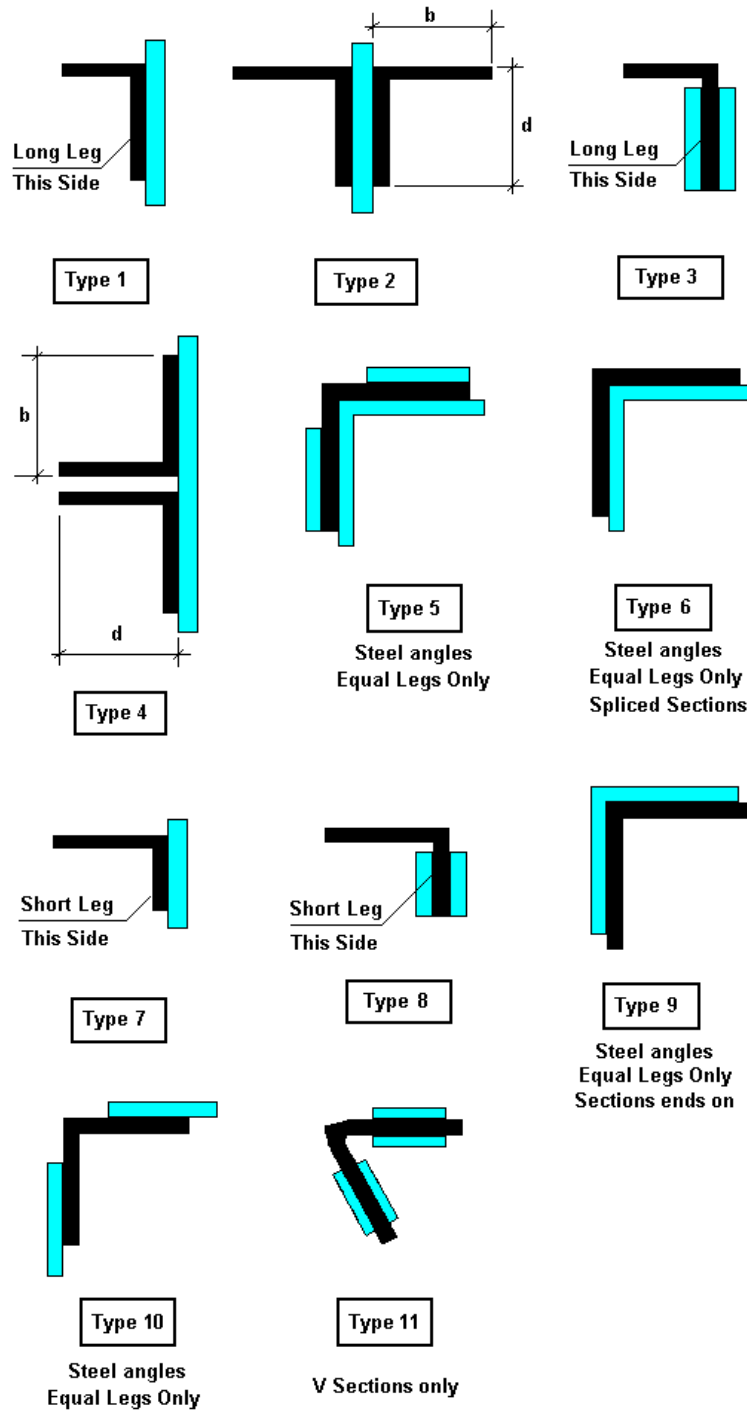
Look at this table to know what types of steel shapes can be included in the definition of bolted connection models. If the shape is invalid, a warning message will appear and analysis will be stopped.

Required Steel Shapes for Bolted Connection Models

Models	Required Section	Particularities
1 – 3 - 7 and 8	Single angle with equal or unequal legs	N/a
5 – 6 – 9 and 10	Steel angles with equal legs only	The angle that is used as a connector in model 5 must be specified in the **Bolted Connections Definition** spreadsheet, at column "Section".
		For models 6 and 9: Bolts must be defined for the smallest steel angle otherwise, the design will not be appropriate.
2 and 4	Double steel angles with equal or unequal legs (2LE, 2SL or 2LL)	N/a
11	V Sections only	N/a

See also
Bolts Layout
Bolted Connections Spreadsheet
Bolted Connection tab – Member dialog box
Checking or Designing Bolted Connections

Bolted Connection Model 1

Type 1

Example:

e_p p p e_m

$g1$
$g2$

Longest
leg

Data

Bolts Layout	Staggered A
No. of transverse lines	3
No. of longitudinal lines	2
No. of planes - Bolts	1 (1 pattern x single shear)
No. of planes - Member	1 (1 memb./pattern x 1 pattern)
No. of planes - Plate	1 (1 pl./pattern x 1 pattern)

Pattern (for the first three rows)
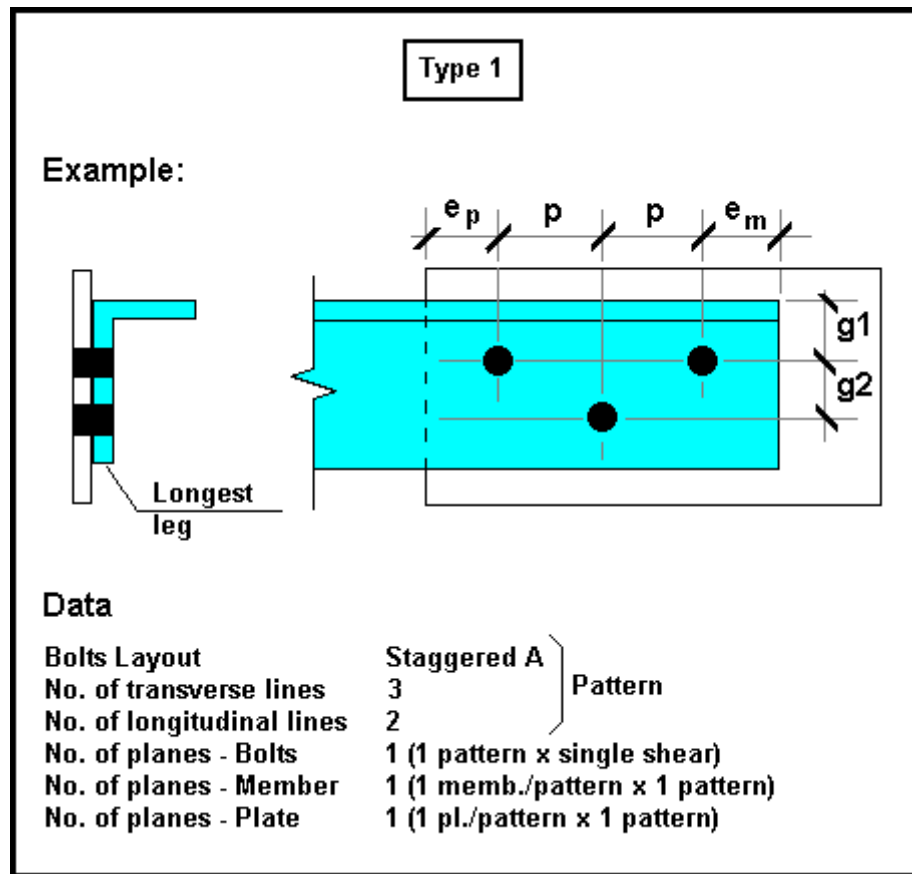
Bolted Connection Model 2

Type 2

Example:

Data

Bolts Layout	Staggered A	
No. of transverse lines	3	Pattern
No. of longitudinal lines	2	
No. of planes - Bolts	2 (1 pattern x double shear)	
No. of planes - Member	2 (2 memb./pattern x 1 pattern)	
No. of planes - Plate	1 (1 pl./pattern x 1 pattern)	

Bolted Connection Model 3

Type 3

Example:

Longest leg

e_p | p | p | e_m

g1

g2

Data

Bolts Layout	Staggered A
No. of transverse lines	3	Pattern
No. of longitudinal lines	2	
No. of planes - Bolts	2 (1 pattern x double shear)
No. of planes - Member	1 (1 memb./pattern x 1 pattern)
No. of planes - Plate	2 (2 pl./pattern x 1 pattern)

Bolted Connection Model 4

Type 4

Example:

Data

Bolts Layout	In line	
No. of transverse lines	3	Pattern
No. of longitudinal lines	2	
No. of planes - Bolts	2	(2 patterns x single shear)
No. of planes - Member	2	(1 memb./pattern x 2 patterns)
No. of planes - Plate	2	(1 pl./pattern x 2 patterns)

Bolted Connection Model 5

Type 5

**Steel Angles
with Equal Legs Only**

Example:

e_p p p e_m

g1
g2

Data

Bolts Layout	Staggered A	
No. of transverse lines	3	Pattern
No. of longitudinal lines	2	
No. of planes - Bolts	4 (2 patterns x double shear)	
No. of planes - Member	2 (1 memb./pattern x 2 patterns)	
No. of planes - Plate	4 (2 pl./pattern x 2 patterns)**	

Notes

** Legs of interior steel angle are considered as plates

Verification: The interior steel angle must be specified in the "Section" column of Bolted Connections Definition spreadsheet by double clicking in the cell.

Design: The interior steel angle that will be chosen as connector will be of the same size or bigger than the connected steel angles.

Bolted Connection Model 6

Type 6

**Steel Angles
with Equal Legs Only**

Example:

e_m p p p p e_m

g1

g2

Data

Bolts Layout	Staggered A. ⎫ Pattern
No. of transverse lines	5
No. of longitudinal lines	2
No. of planes - Bolts	2 (2 patterns x single shear)
No. of planes - Member	2 (1 memb./pattern x 2 patterns)
No. of planes - Plate	0

Note

You must define this bolted connection for the smallest section among the spliced steel angles

Bolted Connection Model 7

Type 7

Example:

e_p p p e_m

g1
g2

Shortest
Leg

Data

Bolts Layout	Staggered A	Pattern
No. of transverse lines	3	
No. of longitudinal lines	2	
No. of planes - Bolts	1 (1 pattern x single shear)	
No. of planes - Member	1 (1 memb./pattern x 1 pattern)	
No. of planes - Plate	1 (1 pl./pattern x 1 pattern)	

Bolted Connection Model 8

Type 8

Example:

Shortest Leg

e_p | p | p | e_m

g1

g2

Data

Bolts Layout	Staggered A	
No. of transverse lines	3	Pattern
No. of longitudinal lines	2	
No. of planes - Bolts	2 (1 pattern x double shear)	
No. of planes - Member	1 (1 memb./pattern x 1 pattern)	
No. of planes - Plate	2 (2 pl./pattern x 1 pattern)	

Bolted Connection Model 9

Type 9

**Steel Angles
with Equal Legs Only**

Example:

Data

Bolts Layout	Staggered A ⎱
No. of transverse lines	3 ⎰ Pattern
No. of longitudinal lines	2
No. of planes - Bolts	2 (2 patterns x single shear)
No. of planes - Member	2 (1 memb./pattern x 2 patterns)
No. of planes - Plate	0

Notes

Verification: Exterior steel angle must be specified in the "Section" column of Bolted Connections Definition spreadsheet by double clicking in the cell.

Design: Exterior steel angle that will be chosen as connector will be of the same size or bigger than the connected steel angles.

Bolted Connection Model 10
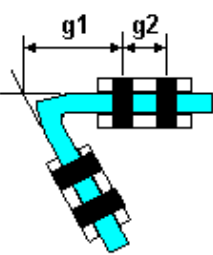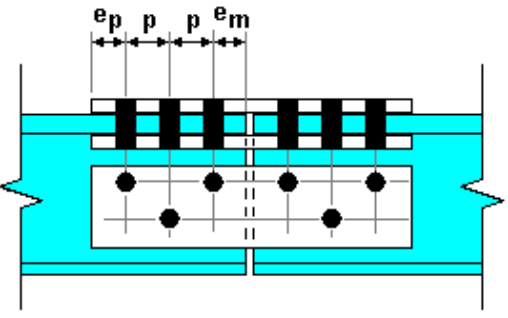
Type 10

**Steel Angles
with Equal Legs Only**

Example:

Data

Bolts Layout	In line	Pattern
No. of transverse lines	2	
No. of longitudinal lines	1	

No. of planes - Bolts   2 (2 patterns x single shear)
No. of planes - Member   2 (1 memb./pattern x 2 patterns)
No. of planes - Plate   2 (1 plate /pattern x 2 patterns)
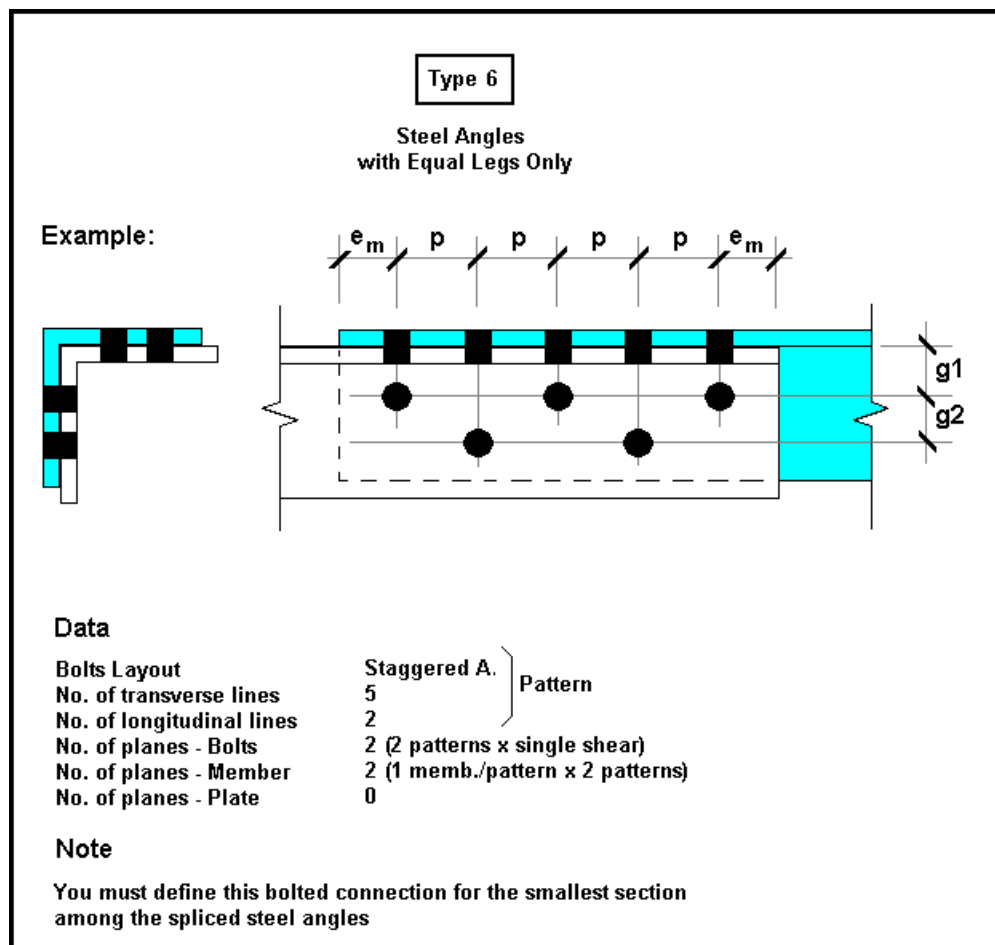
Bolted Connection Model 11

Definition of Bolted Connections

Select **Bolted Connections** heading in **Structure** menu and specify the required parameters for connections and bolts that will be used for the design of the bolted connections of tower members.

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

Column	Description	Editing
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
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

Failure Mechanisms

Member Failure Mechanisms

Design Criteria (Shear Lag)

When designing or verifying members, VisualDesign will automatically fill in the appropriate values for net area and shear lag in the **Steel Design** tab of **Member Characteristics** dialog box.

See

Steel Design tab
Transverse failure of member through bolt holes (with Shear Lag)

Member Steel Design Results

The value of Tr that is written in the **Steel Design Results** spreadsheet is calculated with the reduction factor due to shear lag for a transverse failure of member through bolt holes. This value may differ from the one that is written in the **Bolted Connection Design Results** spreadsheet. In fact, the design of bolted connections is done according to bolt shear strength, bearing and failure mechanisms without shear lag.

See

Steel Design Results spreadsheet
Bolted Connection Design Results spreadsheet

Transverse Failure of Member (with Shear Lag)

VisualDesign calculates the maximum bolt hole widths, as shown below, for a connection with bolts in line or a staggered layout of bolts. When the verification of bolted connection is done, values are written in the **Steel Design** tab of **Member Characteristics** dialog box.

For connections with bolts in line

Maximum bolt hole widths = Number of longitudinal lines * hole diam.
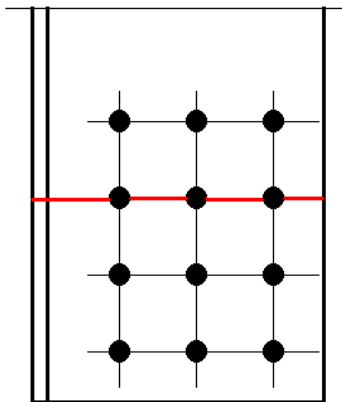
For connections with staggered layout of bolts (Type A and B):

Two types of failure can occur depending on the even or uneven number of longitudinal lines of bolts.
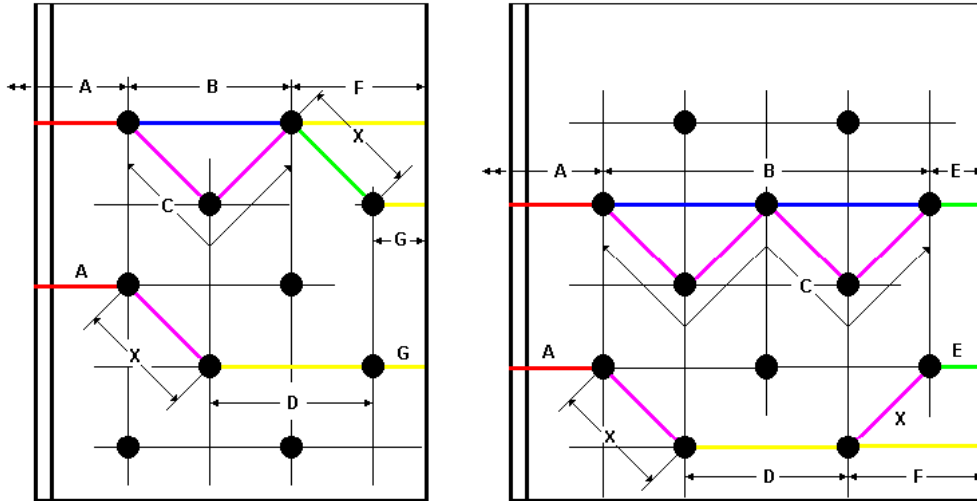
EVEN NUMBER OF LONGITUDINAL LINES:

The maximum width of holes will be the largest failure path among the following:

(La + Lb + Lf)

(La + Lb + Lx + Lg)

(La + Lc + Lf)

(La + Lc + Lx + Lg)

(La + Lx + Ld + Lg)

Where:

La = 0.5*hole diam.

Lb = (No.Long-2)/2 * hole diam

Lc = (No.Long-2) * hole diam + (No.Long-2)* $p^2/4/g2$

Ld = (No.Long-2)* hole diam

Lf = 0.5*hole diam

Lg = 0.5*hole diam

Lx = hole diam + $p^2/4/g2$

The maximum width of holes will be the largest failure path among the following:

(La + Lb + Le)

(La + Lc + Le)

(La + Lx + Ld + Lf)

(La + Lx + Ld + Lx + Le)

Where

La = 0.5 *hole diam

Lb = (No.Long-1)/2 * hole diam

Lc = (No.Long-1) * hole diam + (No.Long-1)* $p^2/4/g2$

Ld = (No.Long-3)* hole diam

Le = 0.5 hole diam

Lf = 0.5 hole diam

Lx = hole diam + $p^2/4/g2$

Failure Mechanisms - Bolted Connection

When designing bolted connections, VisualDesign uses the appropriate value for "p" and/or "g2" in such a way that failure cannot happen in a staggered layout of bolts. The failure will always be for bolts in line (transverse and longitudinal rows).

The design and verification of bolted connections is done according to the maximum shear in bolts and according to member and plate bearing strengths.

The studied bolted connection failure mechanisms occur on a single leg, so we do not consider shear lag in those mechanisms. Shear lag is verified when designing members (**Steel Design** tab).

NOTES:

VisualDesign designs bolted connections according to the maximum tension force obtained in the design. Compression is not considered.

We do not verify plate failure through bolt holes, only bearing is verified.

See also
Connection Failure with Bolts in Line
Connection Failure with a Staggered Layout of Bolts (Type A)
Connection Failure with a Staggered Layout of Bolts (Type B)

Failure Mechanism for Bolts in Line

For bolts in line connections, two failures may occur: Cases 0, A and 0, B, as shown below.
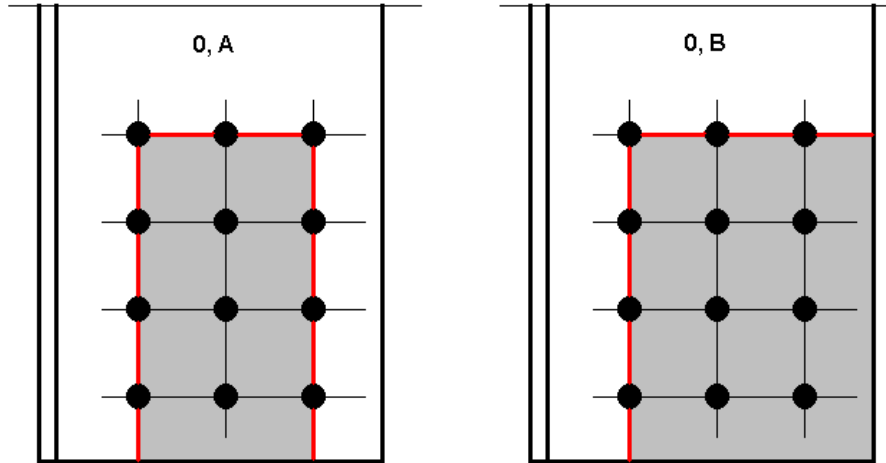
Dimensions used in our calculation:
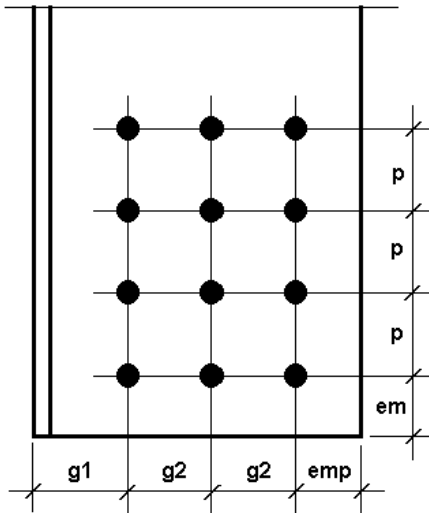
Case 0, A Failure:

Tr = 0.85 * phi * Ane * Fu * (No. Shear planes for member)

Where:

Ane = (No.Longitudinal-1)*(g2-hole.diam)*t

     + (No.Transverse-1)*(p- hole.diam)*0.6*t*2

     +(2*em- hole.diam)*0.6*t

Case 0, B Failure:

Tr = 0.85 * phi * Ane * Fu * (No. Shear planes for member)

Where:

Ane = (No.Longitudinal-1)*(g2- hole.diam)*t

+(emp- hole.diam /2)*t + (No.Transverse-1)*(p- hole.diam)*0.6*t

+(em- hole.diam /2)*0.6*t

Failure of Connection with a Staggered Layout (Type A) of Bolts

Failure in a connection having bolts in staggered layout of type A may be of four types, depending on an odd or even number of longitudinal and/or transverse lines of bolts.

We have to verify the following cases: 1- PI, 1-II, 1-PP and 1-IP.

1, PI
Longitudinal even, Transverse uneven

1, II
Longitudinal uneven, Transverse uneven

1, PP
Longitudinal even, Transverse even

1, IP
Longitudinal uneven, Transverse even

Case 1, PI (Even Longitudinal, Uneven Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = [(No.Trans-1) * p + em – (No.Trans/2)*hole.diam] * 0.6

Lb = (No.Long – 2) * g2 – (No.Long-2)/2 * hole.diam

Lc = (No.Long-2) * g2 – (No.Long-2)* hole.diam + (No.Long-2)*p²/4/g2

Ld = g2 + emp - 0.5 hole.diam

Le = g2 – hole.diam + p²/4/g2

Lf = ((No.Trans-2) * p – (No.Trans-2)/2 * hole.diam + em) * 0.6

Lg = emp – 0.5 hole.diam

And

Lt = Minimum Lt among the failure paths.

Case 1, II (Uneven Longitudinal, Uneven Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = [(No.Trans-1) * p + em – (No.Trans/2)* hole.diam] * 0.6

Lb = (No.Long – 1) * g2 – (No.Long-1)/2 * hole.diam

Lc = (No.Long-1) * g2 – (No.Long-1)* hole.diam + (No.Long-1)* p²/4/g2

Ld = emp - 0.5 hole.diam

Le = [(No.Trans-1) * p + em – (No.Trans)/2 * hole.diam] * 0.6

And

Lt = Minimum Lt among the failure paths.

Case 1, PP (Even Longitudinal, Even Transverse)

Tr = phi * 0.85 * Lt min * t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = [(No.Trans-2) * p + em − (No.Trans-2)/2*hole.diam] * 0.6 + g2 − hole.diam

   $+ p^2/4/g2$

Lb = (No.Long − 2) * g2 − (No.Long-2)/2 * hole.diam

Lc = (No.Long-2) * g2 − (No.Long-2)* hole.diam + (No.Long-2)*p^2/4/g2

Ld = emp - 0.5 hole.diam

Le = [(No.Trans-1) * p + em − (No.Trans-1)/2 * hole.diam] * 0.6

And

Lt = Minimum Lt among the failure paths.

Case 1, IP (Uneven Longitudinal, Even Transverse)

Tr = phi * 0.85 * Lt min * t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La =   [(No.Trans-2) * p + em − (No.Trans-2)/2*hole.diam] * 0.6 + g2 − hole.diam

+ p^2/4/g2

Lb = (No.Long − 3) * g2 − (No.Long-3)/2 * hole.diam

Lc = (No.Long-3) * g2 − (No.Long-3)* hole.diam + (No.Long-3)*p^2/4/g2

Ld = g2 + emp - 0.5 hole.diam

Le = g2 - hole.diam + p^2/4/g2

Lf = [(No.Trans-2) * p + em − (No.Trans-1)/2 * hole.diam] * 0.6

And

Lt = Minimum Lt among the failure paths.

Failure of Connection with a Staggered Layout (Type B) of Bolts

Failure in a connection having bolts in a staggered layout of type B may be of four types, depending on an odd or even number of longitudinal and/or transverse lines of bolts.

We have to verify the following cases: 2- PI, 2-II, 2-PP and 2-IP.
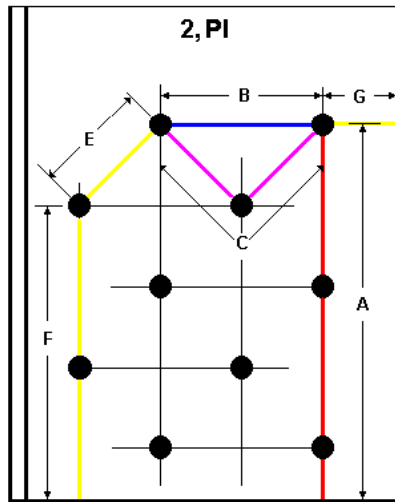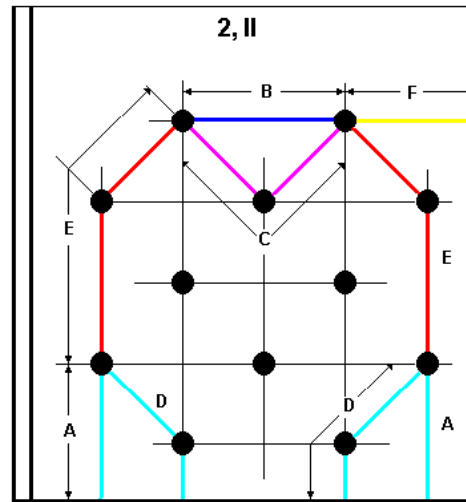
2, PI
Longitudinal even, Transverse uneven

2, II
Longitudinal uneven, Transverse uneven

2, PP
Longitudinal even, Transverse even
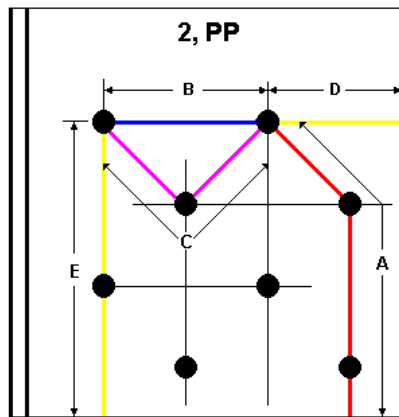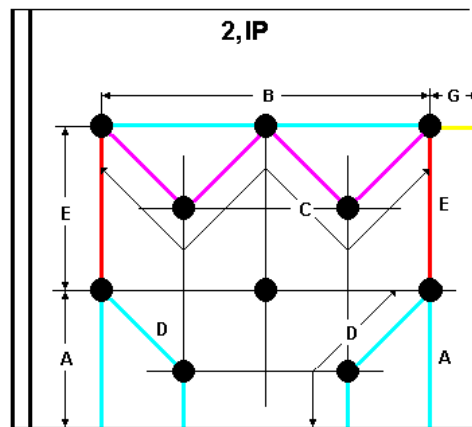
2, IP
Longitudinal uneven, Transverse even

Case 2, PI (Even Longitudinal, Uneven Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = ((No.Trans-1) * p + em – (No.Trans/2)*hole.diam)*0.6

Lb = (No.Long – 2) * g2 – (No.Long-2)/2 * hole.diam

Lc = (No.Long-2) * g2 – (No.Long-2)* hole.diam + (No.Long-2)*p²/4/g2

Le = g2 – hole.diam + p²/4/g2

Lf = ((No.Trans-2) * p – (No.Trans-2)/2 * hole.diam + em) * 0.6

Lg = emp – 0.5 hole.diam

And

Lt = Minimum Lt among failure paths.

Case 2, II (Uneven Longitudinal, Uneven Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = (p + em – 0.5 hole.diam) * 0.6

Lb = (No.Long – 3) * g2 – (No.Long-3)/2 * hole.diam

Lc = (No.Long - 3) * g2 – (No.Long-3)* hole.diam + (No.Long-3)*p²/4/g2;
Lc >= 0.0

Ld = g2 - hole.diam + p²/4/g2 + (em – 0.5 hole.diam)*0.6

Le = [(No.Trans-3) * p - (No.Trans-3)/2 * hole.diam] * 0.6 + g2 – hole.diam + p²/4/g2

Lf = (g2 + emp – 0.5 hole.diam)

And

Lt = Minimum Lt among failure paths.

Case 2, PP (Even Longitudinal, Even Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La =   [(No.Trans-2) * p + em – (No.Trans-1)/2* hole.diam]*0.6 + g2 – hole.diam

+ p²/4/g2

Lb = (No.Long – 2) * g2 – (No.Long-2)/2 * hole.diam

Lc = (No.Long-2) * g2 – (No.Long-2)* hole.diam + (No.Long-2)*p²/4/g2

Ld = g2 + emp - 0.5 hole.diam

Le = [(No.Trans-1) * p + em – (No.Trans-1)/2 * hole.diam] * 0.6

And

Lt = Minimum Lt among failure paths.

Case 2, IP (Uneven Longitudinal, Even Transverse)

Tr = phi * 0.85 * Lt min* t * Fu * No. Member shear planes

Where:

t = Thickness of steel angle leg

Fu = Member specified tensile strength

La = (p + em – 0.5 hole.diam) * 0.6

Lb = (No.Long – 1) * g2 – (No.Long-1)/2 * hole.diam

Lc = (No.Long - 1) * g2 – (No.Long-1)* hole.diam + (No.Long-1)*p²/4/g2

Ld = g2 - hole.diam + p²/4/g2 + (em – 0.5 hole.diam)*0.6

Le = [(No.Trans-2) * p - (No.Trans-2)/2 * hole.diam] * 0.6

Lg = emp – 0.5 hole.diam

And

Lt = Minimum Lt among failure paths.

Members

The Steel Design tab

If you checked the "Activate design criteria" box in **Member Characteristics** dialog box, complete parameters for the design or verification of steel members.

Look at this table to learn more about parameters included in the **Steel Design** tab.

Field	Description
Design Parameters	
Design or Verification	This shaded field indicates the type of analysis that is going to be carried on, as specified in the steel specification spreadsheet.
Specification	Select a steel specification among the drop-down list box. VisualDesign will design this member according to it. Press the […] button next to this field to add a new specification.

Field	Description
Design Group	Select the design group that applies to this member, if required. (A steel specification is chosen within a member design group.). Press the […] button next to this field to add a new design group.

Lateral Supports to avoid buckling

Top and Bottom Fibre	Specify lateral supports, at the top and bottom of section. Tick off the appropriate boxes: Node i, Node j and/or Continuous lateral support. Refer to topic Lateral Support
Position of load	By default, the load is applied at the centre of section. If this is not the case, choose among options: Top or Bottom.

Cantilever

Automatic	Automatic calculation of factor Kux. VisualDesign automatically calculates Kux factor in particular cases. Warning, if you disable the "Automatic" option, you must enter a value for Kux. If you don't, VisualDesign will consider a factor of 10 as compression length and this member will be a cantilever even if you have not specified it. See topic Cantilever
Kux	Specific coefficient used for the calculation of cantilever effective compression lengths. It depends upon the position of lateral supports on the member and the position of load. The default value is 2.5.
At node	If Kux is automatic, indicate the location of the free end of cantilever: Node i or j.

Effective Compressive Length	See topic Automatic Calculation of Kx, Ky, Kt and Kz
Kx Factor	Effective compressive length factor relative to strong axis. If "Auto" is checked, Kx coefficient is automatically calculated.
Ky Factor	Effective compressive length factor relative to weak axis. If "Auto" is checked, Ky coefficient is automatically calculated.
Kt Factor	Effective compressive length factor used for the computation of buckling due to torsion. If "Auto" is checked, Kt is automatically calculated according to the **Steel** tab of **Project Configuration**.
Kz Factor	Effective compressive length factor relative to orthogonal axis system. Kz is used for single steel angles and an orthogonal axis system must have been selected in the member tab. If "Auto" is checked, Kz is automatically calculated according to the **Steel** tab of **Project Configuration**.

Field	Description

Effective Net Area, with or without reduction factor

Width of Hole	If bolts are used as connections, enter the total width of holes for this member.
A'ne = Ane x (Rf)	Reduction factor used for the calculation of net area. Ex.: A'ne = Ane x 0.80
Max. Slenderness	Enter the member maximum slenderness if different from default value, 200.

Stiffeners/Intermittent Fillers (2L)

Spacing	Enter the spacing between I beam web stiffeners for shear capacity calculation <u>OR</u> enter the spacing between intermittent fillers for double steel angles. A value of 0 means that there is no stiffener or filler.
Ft = 0	If Ft = 0, the tension field component of post-buckling stress will not be considered in the calculation of shear resistance of the beam. Refer to clause 10.10.5 – CAN/CSA-S6-00 Standard. N.B. If Ft=0, the bending moment and shear interaction equation will not be verified.
Km Factor	This factor is used in the calculation of effective compression length of double steel angles with intermittent fillers. Refer to clause 6.2.4.3 CAN/CSA-S37-01 Standard..

Allowable Deflection

Lx/	Enter a deflection criterion relative to strong axis.
Ly/	Enter a deflection criterion relative to weak axis.
HSS or Solid Round Shape (Rod)	Tick off this box if you want VisualDesign to account for stress relaxation in the calculation of axial compression according to clause 13.3.1 (S16-01 Standard)

See also
Member Dialog Box
Steel Member Groups
Steel Specifications
Steel tab (Project Configuration)
Steel Design Module
Automatic Calculation of Kx, Ky, Kt and Kz

Lateral Supports against Buckling

Lateral support end conditions are used to calculate members' unsupported lengths. Lateral support may be provided at nodes i and j or continuously, at top and/or bottom fibres.

When a continuous lateral support is provided at the <u>top or bottom</u> of a member, the effective compression length factor Ky will be equal to zero. It means that the member weak axis is supported on the whole length.

<u>It is important to specify lateral support end conditions to get the right bending capacity of the member.</u>

Example showing lateral supports at node i and j, at the top and bottom of a beam. (VisualDesign™ default values):
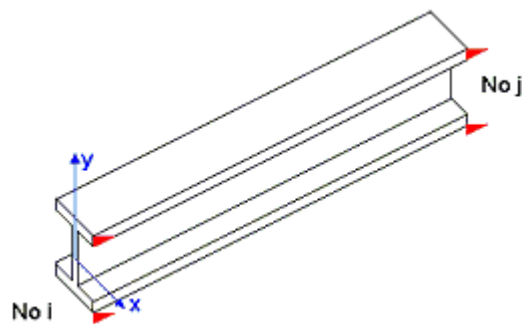
Beam with continuous restraints at the top fibre and lateral supports at bottom fibre at nodes i and j:
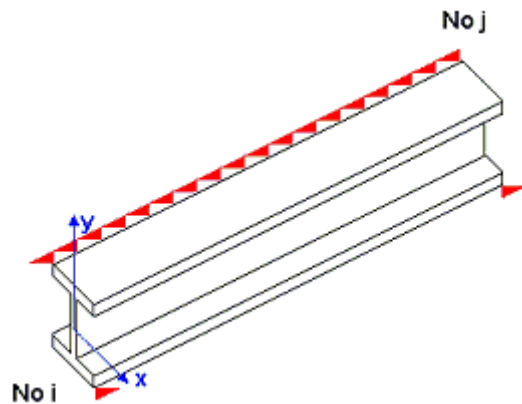
Joist floor:

When modeling a joist floor, do one of the following:

- Split the girder where the (virtual) joists are attached and supply lateral supports at nodes i and j of each partial girders.

- Do not split the girder but supply a continuous lateral support at the top of the girder.

See also
The Member Dialog Box

Cantilever

Steel Design tab of **Member Characteristics** Dialog Box:

User must specify the location of cantilever in the *Cantilever* drop-down list box. Options are: Node i, Node j, and Not applicable. If the cantilever is loaded, the position of load must also be specified (Top, Centre (default value) or Bottom).
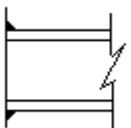
Factor **Kux** represents the effective compression length factor for cantilevers. VisualDesign automatically calculates some of these factors but some particular cases are not.

Warning!!! VisualDesign cannot process cases where only one lateral support is provided at the top flange of a continuous beam, at the location of support. For these particular cases, the user must specify the factor Kux. See Example 2 below and Table 7.2.

Kux factors are listed in Table 7.2 of Picard et Beaulieu (1991) *Calcul des charpentes d'acier.* An identical table (fig. 5.11) is also available in Theodore V. Galambos *Stability Design Criteria for Metal Structures* (5th edition), page 207.

Table 7.2 - Effective Compression Length Factor for Cantilevers

These values are valid for a concentrated load or uniform load.

Lateral Supported End Condition		Factor K	
Supported End	**Free End**	**Load applied to Top Flange**	**Other Cases**
	I	1.4	0.8
	I˙	1.4	0.7
	I⦂	0.6	0.6

Lateral Supported End Condition		Factor K	
Supported End	Free End	Load applied to Top Flange	Other Cases
		2.5	1.0
		2.5	0.9
		1.5	0.8

Cases not considered by VisualDesign:

For this type of lateral support, factor Kux must be entered manually in the appropriate field in the **Steel Design** tab.

If you activated the "Manual" option, please enter a value for Kux. If you don't, VisualDesign will put a factor of 10, as default value!

Lateral Supported End Condition		Factor K	
Supported End	Free End	Load applied to Top Flange	Other Cases
		7.5	3.0
		7.5	2.7
		4.5	2.4

Cases not considered by VisualDesign, nor Picard & Beaulieu or Galambos:

Modeling Valid Cantilevers

Example 1: Continuous beam composed of members M1, M2 and M3. Cantilever is located at the end of member M3 as shown below. The load is applied at top of section.
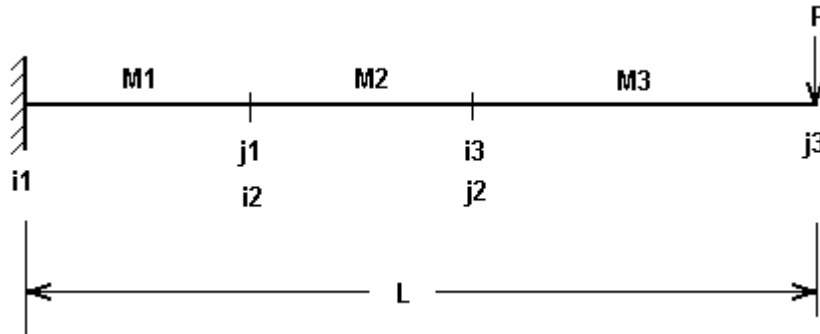
Select the **Steel Design** tab (**Member Characteristics** dialog box) and enter the following lateral supports and cantilever options for each member:

Member 1: Set lateral supports to Node i only. **Cantilever** option is set to *Not applicable*.

Member 2: No lateral supports and cantilever option is set to *Not applicable*.

Member 3: No lateral supports and cantilever option is set to *Node j*. Choose option *Top* for **Position of load**.

Example 2: Continuous beam composed of members M1 and M2. Cantilever is located at the end of member M2 and load is applied at top of section. At intermediate support, the continuous beam has a lateral support at top flange only.
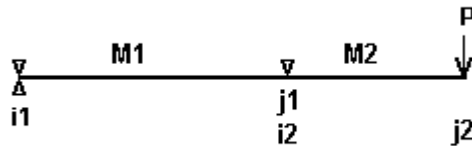
In the **Steel Design** tab, enter required parameters to calculate the beam strength accurately.

Member 1: Lateral support is provided at top fibre, at node i and j, and at bottom fibre, at node i only. **Cantilever** option is set to *Not applicable*.

Member 2: Lateral support is provided at the top fibre of node i only. **Cantilever** option is set to *Node j* and the **Position of load**, to *Top*. Enter a **Kux** factor of 7.5.

Effective Net Area

The design is done according to your design criteria (**Steel Design** tab of **Member Characteristics** dialog box).

The net area is calculated according to the sum of boltholes as specified in the "Width of holes" field. In addition, you can consider shear lag by entering the appropriate reduction factor that will be applied to the calculated net area.

According to clause 12 of the standard S16-01:

$$A_{ne} = A_g - L_t e$$

Where Ag = Gross area of leg of connected flange;

Lt = maximum width of all connection holes;

e = thickness of leg or connected flange;

Reduction of effective net area – Shear lag

According to clause 12.3.3 (*Shear Lag*) of the standard S16-01:

$$A'_{ne} = FrA_{ne}$$

"Fr" is a factor that reduces the member net area because of shear lag. The maximum value for this factor is 0.20 or 20 %.

See also
Steel Design Criteria

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

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 group afterwards 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.
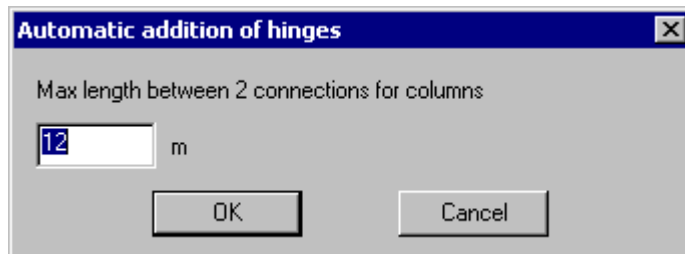
- To look at created hinges, activate the option "End conditions" in the **Attributes** tab of **View Options** dialog box.

- Use function **Auto Group**, located in menu **Structure / Groups**. VisualDesign will automatically group these columns according to the positions of hinges.

Member Steel Design Spreadsheet

This spreadsheet is part of the Members spreadsheet if you own the **Steel Design** module and if member design criteria were activated. Use the spreadsheet to consult and sort data.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Member number.	Single click
Specification	Choose a steel specification in the drop-down list box. Specifications are defined in the Steel Specifications Spreadsheet	Double-click

Column	Description	Editing
Group	Choose a member design group, if desired. (Specifications are assigned to Design Groups.)	Double-click
Kx computation	Choose an automatic or manual computation of effective compression length factor, relative to strong axis.	Double-click
Kx	Effective compression length factor relative to strong axis. For an "Auto" computation, the Kx coefficient is automatically calculated.	Single click
Ky computation	Choose an automatic or manual computation of effective compression length factor, relative to weak axis.	Double-click
Ky	Effective compression length factor relative to weak axis. For an "Auto" computation, the Ky coefficient is automatically calculated.	Single click
Kz computation	Choose an automatic or manual computation of Kz. If the Automatic box is activated, VisualDesign will consider the specified value in the **Steel** tab of **Project Configuration**. See also topic Automatic Calculation of Kx, Ky, Kt and Kz.	Double-click
Kz	Effective compression length factor in the orthogonal z-axis.	Single click
Kt computation	Choose an automatic or manual computation of Kt. If the Automatic box is activated, the Kt factor is automatically calculated according to the **Steel** tab of **Project Configuration**. See also topic Automatic Calculation of Kx, Ky, Kt and Kz.	Double-click
Kt	Effective compression length factor for the calculation of buckling due to torsion.	Single click
Lux Bottom Noi	If the shape bottom flange is laterally supported at node i, choose option [x].	Double-click or Space bar
Lux Bottom Cont	If the shape has continuous lateral support at the bottom flange, choose option [x].	Double-click or Space bar
Lux Bottom Noj	If the shape bottom flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Lux Top Noi	If the shape top flange is laterally supported at node i, choose option [x].	Double-click or Space bar

Column	Description	Editing
Lux Top Cont	If the shape has continuous lateral support at the top flange, choose option [x].	Double-click or Space bar
Lux Top Noj	If the shape top flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Hole Width	Width of the hole if bolts are used.	Single click
Reduction of Ane	Reduction factor used for the calculation of net area. Ex: A'ne = Ane x 0.80	Single click
a: Stiffener or a: Intermittent fillers (2L)	I Beam: Enter the spacing between web stiffeners for shear capacity calculation. <u>OR</u> Double steel angles: Enter the spacing between intermittent fillers along the member.	Single click
Ft = 0	If you do not want to consider the tension field component of post-buckling stress in the calculation of shear resistance of the beam, choose option [x]. Refer to clause 10.10.5 – CAN/CSA-S6-00 Standard. N.B. If Ft=0, moment and shear interaction equation will not be verified.	Double-click or Space bar.
km : intermittent fillers	Factor used in the calculation of effective compression length of double steel angles with intermittent fillers. Refer to clause 6.2.4.3 CAN/CSA-S37-01 Standard.	Single click
Stress-Relieved	For HSS shape or round sections of class H: To consider the stress relaxation for the calculation of axial compression (clause 13.3.1), choose option [x].	Double-click or Space bar.
KL/r max	Enter the maximum slenderness for a member if it is different from the default value of 200.	Single click
Lx/	Enter a deflection criterion for strong axis.	Single click
Ly/	Enter a deflection criterion for strong axis.	Single click
Cantilever	Indicate if cantilever is located at node i or node j. If any of these cases, choose *Not applicable*. See topic Cantilever.	Double click
Kux Calculation	Automatic calculation or user-defined.	Double click

Column	Description	Editing
Kux	Kux value: Specific factor used for the calculation of cantilever effective compression length. This factor also depends on the load position.	Single click
Position of Load	By default, the load is applied at the centre of section. If this is not the case, choose among options: *Top* or *Bottom*.	Double click

See also

The Tower Design module
Cantilever
The Steel tab (Project Configuration)

Design of Bolted Connections

Limitations

- Concentric connections with bolts of the bearing type only.

- For the study of failure mechanisms for a staggered layout of bolts, there must be at least two longitudinal lines and two transverse lines of bolts.

- Only the maximum tension force is considered for the design of bolted connections. Compression is not.

Checking or Designing Bolted Connections

Verification of Bolted Connections

The bolted connection to be verified must be defined in the Bolted Connections Definition spreadsheet. Then, its number must be selected in the **Bolted Connection** tab.

The design criteria for bolted connections must be selected in the Steel specification.

Design of Bolted Connections

The type of bolted connection must be selected in the **Bolted Connection** tab, at node i and node j.

The design criteria for bolted connections must be selected in the Steel specification.

Steel Specification for bolted connections

Group: Structural Data

Column	Description	Editing
Analysis of Connections	Choose the type of analysis that will be done for connections: None, Verification or Design.	Double-click
Type of Connection	Type of bolted connection: Bearing or Slip-Resistant.	Double-click
Type of Hole	Specify if holes had been punched or drilled.	Double-click
Contact Surface	Specify the contact surface of connections: None, class A, class B or class C.	Double-click
Selection of Bolts	Choose the bolts that can be used for the design of bolted connections.	Double-click

When the Steel specification is completed, select one member or more and open the **Member Characteristics** dialog box.

- Select the **Member** tab, enter parameters, and activate design criteria.

- Select the **Steel Design** tab, enter parameters and select a specification.

- Verification: Select the **Bolted Connection** tab and assign bolted connection number to member(s) end nodes.

- Design: Select the Bolted Connection tab and select a connection model to member(s) end nodes.

Bolted Connections Tab – Members

Complete this tab to verify or design member bolted connections.

Description of the dialog box:

Field	Description
Bolted Connections – Node i and Node j	
Number	Connection number at node i or node j.
Model	Choose node i and j connection model if you want VisualDesign to design it for you.
Tower Members	
Equation for KLx/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in x which takes into account the type of bracing member and the number of bolts in the member end connections.
Verify T/C ratio	Activate this option if you want VisualDesign to compute the effective compressive length KLx according to clause 6.2.3.3 (S37-01Standard). Refer to Ratio T/C
Equation for KLy/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in y which takes into account the type of bracing member and the number of bolts in the member end connections.
Verify T/C ratio	Activate this option if you want VisualDesign to compute the effective compressive length KLy according to clause 6.2.3.3 (S37-01Standard). Refer to Ratio T/C
Equation KLz/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in the minor axis system.
	If single steel angles are not oriented in an orthogonal axis system, the drop-down list box will be shaded.

Member Bolted Connections Spreadsheet

This spreadsheet, located in **Structure / Members**, is available if you own the **Steel Design** module and if design criteria were activated in the **Member tab** (**Member Characteristics** Dialog box).

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Member number.	No
Connection Model No. i	Connection model at node i.	Double-click
Connection Number No. i	Connection number at node i for the design of tower only.	Double-click
Connection Model No. j	Connection model at node j.	Double-click
Connection Number No. j	Connection number at node j for the design of tower only.	Double-click
Equation for KLx/r	According to S37-01 standard (towers): Equation that is used for calculating the effective length in x, considering the number of bolts of member end connections.	Double-click
KLx/r: Verify T/C ratio	If you want to verify the T/C ratio according to strong axis, choose option [x]. Refer to Ratio T/C	Double-click or Space bar
Equation for KLy/r	According to S37-01 standard (towers): Equation that is used for calculating the effective length in y, considering the number of bolts of member end connections.	Double-click
KLy/r: Verify T/C ratio	If you want to verify the T/C ratio according to weak axis, choose option [x]. Refer to Ratio T/C	Double-click or Space bar
Equation for KLz/r	According to S37-01 standard (towers): Equation that is used for calculating the effective length in z, considering the number of bolts of member end connections. If single steel angles are not oriented in an orthogonal axis system, the drop-down list box will not be activated.	Double-click

Steel Design Groups

Steel Design Groups - Members

Use the **Group Members** function located in **Structure / Group** menu to quickly create design groups and assign them to selected members.

Group: Structural Data

Column	Description	Editing
ID	ID number for this group. Automatically calculated	No
Number	Name of this design group (16 alphanumerical characters).	Single click
Specification	Choose the steel specification that will be applied to this group.	Double-click

See also

The Group function
Steel Specification spreadsheet
Steel Design Procedure

Grouping Elements

You must specify steel or timber specifications before grouping members.

This tool, available in **Structure** menu at **Groups** heading, allows you to create design groups for members to get a more practical design.

To group members:

- Select members that you wish to group together (keep the [Ctrl] key down while you select members with your mouse);

- Select the **Group members** function under **Structure/Groups** menu or use shortcut keys [Ctrl]+G.

- A dialog box will appear. Give a name to this group and choose the steel or timber specification for this design group.

- Press OK.
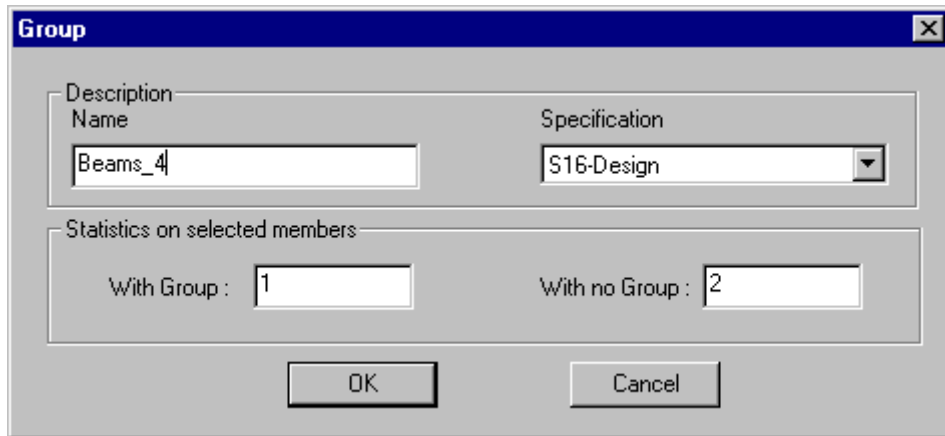
Group names will be written in the **Steel Design Groups** or **Timber Design Groups** spreadsheets that can be accessed through **Structure** menu under **Groups**.

Grouping Members Automatically

This tool, which automatically generates member groups, allows a more practical design. This function is available for grouping steel or timber members. Specifications must be assigned to members before grouping them. Members will be grouped together if they are assigned to the same specification.

Procedure:

- Select **Structure/Groups / Auto Group**. This dialog box will appear on screen:
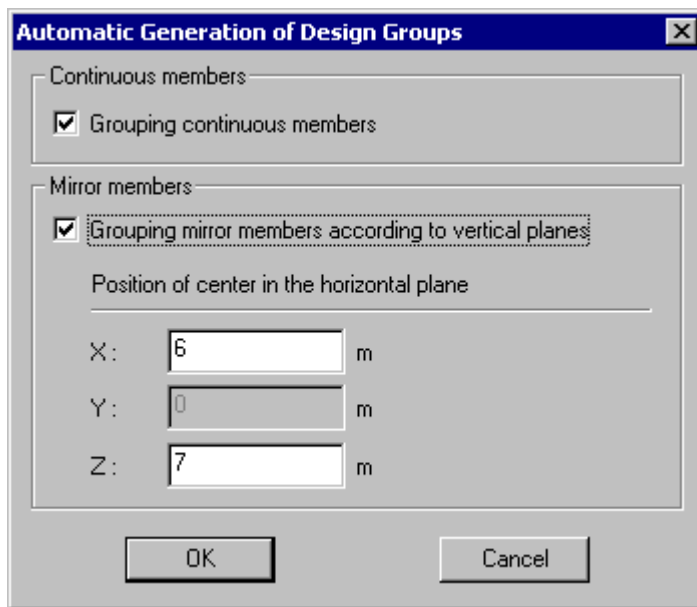
- Members can be grouped according to one of these options:

  - Grouping continuous members. These members must be rectilinear and assigned to the same specification.

  - Grouping mirror members according to vertical planes. The user can specify the horizontal position of the mirror plan.

- The specification assigned to a group is corresponding to the one that was assigned to the first member composing this group. The group number corresponds to the number of the first member part of this group.

Viewing a Group of Members

If you want to look at a group that was just created:

- Select the **Member Groups** spreadsheet under **Structure/Groups**.

- Highlight the line in the spreadsheet that corresponds to the group that you want to look on the screen.

- Click OK.

Members that are part of this group will be highlighted on screen. If you do not like the way members are grouped, delete lines in the **Member Groups** spreadsheet and use the function **Group** under **Structure / Groups**.

Steel Specifications

Sections' Groups

This spreadsheet, available in **Structure / Specifications**, allows creating groups of sections that will be used for the design. A group of sections must be selected afterwards in a steel or timber specification, before launching a design. During the optimization process, VisualDesign will search among the sections that are included in the selected group of sections. Users can include many types of shapes in the same group of sections (W, C, 2L, rectangular, round, etc.) through a selection tree.

A group of sections MUST be selected in the steel specifications spreadsheet or timber specifications spreadsheet.

The default group of sections is "Canada". This group includes sections and shapes that are available in Canada, as indicated in the column "Canada" in the shapes spreadsheets (**Common** menu).

See also
Steel Specifications
Generator of Steel Specifications
Timber Specifications

Steel Specification Generator

A generator of steel specifications is available in **Structure / Specifications / Auto Generation- Steel**. The dialog box is composed of a general section, a selection tree and options for managing generated specifications.

A building code or Standard must be selected in the general section along with types of analyses: Verification or/and Design. Then, types of shapes can be selected through check boxes in the expansion tree. Generated steel specifications can overwrite existing ones or be added to existing ones.
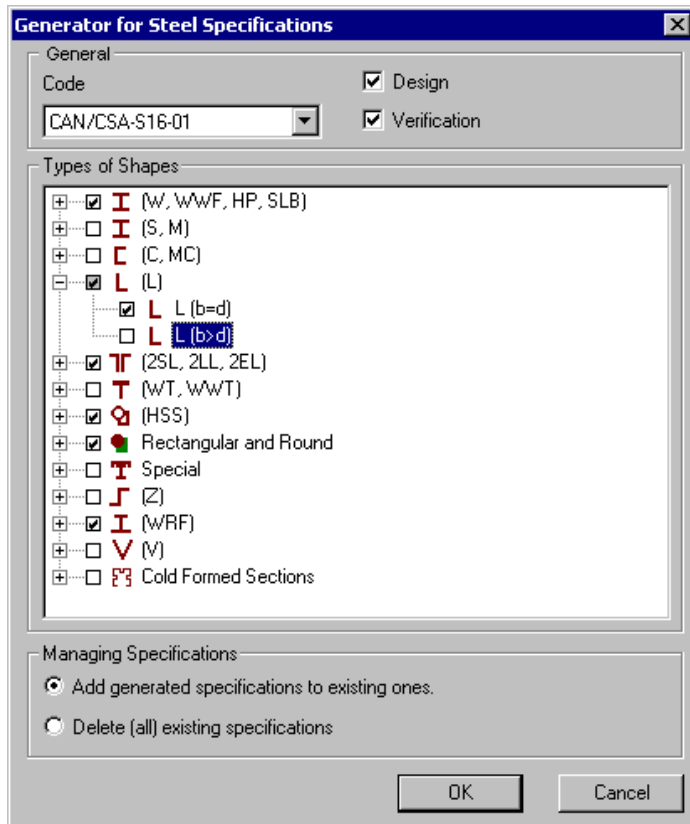
VisualDesign creates as many specifications as selected types of analyses and types of shapes.

When the OK button is clicked, generated steel specifications will be listed in the **Steel Specifications** spreadsheet.

	Number	Code	Type of analysis	Optimization	Group of sections	Shape	Class max	Maximum height mm
1	S16 WWF Des 011	CAN/CSA-S16-01	Design	Area	Canada	WWF	3 : Noncompact	5000.00
2	S16 WWF Ver 013	CAN/CSA-S16-01	Verification	Area	Null	WWF	3 : Noncompact	5000.00
3	S16 W Des 015	CAN/CSA-S16-01	Design	Area	USA	W	3 : Noncompact	5000.00
4	S16 W Ver 017	CAN/CSA-S16-01	Verification	Area	Null	W	3 : Noncompact	5000.00
5	S16 HP Des 019	CAN/CSA-S16-01	Design	Area	USA	HP	3 : Noncompact	5000.00
6	S16 HP Ver 021	CAN/CSA-S16-01	Verification	Area	Null	HP	3 : Noncompact	5000.00
7	S16 SLB Des 023	CAN/CSA-S16-01	Design	Area	Canada	SLB	3 : Noncompact	5000.00
8	S16 SLB Ver 025	CAN/CSA-S16-01	Verification	Area	Null	SLB	3 : Noncompact	5000.00
9	S16 M Des 027	CAN/CSA-S16-01	Design	Area	USA	M	3 : Noncompact	5000.00

(64)

Specification Numbers

Each specification is numbered with respect to the following nomenclature:

Standard-Shape-Analysis-Number, where Des = design and Ver = verification.

Groups of sections

"Canada" or "US" groups of sections are assigned to generated design specifications according to the availability of sections, as indicated in all VisualDesign sections spreadsheets. Groups of sections are not required for verification.

W and HP Shapes

Since the year 2000, the Canadian steel manufacturer Algoma Steel inc, is no longer producing W and HP shapes. This is why the US group of sections is selected by default in the steel specification spreadsheet. To learn more about steel shapes availability, please read the endpaper of CAN/CSA-S16-01 standard (Handbook of Steel Construction from CISC), page 6-38 and following.

Generated Parameters

If Canada is selected in the **Preferences** tab (**Project Configuration**), the following parameters will be initialized in the **Steel Specifications** spreadsheet.

Activated Shape	Default Group of section	Default Material
WWF	Canada	G40.21M-350W
W	USA	ASTM-A572-50
HP	USA	ASTM-A572-50
M	USA	ASTM-A572-50
S	USA	ASTM-A572-50
SLB	Canada	G40.21M-350W
C	Canada	G40.21M-350W
MC	USA	ASTM-A572-50
L	Canada	G40.21M-350W
2L	Canada	G40.21M-350W
WWFT	Canada	G40.21M-350W
WT	USA	ASTM-A572-50
HSS	USA	ASTM-A500-C50
HSS - Pipe	Canada	ASTM-A53-B35
Solid Rectangular	Nil	ASTM-A572-50

Activated Shape	Default Group of section	Default Material
Solid Round	Nil	ASTM-A572-50
Z	Nil	G40.21M-350W
WRF	Canada	G40.21M-350W
V	Canada	G40.21M-350W

Steel Specifications Spreadsheet

This spreadsheet is completed if the steel specifications generator has been used. Assign specifications to members.

Specifications must be assigned to members that will be design or verify. The type of analysis (design or verification) and design code or Standard must be specified, among others.

Design Specifications

A steel material, a group of sections and a type of shape must be selected in each specification. Therefore, if different shapes are composing the structure, specifications must be added in the spreadsheet because each type of shape must possess its specification.

Adding specifications

Insert lines at the end of the spreadsheet (or copy/paste a line) and give a name to each one. Select the type of analysis, design code, a group of sections and a steel shape. Then, assign specifications to members. During optimization, VisualDesign will select the specified type of shape among the sections that are listed in the group of sections.

Member Design Groups

A specification must be assigned to each design group. The same specification can be assigned to many design groups. Refer to the Design Groups spreadsheet.

Steel Specifications Spreadsheet

	Number	Code	Type of analysis	Optimization	Group of sections	Shape	Material	HSS t(design)	Interaction
1	S16-Design	CAN/CSA-S16-01	Design	Area	Canada	W	350G/W/WT/AT	1.0t CISC/Can	Standard
2	S16-Vérif.	CAN/CSA-S16-01	Verification	Area	Canada	W	Null	1.0t CISC/Can	Standard
3	S6-Design	CAN/CSA-S6-88	Design	Area	Canada	W	350G/W/WT/AT	1.0t CISC/Can	Standard
4	S6-Vérif.	CAN/CSA-S6-88	Verification	Area	Canada	W	Null	1.0t CISC/Can	Standard

Specifications used for a verification

The selection of a shape, material and group of section is not required. VisualDesign will verify assigned shapes according to the chosen code or Standard.

During modeling

For each member that will be designed, select either a design group or a specification in the **Steel Design** tab (**Member Characteristics** dialog box). For each member that needs to be verified, select a check specification.

Look at the table below to know the definition of parameters that are included in this spreadsheet.

Group: Structural data

Column	Description	Editing
Specification ID	Automatically calculated	No
Number	16 alphanumerical characters	Single click
Code	Choose the standard that will be used for the design or verification of the structure. For bridge Evaluation, choose code S6-00.	Double-click
Type of Analysis	Indicate in which case the specification will be used for: Design or Verification. If you chose code S6-00, option "Bridge Evaluation" will be available.	Double-click
Optimization	Design: Choose a type of shape optimization for the design: Area, Inertia or Height of the section. Verification: Not required.	Double-click
Group of sections	Design: Choose the group of sections that can be used for the design. Verification: Not required.	Double-click
Shape	Design: Specify the steel shape that will be assigned to selected members. VisualDesign will search for appropriate shapes among the selected group of sections. Verification: Not required.	Double-click
Material	Design only: Select the steel material that will be assigned to optimized sections.	Double-click
HSS t (design)	If a HSS shape is specified as the design shape, you must select its design thickness in the list box: 1.0t CASC/Can), 0.9t (CISC/US) or 0.93t (AISC/US).	Double-click

Column	Description	Editing
Category	Cold-Formed Section only: Specify a category. This name must correspond to the one entered in the cold-formed section spreadsheet with respect to lower/upper-case. This column is not useful for other shapes.	Single click
Max. class	Design only: Specify the maximum class for optimized shapes.	Double-click
Max. height Mx	Design only: Specify the maximum height for optimized sections.	Single click
Min. height Mx	Design only: Specify the minimum height for optimized sections.	Single click
Max. width My	Design only: Specify the maximum width for optimized sections.	Single click
Min. width My	Design only: Specify the minimum width for optimized sections.	Single click
Maximum Capacity Factor	Design only: The design is done with respect to this given percentage of capacity. For example, if a value is inferior to 100%, it means that you are more severe than the chosen code.	Single click
Interaction	Design with S37-01 only: The "Compression-Tension" option is used to verify and design tower or antenna members according to compression and tension forces only (without bending). The "Standard" option (with bending) is automatically selected for all other users	Double-click
Analysis of Connections	Choose the type of analysis that will be done for connections: None, Verification or Design.	Double-click
Type of Connection	Type of bolted connection: Bearing or Slip-Resistant.	Double-click
Type of Hole	Specify if holes had been punched or drilled.	Double-click
Contact Surface	Specify the contact surface of connections: None, class A, class B or class C.	Double-click
Selection of Bolts	Choose the bolts that can be used for the design of bolted connections.	Double-click

Standard Steel Design

Steel Design

The "Cyclic Analysis and Design" icon of Tools toolbar

The **Analysis and Design** function launches an iterative process used in VisualDesign™ to calculate and optimize the selection of sections or to verify these sections according to the user specifications.

The **Analysis and Design** automatically uses a non-linear analysis. VisualDesign™ considers P-δ effects when calculating internal forces and displacements along members. These effects include amplified forces caused by the deflections, whatever type of transverse load is applied to the member (concentrated, triangular, trapezoidal, etc.). This unique method to VisualDesign™, allows ignoring U1 and U2 correction factors specified in the CAN/CSA-S16-01 standard and B1 and B2 factors used in the LRFD-1994 American Standard.

Deflection criteria are verified while executing the cyclic analysis and design process. To obtain deflection results, the user must specify a *Deflection* load combination status inside the **Load Combination** dialog box. The deflection is calculated using live loads only. VisualDesign™ automatically evaluates the span between inflexion points to verify allowable limits. Optimized members are based on inertia that satisfies a minimum stiffness allowing a control over deflections. Deflections on strong and weak axes are controlled the same way according to their respective deflection criterion.

DATE AND TIME

VisualDesign now displays the date of analysis and design in the **Design** 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.

Steel Design Procedure

To proceed with a cyclic design or a check of your structure, you must follow these steps:

- Complete the parameters in the **Analysis** tab in the **Project Configuration** dialog box. Two parameters are important to have a correct design, namely: The number of cycles for optimization and the number of cycles for correction.

- Activate design criteria in the **Member** tab of the **Member Characteristics** dialog box.

- Go to **Structure / Specifications** and define your own **Group of Sections** to be used during the design process. (A pre-defined one can also be selected in the **Steel Specifications** spreadsheet.)

- Select the **Steel Specifications** spreadsheet (**Structure** menu). Choose a Design specification, complete the required parameters, and modify the defaults values, if needed.

- If you wish to have identical sections for continuous members or wish to group members, select them and use the **Group Members** function (or use the short-cut keys [Ctrl] + G) in the **Structure/Group** menu.

- For all members that you wish to design, you must complete the information supplied in the **Steel Design** tab of the **Member Characteristics** dialog box. Select a steel specification OR a design group for the selected members.

- If you have to consider a seismic analysis in the design of your structure, complete the information in the **Seismic** tab of **Project Configuration** dialog box.

- If you have moving loads to consider, you must specify mobiles and moving load cases.

- Use the **Load Combination Generator** to generate required load combinations for a selected Code or Standard.

- Click the Analysis and Design icon and start the design process by clicking the "Analyse" button.

- To look at the results, consult the **Steel Design Results Spreadsheet** in the **Results/Structure Design** menu. With the **View Options**, you can display the sections number, capacity, and insufficient resistant members. To look at load combination results, activate the "Load Combination" mode and look at results through the **Results/Load Combinations** menu or through **View Options** (**Results** tab).

Optimization Problems

If you do not model your structure properly, design results will not be correct either. Here are problems that some clients encountered.

Problem 1: After a steel design, results shown that some members have a design load equals to 110% and 140%.

Answer: VisualDesign™ designed members according to your design criteria and the optimized shapes were chosen among the sections included in the selected group of sections, in the **Steel Specifications** spreadsheet.

Ex.: You specified a W shape for the design of columns in the **Steel Specification**s spreadsheet. Bigger shapes of this type are not included in the selected **Group of sections**.

Modify the Group of sections y including bigger shapes. If results are not good, choose a shape that is more resistant, such as WWF, in the **Steel Specification**s spreadsheet but make sure that these shapes are also included in the group of sections.

Problem 2: After having ran a steel design of a structure with bracings, as shown below, the design load of columns is equal to only 15%.

Answer: The modeling of bracings is not done the right way. You must attach bracings that are crossing the beams because the P-delta analysis will consider a global buckling of the columns using their full height.

Explanation: At the first cycle of analysis, VisualDesign™ will choose a small shape, such as W200x27, using a K value of 1.0 for local buckling, which is correct. For the second cycle of analysis, there will be a global buckling of the columns because the bracings do not act properly. The program will choose a shape that is strong enough to resist this global buckling (the value of K is now equal to 3). Results will be incorrect.
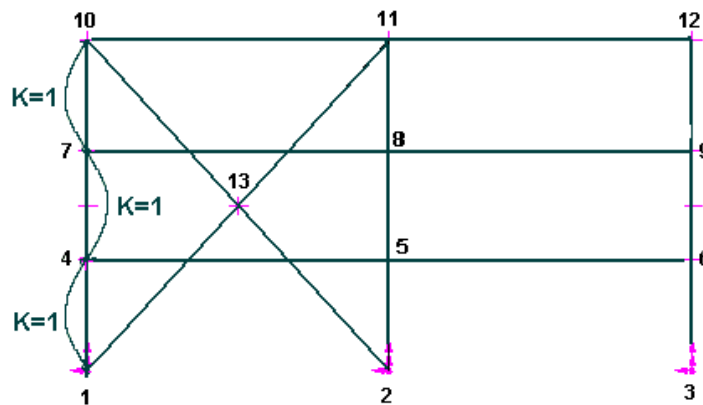
Figure 1. First cycle of analysis
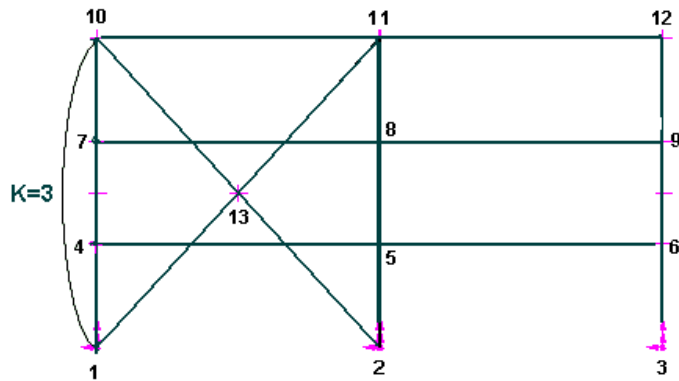
Figure 2. Second cycle of analysis

Modification of Bracings:

To correct the problem, create pinned nodes at the junction of horizontal beams and bracings and redo the design.

To create nodes at the junction of beams and bracings:

• Select the two members that are meeting (for a multiple selection, press the [Ctrl] key down will selecting several members) and click the icon ⊠ "Split with rigid connections".

• Do the same to create the three other nodes.

• Now, you have to change the new members end conditions to pinned-pinned. To do so, select all new members and press the "Properties" icon " 🏷 to call up the dialog box. In the Member tab, choose the "pinned-pinned" end conditions.

• Redo the same steps for other bracings, if any, and run a new steel design.
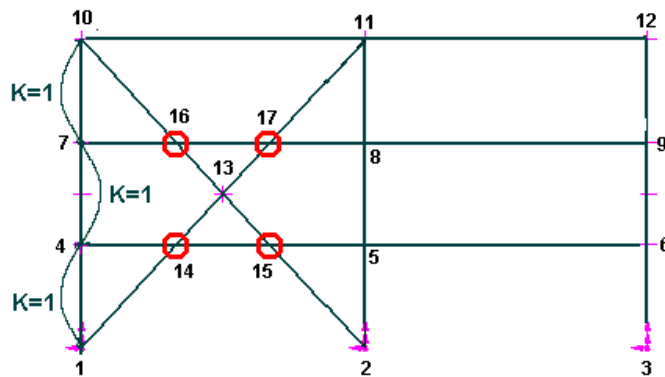
Figure 3. Modified model

View Options

Graphic Display

Coloured Display for Design Load

To look at design results in a graphic form, open the View Options dialog box and select the **Results** tab. Tick off the boxes corresponding to the display of members' design load.

Colours and corresponding numerical intervals are set in the **Limits** tab of **View Options** dialog box. Click the coloured square to choose a new colour. Enter the values for corresponding intervals.

Coloured Display of Design Groups

Select the **Colours** tab and choose a colour that represents the members' design groups. Then, select the **Attributes** tab, activate the display of design groups and select one among the list box.

Numerical Display

Internal Forces and Deflections

Select the **Results** tab of **View Options** dialog box to display results in a graphic or numerical form. You can display support reactions and moments at nodes, shear stresses, axial stresses, torsion and deflection of members by activating the "Numerical" option.

Design Brief

Display the member design brief by selecting the appropriate line in the members design results spreadsheet. Access this spreadsheet by selecting **Design Structure** under the **Results** menu. Then, highlight a spreadsheet line that corresponds to the chosen member and click the "Design Brief" icon. Use the Print Preview icon to have a look on the design brief before printing it. More, you can look at the internal stresses diagram from the same spreadsheet by clicking on the icon "Access to internal stresses diagrams" that is posted in the lower part of the spreadsheet.

See also
The Attributes tab
The Limits tab
The Results tab
The Colours tab
Displaying Members Results
Design Brief
Access to Internal Forces Diagram

Numerical Display of Members' Design Load

Select the **View** tab of **View Options** dialog box and activate the "Member" check box. Then, go to the **Results** tab and activate the "Design load" and "Numerical" options. Percentage of members' design load will be displayed on screen.

See also
Steel Design
The View tab
The Results tab

Steel Design Results

Steel Design Results Spreadsheet

The spreadsheet includes, among others, the name of shapes, critical load combination and clause that controlled the design, members design load, bending, shear and axial resistances, net area and members stability. A message also informs you if the design is OK or not. If not, lines will be coloured in red in the spreadsheet. However, the colour will not appear at the printing of the spreadsheet.

As soon as the design is finished, the "Design Results" mode is automatically activated. You can look at results in many ways:

- Go to **Results** menu and select the heading **Structure Design /Steel** or **Structure Design / ASD Steel**. If no members were selected, the spreadsheet will include all members;

- Double-click a member to access the spreadsheet. Results are supplied in the spreadsheet for this member only;

- Click on many members while pressing down the [Ctrl] key and click the **Properties** icon. Selected members will be part of the Steel Design Results spreadsheet.

Three buttons are posted in the lower part of this spreadsheet. They give access to internal forces in member and to a print of a design brief (if a member (line) is selected in the spreadsheet). A print preview is also supplied for the design brief. To learn more, see the following topics:

Design Brief
Access to internal forces and deflections
Print Preview of Design Brief

Group: Design Results

Column	Description	Editing
Number	Section number	No
Group	Group to which the member belongs.	No
Section	Name of shape.	No
Load Combination Mf + Nf	Load combination that controlled the combined bending and compression forces in the member.	No

Column	Description	Editing
Design load Mf + Nf	Member design load considering the combined bending and compression forces. A value greater than 100 means that section capacity is too small.	No
Load Combination Shear	Load combination that controlled the shear force in the member.	No
Design load Shear	Member Design load considering the maximum shear force. A value greater than 100 means that the shear capacity is insufficient.	No
Clause	Clause that controlled the design	No
F	Bridge Evaluation only: F is the calculated Live Load Capacity Factor for this member. It can be expressed as a percentage or factor.	No
Class Bending Mx	Class considered in the calculation of Mx.	No
Class Bending My	Class considered in the calculation of My.	No
Class Web	Class considered in the calculation of web buckling resistance	No
Class Compression	Class considered in the calculation of compression strength.	No
Type of Mfx	Indicates if a positive or negative moment (strong axis) controls the design for flexion.	No
Mrx Lu=0	Resisting moment (strong axis) when Lu = 0	No
Mrx Lu>0	Resisting moment (strong axis) when Lu > 0	No
Lux	Unsupported length (strong axis)	No
w2x	Factor allowing for the increase of the resisting moment of a laterally unsupported segment (strong axis) caused by a moment gradient. Please refer to topic Elastic Lateral-Torsional Buckling.	No
Type of Mfy	Indicates if a positive or negative moment (weak axis) controls the design for flexion	No
Mry Luy=0	Resisting moment (weak axis) when Lu = 0	No
Mry Luy>0	Resisting moment (weak axis) when Lu > 0	No

Column	Description	Editing
Luy	Unsupported length on weak axis. This value is null for sections with single and double symmetry because there is no lateral buckling on weak axis. VisualDesign supposes that all sections strong axis for bending is around local x-axis.	No
w2y	Factor allowing for the increase of the resisting moment of a laterally unsupported segment (weak axis) caused by a moment gradient. Please refer to topic Elastic Lateral-Torsional Buckling.	No
Type of Nz	Indicate tension or compression.	No
Tension Tr	Tensile strength of the section (Tr min using net area).	No
Cr Resistance	Compressive resistance of the section. (where $\lambda=0$).	No
Cr Stability	Compressive resistance according to KL/rx, KLy/ry, KL/rz.	No
KLx	Effective compression length on strong axis.	No
KLy	Effective compression length on weak axis.	No
KLz	Effective compression length according to orthogonal axis system. Refer to the Steel tab of Project Configuration	No
KLt	Effective compression length for torsional buckling. Refer to the Steel tab of Project Configuration	No
KL/rx	Slenderness ratio according to the strong axis	No
KL/ry	Slenderness ratio according to weak axis	No
KL/rz	Slenderness ratio according to orthogonal axis system.	No
KL/r max	Maximum slenderness ratio	No
T/C Ratio	Tension/Compression ratio corresponding to maximum design load for this member.	No
Net Area	Net Area of the section: gross area minus bolt holes and affected by the reduction coefficient.	No

Column	Description	Editing
Vrx	Shear resistance on weak axis.	No
Vry	Shear resistance on strong axis.	No
Trz	Resistance to torsion.	No
Message	Design result: Ok, Insufficient capacity, etc.	No
Deflection	Status of the deflection result: n/a, Ok < Allowable Deflection, Too much deflection.	No
Load Combination Mx	Load combination that governs the deflection criterion on strong axis.	No
Lx	Calculated span between inflexion points, on strong axis.	No
Deflection (Mx)	Deflection between two inflexion points, on strong axis.	No
Lx/	Ration between Lx/Deflection (Mx)	No
Load Combination My	Load combination that governs the deflection criterion on weak axis.	No
Ly	Calculated span between inflexion points, on weak axis.	No
Deflection (My)	Deflection between two inflexion points, on weak axis.	No
Ly/	Ratio between Ly/Deflection (My)	No
Stud (0:M+)	Number of required studs to transfer the shear stress. Number corresponding between the zero moment and the maximum positive moment	No
Stud (0:M-)	Number of required studs to transfer the shear stress. Number corresponding between the zero moment and the maximum negative moment	No

See also
Design Brief
Access to diagrams from the Design Results spreadsheet
Print Preview of Design Brief
Failure Modes - Steel Design Results (S37-01)

Member Internal Forces from the Design Results spreadsheet

The Internal Forces icon of Design Results spreadsheet

This icon, posted in the lower part of the **Steel Design Results** spreadsheet or the **Timber Design Results** spreadsheet, allows you to display member internal stresses and deformations spreadsheet, for a selected member. To do so, select a line in the spreadsheet and press the icon.

See also
Design Brief
Steel Design Result
Timber Design Results

Print Preview of Design Brief

The "Print Preview" icon for Design Brief

This icon is available in the Steel Design Results Spreadsheet and Timber Design Results Spreadsheet. It allows you to have a look at the **Design Brief** before printing.

To activate this function, select a line in the **Steel Design Results** spreadsheet or the **Timber Design Results** spreadsheet and press this icon. The Design Brief (for the selected member) will appear on your screen. If you wish to print it, select the "Print" command that is displayed. Notice that member results are for load combinations that controlled the design of this member.

See also
Design Brief
Steel Design Result
Timber Design Results spreadsheet
Title Bar

Design Brief

The Design Brief icon of Design Results spreadsheet

This icon, posted in the lower part of the **Steel Design Results** spreadsheet or the **Timber Design Results** spreadsheet, allows printing the design brief for a selected member. To do so, select a line in the spreadsheet and press the icon. Then, select a printer in the "Print" dialog box. Do not forget that results are according to critical load combinations (Combined bending/compression, Shear, and Deflection). Deflection is calculated according to inflexion points and shear energy is included.

The **Design Brief** is divided in three sections:

- The left part shows forces diagrams for critical load combinations, namely, **Bending-Compression** and **Shear** and finally, deflection diagrams for **Deflection** load combinations.

- The middle part shows concentrated and distributed load cases applied to the member, for the given load combination. Example: The loading diagram representing the dead load on the member and the loading diagrams representing the two-way floor triangular distribution of live load transferred on the member. Up to eighteen loading diagrams can be drawn.

- Finally, the third part includes the following verification:

  - Name of shape and building code used for the design;

  - Shape properties;

  - Material properties;

  - Critical load combination for combined bending/compression forces in the member, and forces that are considered in the design.

  - Critical load combination for maximum shear force acting on the member, and corresponding forces;

  - Verification of KL/r max, Mr with and without lateral buckling, verification of code provisions and deflections for *Deflection* load combinations, if any.

  - Conclusion.

N. B. The CAN/CSA-S6-88 and CAN/CSA-S6-00 standards are integrated in VisualDesign™. Design brief are available for these codes.

MESSAGES ABOUT KL/R:

If you get a message telling that the parameter KL/r is exceeding the allowable limit, it means that a load combination created this maximum value for KL/r. It may not correspond to the critical combined forces governing the design.

Internal Forces and Deflections for Design Groups

Activate an envelope, go to **Results / Envelope** and select this spreadsheet to look at maximum and minimum forces for a specific design group.

Group: Envelope Results: (title)

Column	Description	Editing
Number	Design group number (12 alphanumeric characters).	No
Shape	Chosen shape for this design group.	No
Value	Type of results: Max. or Min. An empty field indicates a standard analytical result.	No
Bending Mx	Min. or max. bending moment on strong axis for this design group.	No
Shear Vy	Min. or max. shear force on strong axis for this design group.	No
Bending My	Min. or max. bending moment on weak axis for this design group.	No
Shear Vx	Min. or max. shear force on weak axis for this design group.	No
Axial Nz	Min. or max. axial force for this design group (positive in tension).	No
Torsion Tz	Max or min. torsion force for this design group.	No
u (weak axis)	Max or min displacement on weak axis for this design group.	No
v (strong axis)	Max or min displacement on strong axis for this design group.	No
w (axial)	Max or min axial deformation for this design group (accurate solution only at member ends).	No

Bolted Connection Design Results

When the steel design is finished, go to **Results / Structure Design / Bolted Connections.**

The **Bolted Connections Design Results** includes information for each member end designed connections (node i and j). Look at the table below to know the definition of each column:

Group: Design Results (title)

Column	Description	Editing
ID	Automatically calculated.	No
Node Number	Structure node number	No
Shape	Member shape	No
Node	Node i or j. Results are given for each node.	No
Connection Number	Connection number at this node.	No
Connection Model Node i or j	Connection model at this node. See topic Connection Models	No
Type of Analysis	Type of analysis: Verification or Design.	No
Nos. of bolts	Number of bolts for this connection.	No
Bolts	Type of bolts used for this connection.	No
Nos. of transverse lines	Number of transverse lines for this connection. See topic Bolts Layout.	No
Nos. of longitudinal lines	Number of longitudinal lines for this connection. See topic Bolts Layout	No
Bolts Layout	Bolts layout: In line, Staggered A, or Staggered B. See topic Bolts Layout	No
Plate ep	Longitudinal distance measured from free edge of plate to the first bolt in a row.	No
Plate t	Designed plate thickness.	No
Other section	For connection model 5: specify the other section used as the connector.	No

Column	Description	Editing
Member em	Longitudinal distance measured from free edge of steel angle to the first bolt in a row.	No
Member p	Longitudinal distance measured centre-to-centre between two bolts.	No
Member g1	Transverse distance measured from outside edge of bent angle to the nearest bolt row.	No
Member g2	Transverse distance measured centre-to-centre between two consecutive bolt rows.	No
Member e	Transverse distance measured from free edge of member to nearest bolt hole.	No
Member Br	Bearing resistance of the connected member according to clause 6.5.2.2 i) and ii).	No
Member Tr1	Member tensile strength according to the failure mechanism for bolts layout or bolt bearing.	No
Member Tr2	Member tensile strength considering shear lag and member net area, as specified in the Steel Design tab of Member Characteristics dialog box.	No
Member Tf	Maximum factored tension in member.	No
Member Failure	Failure mechanism and failure path. See topic Failure Mechanisms in Bolted Connections	No
Plate Br	Bearing resistance of the plate in this connection according to clause 6.5.2.2 i) and ii).	No
Bolts Vr	Shear resistance of bolts in this connection.	No
Load Combination	Most critical load combination used for the design of this bolted connection.	No
Nf	Axial force in this bolted connection.	No
% Design load	Design load of the bolted connection.	No
Message	Message from VisualDesign about the design of this connection.	No
Inconsistent Dimensions	If some dimensions do not follow the code requirement, they will be written here.	No

Seismic Steel Design S16-01

Commentary on Seismic Design Requirements - Clause 27 CAN/CSA-S16-01

Excerpts from Standard CAN/CSA-S16-01, pp 2-104, 2-105.

[…] The objective of Clause 27 is to provide details that will exhibit ductility consistent with the R-values assumed in the analysis. The Clause applies to all structure in Canada for which energy dissipation capability is required.

The NBCC (1995) assigns force modification factors (i.e. load reduction factors), R, to various structural systems in relation to their capacity to dissipate energy by undergoing inelastic deformations. The greater the ability of the structure to dissipate energy, the higher is the assigned value of R, which is used as a divisor to reduce the magnitude of the seismic forces. R-values greater than 1.0 can only be justified if the structure has the ability to undergo inelastic deformations without loss of resistance.

This edition of the Standard also introduces an additional moment-resisting frame category that provides the more limited ductility associated with a force reduction factor of R=2.0, and that was judged appropriate for lower seismic zones. The force modification factors for the more ductile moment-resisting frame systems have been increased (to R=3.5 and 5.0) and the associated detailing requirements have become more demanding in the light of experience in recent earthquakes, and it was recognized that as a result these systems were less likely to be chosen in lower seismic zones. Although limited, the ductility of the new system provides for an improved design compared with the other alternative of an R=1.5 system.

Classes of Frames with R > 1.5

- Ductile moment-resisting frames (Type D, with R=5.0)

- Moderately ductile moment-resisting frames (Type MD, with R=3.5)

- Moment-resisting frames with limited ductility (Type LD, with R=2.0)

- Moderately ductile concentrically braced frames (Type MD, with R=3.0)

- Limited ductility concentrically braced frames (Type LD, with R=2.0)

- Eccentrically braced frames (R=4.0)

- Ductile plate walls (Type D, with R=5.0) (Not yet available in VisualDesign)

- Limited ductility plate walls (Type LD, with R=2.0) (Not yet available in VisualDesign)

Dissipating Elements according to Types of Frames

For each structural system, elements that dissipate energy are:

- The beams in moment-resisting frames;

- The bracings in concentrically braced frames;

- The links in eccentrically braced frames;

- The wall panels in steel plate shear walls.[…]

See also
Member Usages for Seismic Design
Moment-Resisting Frames
Concentrically Braced Frames
Eccentrically Braced Frames
Plate Walls

Member Usages for Seismic Design (S16-01)

Before running a seismic design as per CAN/CSA-S16-01, usages must be assigned to specific braced members (beams, columns, diagonals, etc.), which are considered as dissipating elements (energy) during an earthquake, according to the type of system they are part of (moment-resisting frames, concentrically braced frames, eccentrically braced frames, and plate walls) and the ductility of the system. Usages are assigned to members through the "Usage" list box, which is part of the **Member** tab (**Member Characteristics** dialog box).

Specific usages are given for each type of braced systems namely, Moment-Resisting Frames, Concentrically Braced Frames and Eccentrically Braced Frames.

Moment-Resisting Frames

Excerpts from Standard CAN/CSA-S16-01, pp 2-104, 2-105.

[…] Three categories of moment-resisting frames are recognized:

Ductile moment-resisting, or Type D, frames, in which members and connections are selected and braced to ensure that severe inelastic straining can take place.[…]

[…] Moment-resisting frames with moderate ductility, or Type MD, in which the member details are adequate to provide the more limited inelastic straining demanded in structure proportioned to resist the greater design loads, while at the same time, connections are adequate to accommodate the associated forces and deformations. […]

[…] Type LD for limited ductility, newly introduce. This system undergoes still less inelastic demand consistent with the higher design loads, and can in general make use of traditional connection detailing, combined with special requirements associated with welding. […]

Usages:

Before executing a seismic design as per CAN/CSA-S16-01 Standard, member usages must be defined according to the type of system and dissipating elements. Usages are assigned through the "Usage" list box, which is part of the **Member** tab (**Member Characteristics** dialog box).
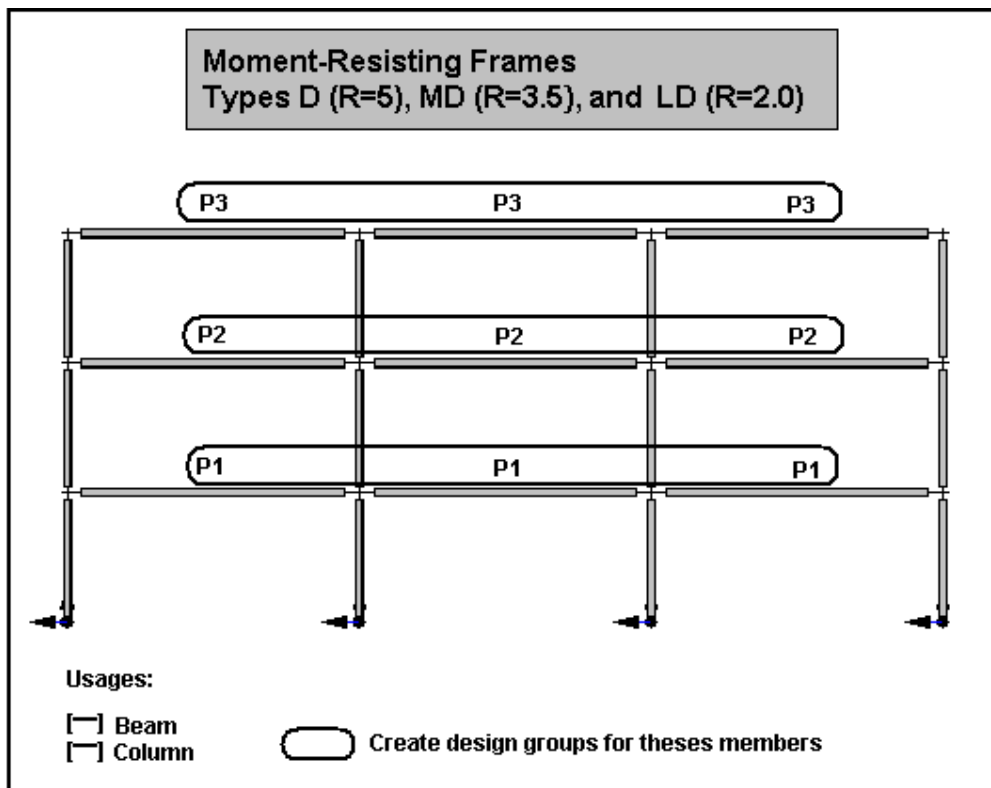
See also

Concentrically Braced Frames

Excerpts from Standard CAN/CSA-S16-01, pp 2-104, 2-105.

[…] Concentrically braced frames are those in which the centre-lines of diagonal braces, beams and columns are approximately concurrent with little or no joint eccentricity. […]

[…] Inelastic straining must take place in bracing members subjected principally to axial load. Compression members dissipate energy by inelastic bending after buckling, and in subsequent straightening after load reversal. Cyclic local buckling can lead to early fracture, and consequently width-to-thickness limits are restricted for braces. These frames usually have limited redundancy, and are prone to weak and soft storey response in which energy dissipation is localized. In this edition of the Standard, emphasis in these categories is placed on the presence of redundancy in the system to provide strength and stiffness in the event that the braces in a storey lose much of their stiffness. […]

[…] Two categories of concentrically braced frames are considered, those with moderate ductility (MD) and limited ductility (LD). Both permit several different bracing configurations, although some are proscribed. Compared with past editions of the Standard, the provisions maintain strict limits on width-thickness ratios; overall slenderness limits of braces are relaxed, and changes have been made to the requirements for connection design forces. However, height limitations are introduced. […]

Usages:

Before executing a seismic design as per CAN/CSA-S16-01 Standard, member usages must be defined according to the type of system and dissipating elements. Usages are assigned through the Usage list box, which is part of the **Member** tab (**Member Characteristics** dialog box).

Concentrically Braced Frames - Types MD (R=3.0) and LD (R=2.0)

Tension/compression
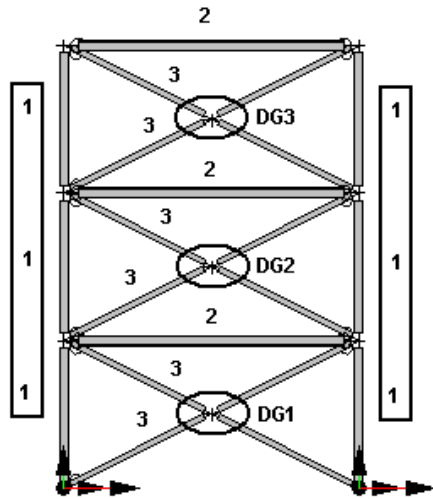
Tension only

Usages:

1: [X] Column
2: [X] Beam
3: [X t/c] Diagonal

End conditions for diagonals in
tension/compression : <-> [ ] <->
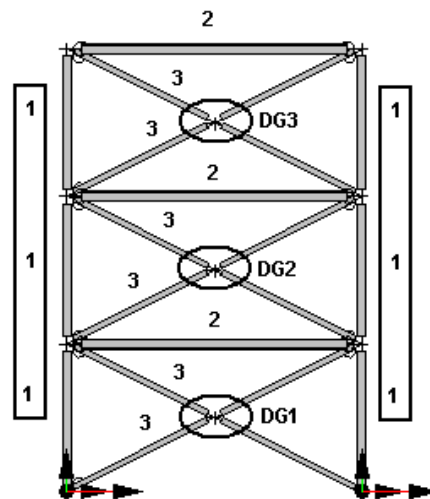
Usages:

1: [X] Column
2: [X] Beam
3: [X t] Diagonal

End conditions for diagonals in
tension only: <- [ ] ->

DG: Design groups are required

Continuous columns (4 levels) = Design group
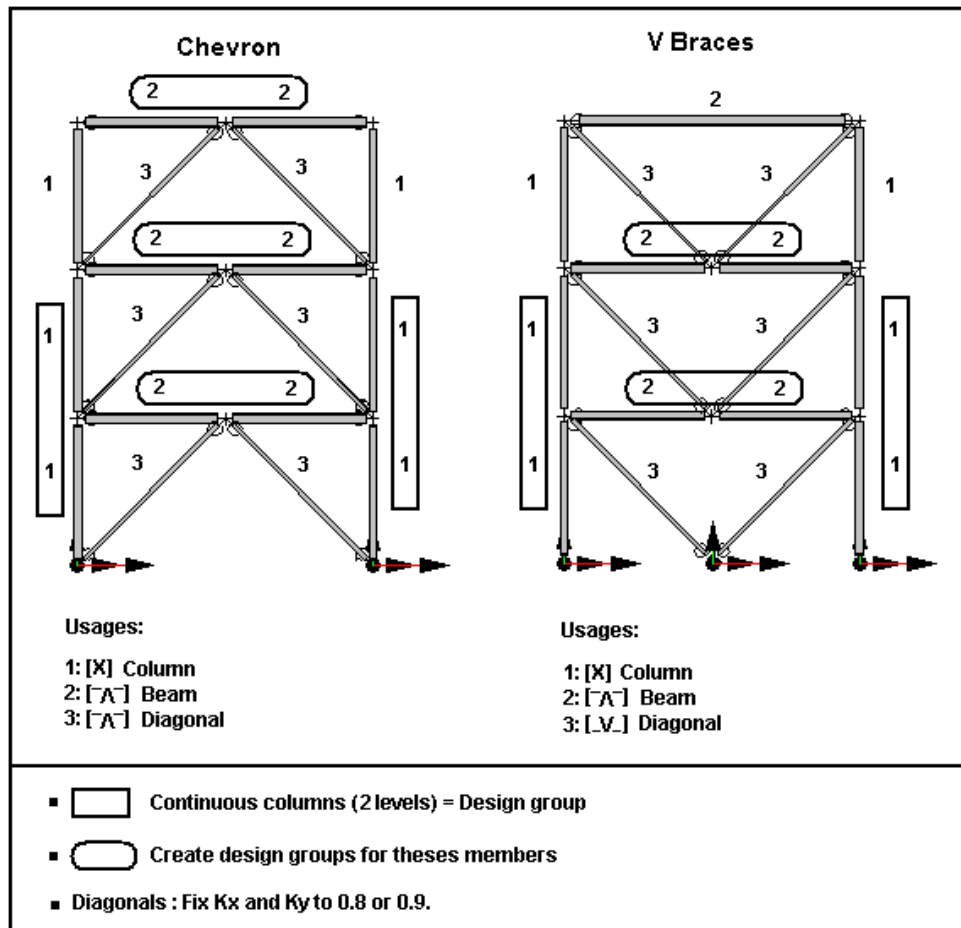
See also

Commentary on Seismic Design Requirements-Clause 27-CAN/CSA-S16-01
Member Usages for Seismic Design
Moment-Resisting Frames
Eccentrically Braced Frames
Plate Walls
Seismic Design Results

Eccentrically Braced Frames

Excerpts from Standard CAN/CSA-S16-01, pp 2-104, 2-105.

[…] Eccentrically braced frames are those in which diagonal braces, at least at one end, do not intersect the beam and column intersection points or, in the case of chevron bracings, the two braces do not intersect on the beam axis. These configurations produce a link element that for ductile systems is designed as the dissipating element. The Standard gives provisions for frames with links in the beams. […]

Usages:

Before executing a seismic design as per CAN/CSA-S16-01 Standard, member usages must be defined according to the type of system and dissipating elements. Usages are assigned through the "Usage" list box, which is part of the **Member** tab (**Member Characteristics** dialog box).

Eccentrically Braced Frames - Type D (R=4.0)

Usages:

1 : [⌒] Column above
2 : [⌒] Column - others
3 : [⌒] Diagonal
4 : [⌒] End beam
5 : [⌒] Link beam

Notes:

- N. B. Add lateral supports to the bottom of link beams, at node i and j
- ⬭ Create design groups for theses members
- Diagonals : Fix Kx and Ky to 0.8 or 0.9.
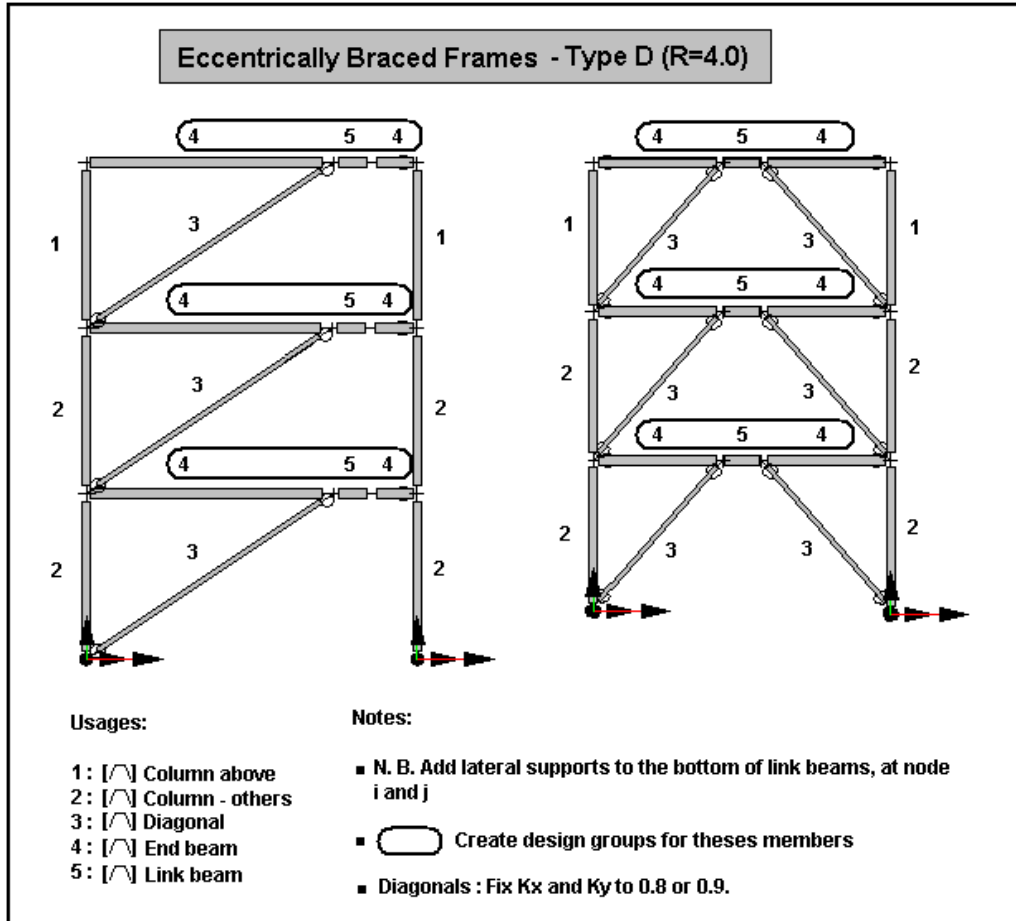
See also
Commentary on Seismic Design Requirements-Clause 27-CAN/CSA-S16-01
Member Usages for Seismic Design
Moment-Resisting Frames
Concentrically Braced Frames
Plate Walls
Seismic Design Results

Plate Walls (not yet available)

Excerpts from Standard CAN/CSA-S16-01, pp 2-104, 2-105.

[…] Plate walls are formed by thin wall plates framed by beams and columns. These highly redundant and stiff systems dissipate energy by yielding of the plate and/or the framing members, and the good seismic performance anticipated is reflected in the high values of the force modification factors applicable. Two categories are defined, Types D and LD […]

Usages:

Before executing a seismic design as per CAN/CSA-S16-01 Standard, member usages must be defined according to the type of system and dissipating elements. Usages are assigned through the Usage list box, which is part of the **Member** tab (**Member Characteristics** dialog box).
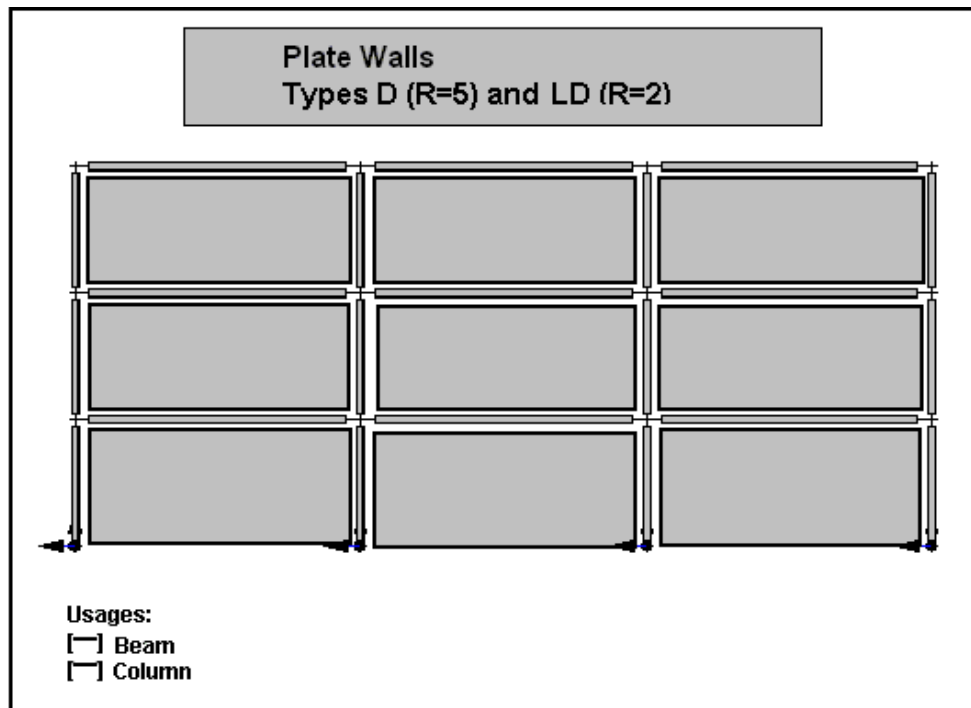
See also
Commentary on Seismic Design Requirements-Clause 27-CAN/CSA-S16-01
Member Usages for Seismic Design
Moment-Resisting Frames
Concentrically Braced Frames
Eccentrically Braced Frames
Seismic Design Results

Procedures - Seismic Steel Design - S16-01

Follow the next steps to model and run a seismic design of a building according to Clause 27 of CAN/CSA-S16-01:

Project Configuration

- Go to **File / Project Configuration** and select the **Analysis** tab. Check parameters for the non-linear analysis.

- Go to the **Seismic** tab and select a building code. Then, fill in the required seismic parameters. Activate the option "Add ductility effects [/¯\]" to design members according to Clause 27 of S16-01 Standard.

Modeling

- According to the type of braced system, continuous columns must be modeled on each side of the system, over two or four stories, as per building code. Continuous columns must be defined further on as part of a design group.

- For tension-only X-bracings (Concentrically Braced Frame – X), select tension-only axial end conditions [<- [ ]->] in the **Member** tab of **Member Characteristics** dialog box.

- For tension/compression X-bracings (Concentrically Braced Frame – X), axial end conditions must be [<->[ ]<->].

Member Usages

- Adequate member usages must be assign to columns, braces, and beams that are part of a braced system that will absorb and dissipate energy. Usages are defined in the **Member** tab of **Member Characteristics** dialog box.

Steel Specifications

- Define a group of sections to be used for the design.

- Open the **Steel Specifications** spreadsheet. Add specifications if different steel shapes are needed for the design. Select the group of sections.

- Class of shapes: VisualDesign will automatically select the right class according to the type of braced system and member usages, even if a class 3 or 4 is specified in the steel specifications.

Design Groups

- Create design groups for beams and continuous columns, depending on the type of braced system.

Design Criteria

- Activate design criteria in the **Member** tab of **Member Characteristics** dialog box.

- Go to the **Steel Design** tab and choose a steel specification or a design group for each member that will be designed. Fill in other design criteria:

  - For diagonals, Kx and Ky can be fixed to 0.8 or 0.9.

  - Add lateral supports at the bottom of link beams, at node i and j, if applicable.

Load Combinations

- Use the **Load Combination Generator** and include spectral envelopes E01 and E02 in the generation.

Linear Seismic Directions

- Open this spreadsheet (**Loads / Seismic Directions / Linear**) and insert two lines, as two main and orthogonal directions are required. Give a short name to each direction, such as Dir. x and Dir. z. Close the spreadsheet.

Modal Analysis

- Launch the modal analysis.

- Display the deflection of each vibration mode by using the function **Animation** in order to detect local vibration modes and eliminate them. Increase the amplitude of diagram. We recommend modifying the model to eliminate local vibration modes. The model is adequate when the two main vibration modes are present within the tenth first calculated modes (not always possible, though).

- Consult the frequencies and vibration modes spreadsheet (**Results / Modal/Spectral / Frequencies**). Find the maximum modal contributions for each seismic direction (columns γ). Note the corresponding components Dir. x, Dir. y, and Dir. z and copy them in the **Linear Seismic Directions** spreadsheet, for each main direction. N. B. These components can be different from 1.0.

Ductility and Spectral Envelopes

- According to the building code that was selected in the Seismic tab, the following parameters must be selected in the Linear Seismic Directions spreadsheet: For CNB-95, complete the columns "Structure", "Ds", and force modification factor "R". For CNB-2005, complete columns "Structure", "Lateral Force Resisting System", "Ro", and "Rd". Force modification factors can be different for each seismic direction, so select a spectral envelope for each one.

- Launch the steel design by clicking the icon ![icon]. The iterative process will start by the modal and spectral analyses and then, the steel shapes will be modified. During this process, the software automatically considers maximum modal contributions. Main vibration modes are modified in the Linear Seismic Directions spreadsheet, if they changed.

Results

- Go to **Results / Spectral/Modal** and open the **Levels** spreadsheet. Check interstory drifts.

- Activate the **View Options** and display the member design loads.

- Consult steel design results (**Results /Structure Design /Steel)**. .

- Select the **Seismic Design Results** spreadsheet (Results /Structure Design /Steel - seismic Design).

Convergence

- If the convergence is not easy to reach, increase the number of correction cycle to 5-7. (**Analysis** tab – **Project Configuration**).

See also
Commentary on Seismic Design Requirements-Clause 27-CAN/CSA-S16-01
Member Usages for Seismic Design
Steel Design Results
Design Brief
Seismic Design Results

Seismic Design Results

Steel Design Results - Seismic Design

CAN/CSA-S16-01

This spreadsheet includes calculated parameters and additional seismic results based on clause 27 of standard S16-01 for seismic design, for a chosen load combination. Therefore, load combinations that include a seismic envelope can be consulted.

Following a steel design with standard CAN/CSA-S16-01 with seismic loads, activate a load combination and go to **Results / Structure Design / Steel – Seismic Design**.

AISC-LRFD-95

Results are also available for American Standard AISC-LRFD-95.

Group: Load Combination Results

Column	Description	Editing
Number	Member number	No
Section	Steel shape assigned to this member.	No
Group	Group in which belongs this member.	No
Usage	Member usage	No
Ag	Gross area of this member.	No
Aw	Web area of this section.	No
Aw / Ag	Web area divided by the gross area.	No
Zx	Plastic section moment on strong axis	No
Ry	Factor applied to Fy for evaluating the probable yield strength of this member	No
Fy	Yield strength of steel	No
Class Bending Mx	Calculated class for bending on strong axis, Mx.	No
Class Bending My	Calculated class for bending on weak axis, My.	No
Class Web	Class considered in the calculation of web buckling resistance.	No

Column	Description	Editing
Class Compression	Class considered in the calculation of compression strength.	No
e	Length of the link beam.	No
Pf	Factored axial force in the link beam, for current load combination.	No
Vf	Factored shear force in the link beam, for current load combination.	No
Vp	0.55wdFy.	No
V'p	Value V'p based on clause 27.7.2 "Link beam Resistance", for current load combination.	No
Mp	Resisting plastic moment = Z Fy.	No
M'p	Value M'p based on clause 27.7.2 "Link beam Resistance", for current load combination.	No
ΦV'p 2ΦM'p/e	Shear resistance of the link beam. This resistance represents the lesser value of ΦV'p and 2ΦM'p.	No
1.1 RyMp	Probable moment for developing a plastic hinge.	No
AgRyFy	Probable tension to attain plasticity of the section.	No
1.2 Cpr	Probable compression, calculated with 1.1 fy.	No
0.2 AgRyFy	This value must be used along with 1.2 Cpr.	No
γ	Link rotation, according to clause 27.7.4.	No
γ max	Maximum link rotation, according to clause 27.7.4.	No
e min	Minimum length of the link beam.	No
e max	Maximum length of the link beam.	No
1.15 Ry Vn	Probable shear force to attain plasticity of the section.	No
1.30 Ry Vn	Probable shear force to attain plasticity of the section	No

TOWER DESIGN

TABLE OF CONTENTS

Chapter 11 Tower Design

General

The Tower Design Module

This module verifies and designs towers, such as guyed and self-supporting structure, according to CAN/CSA S37-01 and ACSE-10-97 Standards. Specific equations are available for the calculation of effective compression lengths. They depend on the shape of bracings and number of bolts at end connections. The module also considers members' behaviour in compression-only, tension-only or with bending, as the user specified in the **Steel Specification** spreadsheet. It also designs and verifies member bolted connections. Many bolted connections are already defined.

Tools are available in the **Loads** menu, to automatically generate ice and wind loads on open structure such as towers. These tools are useful because ice loads and wind loads are automatically recalculated as shapes are modified during the design process.

See also
Steel Design tab
Limitations of the Steel Design module
Project Configuration
Steel Specifications spreadsheet
Automatic Generation of Ice loads
Automatic Generation of Wind loads
Bolts spreadsheet
Bolted Connections spreadsheet
Member Bolted Connections
Selection of a KL/r Formula for Braced members
Major/Minor and Orthogonal Axis Systems
Automatic Calculation of Kx, Ky, Kt or Kz
Analysing a Guyed Tower
Results of Bolted Connection Design
Failure Modes - Steel Design Results
Tower Results

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.

This table describes the fields in this dialog box:

Field	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.

Field	Description
Tributary Area	You must specify if the reduction factor due to tributary area should apply to compression or bending.
	With the drop-down list box, select the code that will be use for the reduction of tributary surface
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 will terminate when the variation falls below this value.
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 optimization cycles for the design of members.
Number of cycles (corrections)	When the optimized number of cycles is reached, the members that have not been optimized are evaluated in 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.

The Steel tab

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".

Field	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).

Field	Definition
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).
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

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

Member Characteristics

Major/minor and Orthogonal Axis System

If you chose single steel angle 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.

At 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.
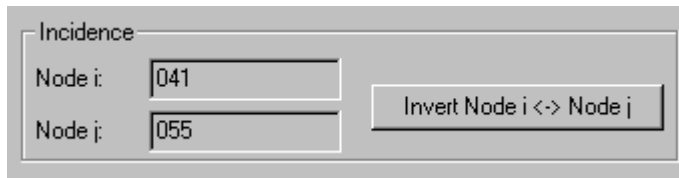
If you cannot orient the steel angle properly, use the "Invert Node i ↔ Node j" button shown above.

Orthogonal Axis System and 180 deg. Beta Angle
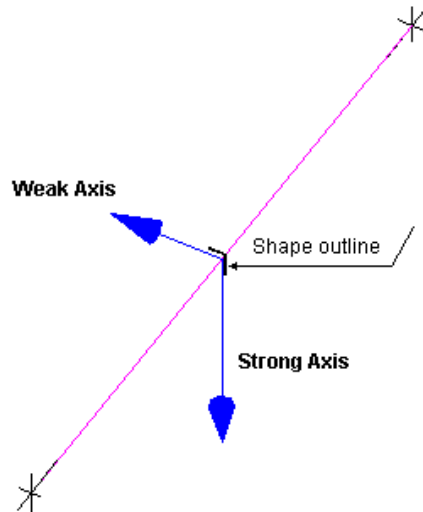
For a Steel Design:

When steel angles are correctly positioned, select them and switch to a major/minor axis system. You will notice that the beta angle is automatically readjusted.

Major and minor axis will be displayed as follows:

Major/minor axis system
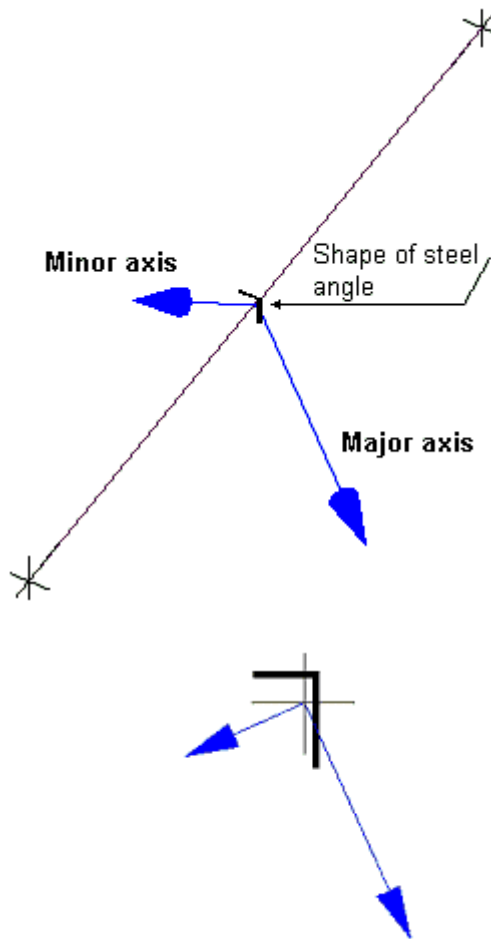
Note The orientation of beta angle can vary during cycles of design (when steel angle changes) in order to maintain the orientation of orthogonal axes.

See also
Convention – Forces in Member
Steel Design Criteria

Member Usage for Tower Design

The "Usage" list box has been initially created for the Tower Design module. Items *Upright*, *Guy*, *Horizontal*, *Diagonal*, *Horizontal Secondary*, *Diagonal Secondary*, *Vertical Secondary*, *Stabilizer,* and *Internal Bracing* are specifically used to define tower components.

Tower member usage is specified in the **Member** tab of **Member Characteristics** dialog box. Usage is also required when defining wind and ice loads. Option *All* has been included in the **Usage** list box to apply wind and ice loads on all members.

You can select member with a specific usage with function **Select**/**Usage** in **Edit** menu. Choose a member usage in the dialog box.

See also
Select Members according to Usage
Guy Elements

Select Members According to Usage

The type of member usage is specified in the **Member** tab (**Member Characteristics** dialog box). Members having a common usage can be quickly selected using the function **Select /Usage** in **Edit** menu.

In the **Selection of Member Usage** dialog box, choose a member usage in the drop-down list box. Click OK.
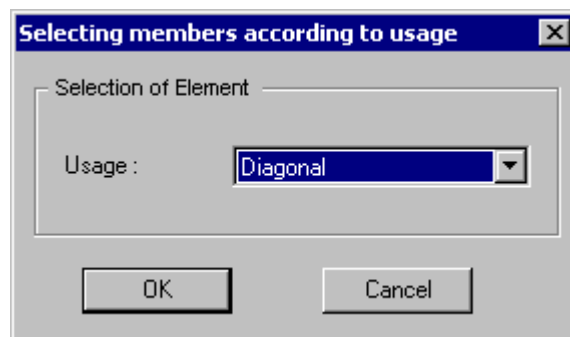
Members will be highlighted on screen.

The Steel Design tab

If you checked the "Activate Design Criteria" box in **Member Characteristics** dialog box, complete parameters for the design or verification of steel members.

Look at this table to learn more about parameters included in the **Steel Design** tab.

Field	Description
Design Parameters	
Design or Verification	This shaded field indicates the type of analysis that is going to be carried on, as specified in the steel specification spreadsheet.
Specification	Select a steel specification among the drop-down list box. VisualDesign will design this member according to it. Press the […] button next to this field to add a new specification.
Design Group	Select the design group that applies to this member, if required. (A steel specification is chosen within a member design group.). Press the […] button next to this field to add a new design group.

Field	Description
Lateral Supports to avoid buckling	
Top and Bottom of section	Specify lateral supports, at the top and bottom of section. Tick off the appropriate boxes: Node i, Node j and/or Continuous lateral support. Refer to topic Lateral Support
Position of load	By default, the load is applied at the centre of section. If this is not the case, choose among options: *Top* or *Bottom*.
Cantilever	
Automatic	Automatic calculation of factor Kux. VisualDesign automatically calculates Kux factor in particular cases.
	Warning, if you disable the "Automatic" option, you must enter a value for Kux. If you don't, VisualDesign will consider a factor of 10 as compression length and this member will be a cantilever even if you have not specified it. See topic Cantilever
Kux	Specific coefficient used for the calculation of cantilever effective compression lengths. It depends upon the position of lateral supports on the member and the position of load. The default value is 2.5.
At node	If Kux is automatic, indicate the location of the free end of cantilever: Node i or j.
Effective Compressive Length	See topic Automatic Calculation of Kx, Ky, Kt and Kz
Kx Factor	Effective compressive length factor relative to strong axis. If "Auto" is checked, Kx coefficient is automatically calculated.
Ky Factor	Effective compressive length factor relative to weak axis. If "Auto" is checked, Ky coefficient is automatically calculated.
Kt Factor	Effective compressive length factor used for the computation of buckling due to torsion. If "Auto" is checked, Kt is automatically calculated according to the **Steel** tab of **Project Configuration**.
Kz Factor	Effective compressive length factor relative to orthogonal axis system. Kz is used for single steel angles and an orthogonal axis system must have been selected in the member tab. If "Auto" is checked, Kz is automatically calculated according to the **Steel** tab of **Project Configuration**.
Effective Net Area, with or without reduction factor	
Total D holes	If bolts are used as connections, enter the total width of holes for this member.

Field	Description
A'ne = Ane x (Rf)	Reduction factor used for the calculation of net area. Ex.: A'ne = Ane x 0.80
Max. Slenderness	Enter the member maximum slenderness if different from default value, 200.
Stiffeners/Intermittent Fillers (2L)	
Spacing	Enter the spacing between I beam web stiffeners for shear capacity calculation <u>OR</u> enter the spacing between intermittent fillers for double steel angles. A value of 0 means that there is no stiffener or filler.
Ft = 0	If Ft = 0, the tension field component of post-buckling stress will not be considered in the calculation of shear resistance of the beam. Refer to clause 10.10.5 – CAN/CSA-S6-00 Standard. N.B. If Ft=0, the bending moment and shear interaction equation will not be verified.
Factor km	This factor is used in the calculation of effective compression length of double steel angles with intermittent fillers. Refer to clause 6.2.4.3 CAN/CSA-S37-01 Standard..
Allowable Deflection	
Lx/	Enter a deflection criterion relative to strong axis.
Ly/	Enter a deflection criterion relative to weak axis.
HSS or Solid round shape	Tick off this box if you want VisualDesign to account for stress relaxation in the calculation of axial compression according to clause 13.3.1 (S16-01 Standard)

The Bolted Connections Tab

Complete this tab in order to specify the right effective length for bracing members. In addition, you will find information about the verification or design of the member bolted connections.

Once in the **Member Dialog Box**, select this tab and complete the following parameters.
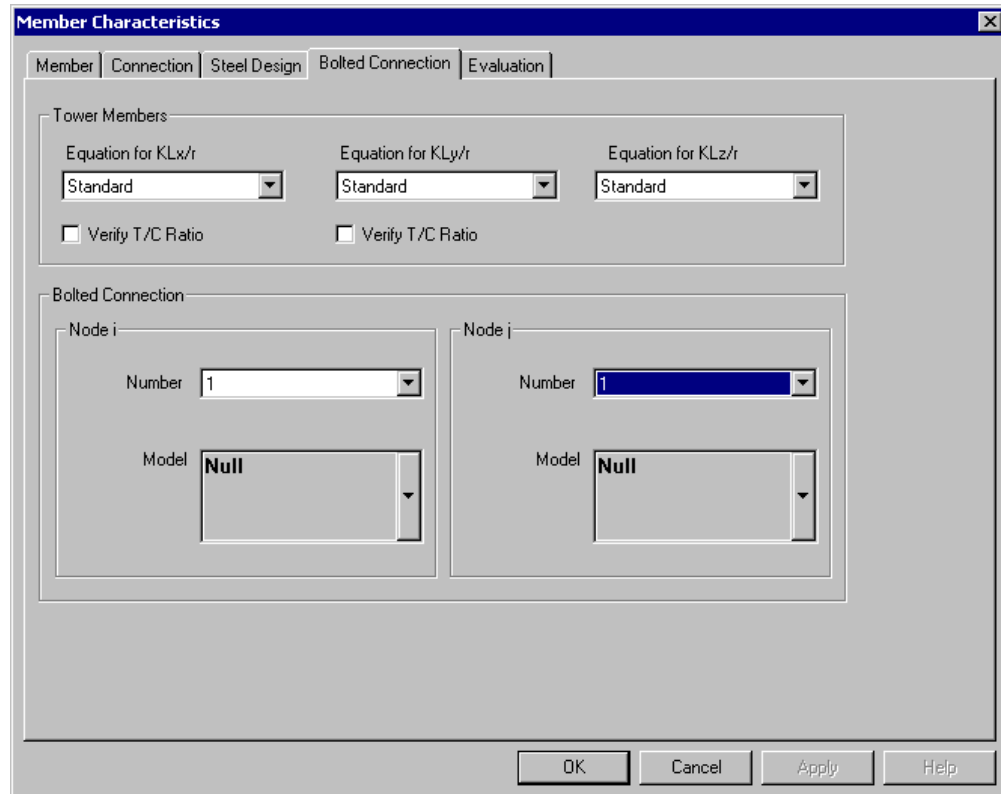
Look at the table below for description of the topics included in the tab.

Field	Description
Bolted Connections – Node i and Node j	
Number	Connection number at node i or node j.
Model	Choose node i and j connection model if you want VisualDesign to design it for you.

CivilDesign inc. Engineering Software

Field	Description
Tower Members	
Equation for KLx/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in x which takes into account the type of bracing member and the number of bolts in the member end connections.
Verify T/C ratio	Activate this option if you want VisualDesign to compute the effective compressive length KLx according to clause 6.2.3.3 (S37-01Standard). Refer to Ratio T/C
Equation for KLy/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in y which takes into account the type of bracing member and the number of bolts in the member end connections.
Verify T/C ratio	Activate this option if you want VisualDesign to compute the effective compressive length KLy according to clause 6.2.3.3 (S37-01Standard). Refer to Ratio T/C
Equation KLz/r	According to S37-01 standard (towers): Select the equation that will be used for the calculation of effective length in the minor axis system.
	If single steel angles are not oriented in an orthogonal axis system, the drop-down list box will be shaded.

Ratio T/C

The verification of tension/compression ratio in a member (bolted), on strong or weak axis, can be specified in the **Bolted Connections** tab of **Member Characteristics** dialog box.

However, the following conditions must apply to members:

- The option (check box) must be activated;

- Member axis must be perpendicular to panel with a tolerance of 10 degrees.

If some members do not respect these conditions, it is OK but the analysis will take more time.

Selection of a KL/r Formula for Braced Members

Formulas that are used for the calculation of KL/r can be associated to a type of member (angle, HSS or rod) and/or to eccentricity and end restraint conditions at the member ends connections.

Formulas 1 to 6 are described below:

For L/r ≤ 120:

Formula 1:   Concentric load at both ends

$$KL/r = L/r$$

Formula 2:   Concentric load at one end and normal framing eccentricity at the other end

$$KL/r = 30 + 0.75 \, L/r$$

Formula 3:   Normal framing eccentricity at both ends

$$KL/r = 60 + 0.50 \, L/r$$

For L/r > 120:

Formula 4:   Unrestrained against rotation at both ends

$$KL/r = L/r$$

Formula 5:   Partially restrained against rotation at one end

$$KL/r = 28.6 + 0.762 \, L/r$$

Formula 6:   Partially restrained against rotation at both ends

$$KL/r = 46.2 + 0.615 \, L/r$$

The following tables are based on S37-01 Standard and supply formulas used for the calculation of KL/r (< 80, <120 or >120) for different types of members.

Table 1. KL/r Formulas for Angle and Tubular Bracing Members:

(Refer to Clause 6.2.5.1 and Table 5 of S37-01 standard)

Connections	L/r ≤ 120	L/r > 120
Standard	KL/r = L/r	KL/r = L/r
CNR---CNR	Formula 1	Formula 4
CNR---ENR	Formula 2	Formula 4
CNR---EPR	Formula 2	Formula 5
CPR---CNR	Formula 1	Formula 5
CPR---CPR	Formula 1	Formula 6
CPR---ENR	Formula 2	Formula 5
CPR---EPR	Formula 2	Formula 6
ENR---ENR	Formula 3	Formula 4
EPR---ENR	Formula 3	Formula 5
EPR---EPR	Formula 3	Formula 6

N.B. If the "Standard" option is selected, VisualDesign will make no adjustment on KL/r value. The program will consider a standard KL/r.

Description of Member End conditions:

Three letters are used to describe member end conditions:

C: Concentric load;

E: Normal framing eccentricity;

NR: Unrestrained against rotation (hinge) (*No Restriction*);

PR: Partially restrained against rotation (*Partially Restrained)*

Example: CPR---ENR code means:

The first end connection is concentric and partially restrained against rotation and the other end connection is unrestrained against rotation with a normal framing eccentricity.

Table 2. KL/r Formulas for Solid Round Welded Bracing Members:

(Refer to Clause 6.2.5.3 and Table 7 of S37-01 standard)

Bracing	Definition	L/r ≤ 80	80 < L/r ≤ 120	L/r > 120
Straight Rod	Single diagonal	0.90DL/r	[0.70+0.005(120-DL/r)]DL/r	0.70DL/r
Bent Rod	Single diagonal composed of a continuous bent rod.	1.00DL/r	[0.80+0.005(120-DL/r)]DL/r	0.80DL/r
Xa Rod	X Bracing: One continuous, one split diagonal, all in the same plane, welded all around at intersection.	0.95DL/r	[0.75+0.005(120-DL1/r)]DL1/r	0.75DL/r
Xb Rod	X Bracing: Both diagonal continuous, one bent out of plane, tack welded at intersection.	1.05DL/r	[0.85+0.005(120-DL1/r)]DL1/r	0.85DL/r
Xc Rod	X Bracing: Both diagonal continuous, both bent out of plane, tack welded at intersection.	1.10DL/r	[0.90+0.005(120-DL1/r)]DL1/r	0.90DL/r

Where:

D: Rod outside diameter

L1: Refer to Figure 7- K Bracing and Clause 6.2.3.5.1 of Standard S37-01.

Table 3. KL/r Formula for Single Tubular Bracing Members:

KL/r = 0.85DL/r

Where D is the outside diameter of HSS

See also

Steel Design tab
Automatic Calculation of Kx, Ky, Kt or Kz
Steel Specifications spreadsheet
Automatic Calculation of Ice Loads
Automatic Calculation of Wind Loads
Bolts Spreadsheet

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

Maximum Slenderness Ratio for Tower Member

VisualDesign designs tower members according to the following slenderness ratios per CAN/CSA-S37-01 Standard:

Member Usage	KL/r max
Upright	120
Guy	200
Horizontal	200
Diagonal	200
Stabilizer	200
Internal Bracing	200
All Secondary members	240

See also

Steel Design Results

Members Spreadsheets

Member Steel Design Spreadsheet

This spreadsheet will be available if you own the **Steel Design** module and if you activated design criteria in the **Member Characteristics** Dialog box. Complete the information before running a Steel Design.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Member number.	Single click
Specification	Choose a steel specification in the drop-down list box. Specifications are defined in the Steel Specifications Spreadsheet	Double-click
Group	Choose a member design group, if desired. (Specification is included in the Groups spreadsheet.)	Double-click
Kx computation	Choose an automatic or manual computation of effective compression length factor, relative to strong axis.	Double-click
Kx	Effective compression length factor relative to strong axis. For an "Auto" computation, the Kx coefficient is automatically calculated.	Single click
Ky computation	Choose an automatic or manual computation of effective compression length factor, relative to weak axis.	Double-click
Ky	Effective compression length factor relative to weak axis. For an "Auto" computation, the Ky coefficient is automatically calculated.	Single click
Kz computation	Choose an automatic or manual computation of Kz. If the Automatic box is activated, VisualDesign will consider the specified value in the **Steel** tab of **Project Configuration**. See also topic Automatic Calculation of Kx, Ky, Kt and Kz.	Double-click
Kz	Effective compression length factor in the orthogonal z-axis.	Single click

---

CivilDesign inc. Engineering Software **11-19**

Column	Description	Editing
Kt computation	Choose an automatic or manual computation of Kt. If the Automatic box is activated, the Kt factor is automatically calculated according to the **Steel** tab of **Project Configuration**. See also topic Automatic Calculation of Kx, Ky, Kt and Kz.	Double-click
Kt	Effective compression length factor for the calculation of buckling due to torsion.	Single click
Lux Bottom Noi	If the shape bottom flange is laterally supported at node i, choose option [x].	Double-click or Space bar
Lux Bottom Cont	If the shape has continuous lateral support at the bottom flange, choose option [x].	Double-click or Space bar
Lux Bottom Noj	If the shape bottom flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Lux Top Noi	If the shape top flange is laterally supported at node i, choose option [x].	Double-click or Space bar
Lux Top Cont	If the shape has continuous lateral support at the top flange, choose option [x].	Double-click or Space bar
Lux Top Noj	If the shape top flange is laterally supported at node j, choose option [x].	Double-click or Space bar
Hole Width	Width of the hole if bolts are used.	Single click
Reduction of Ane	Reduction factor used for the calculation of net area. Ex: A'ne = Ane x 0.80	Single click
a: Stiffener or a: Intermittent fillers (2L)	I Beam: Enter the spacing between web stiffeners for shear capacity calculation. OR Double steel angles: Enter the spacing between intermittent fillers along the member.	Single click
Ft = 0	If you do not want to consider the tension field component of post-buckling stress in the calculation of shear resistance of the beam, choose option [x]. Refer to clause 10.10.5 – CAN/CSA-S6-00 Standard. N.B. If Ft=0, moment and shear interaction equation will not be verified.	Double-click or Space bar.

Column	Description	Editing
km : intermittent fillers	Factor used in the calculation of effective compression length of double steel angles with intermittent fillers. Refer to clause 6.2.4.3 CAN/CSA-S37-01 Standard.	Single click
Stress-Relieved	For HSS shape or round sections of class H: To consider the stress relaxation for the calculation of axial compression (clause 13.3.1), choose option [x].	Double-click or Space bar.
KL/r max	Enter the maximum slenderness for a member if it is different from the default value of 200.	Single click
Lx/	Enter a deflection criterion for strong axis.	Single click
Ly/	Enter a deflection criterion for strong axis.	Single click
Cantilever	Indicate if cantilever is located at node i or node j. If any of these cases, choose *Not applicable*. See topic Cantilever.	Double click
Kux Calculation	Automatic calculation or user-defined.	Double click
Kux	Kux value: Specific factor used for the calculation of cantilever effective compression length. This factor also depends on the load position.	Single click
Position of Load	By default, the load is applied at the centre of the section. If this is not the case, choose among options: *Top* or *Bottom*.	Double click

See also

The Tower Design module
Cantilever
The Steel tab (Project Configuration)

Member Bolted Connections Spreadsheet

This spreadsheet, located in the **Structure** menu at heading **Members**, will be available if you own the **Tower Design** module and if you activated design criteria in the **Members** Dialog box. Complete the information before running a steel design.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Member number.	No
Connection Model No. i	Connection model at node i.	Double-click
Connection Number No. i	Connection number at node i for the design of tower only.	Double-click
Connection Model No. j	Connection model at node j.	Double-click
Connection Number No. j	Connection number at node j for the design of tower only.	Double-click
Equation for KLx/r	According to S37-01 standard (towers): Equation that will be used for the calculation of effective length in x which takes into account the number of bolts of member end connections.	Double-click
KLx/r: Verify T/C ratio	If you want to verify the T/C ratio according to strong axis, choose option [x]. Refer to Ratio T/C	Double-click or Space bar
Equation for KLy/r	According to S37-01 standard (towers): Equation that will be used for the calculation of effective length in y which takes into account the number of bolts of member end connections.	Double-click
KLy/r: Verify T/C ratio	If you want to verify the T/C ratio according to weak axis, choose option [x]. Refer to Ratio T/C	Double-click or Space bar
Equation for KLz/r	According to S37-01 standard (towers): Equation that will be used for the calculation of effective length in z which takes into account the number of bolts of member end connections. If single steel angles are not oriented in an orthogonal axis system, the drop-down list box will not be activated.	Double-click

Guyed Tower

Guy Member

To define a member as a guy, you must choose the option *Guys* in the "Usage" drop-down list box included in the **Member** tab (**Member Characteristics** dialog box).

Guy Section and Material

Guys have been included in our databases, in the **Round Sections** spreadsheet and in the **Steel Materials** spreadsheet. The name of guy sections begins with letters GS (*Guy Strand)* and BS (*Bridge Strand).*

Ex.: Guy BS19x5/8" = A 19 wire *Bridge Strand* guy of 5/8" diameter. Material must be "BS19".

Young Modulus and Guy Area:

In VisualDesign, the definition of cables includes the cable nominal diameter and area represents the real amount of steel. Consequently, the Young modulus defined in the corresponding materials is calculated considering the real area of steel. Young modulus is the average value of *Prestretch* E (bigger value) and E that is calculated by the manufacturer (which is smaller). In the same way, density is computed considering the real weight of cable.

Dead Load of Guys

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. .

Automatic Split of Guy Members

When modeling guys, they must be split up into smaller pieces in order to get accurate results and to speed up convergence of computations.

Use function **Multiple Split** and check the specific option to automatically divide guys into smaller members of appropriate lengths. They are as follows:

1/2" guy: lengths of about 1 m.

1/2" or 5/8" guy: lengths of about 2 m.

3/4" guy: lengths of about 3 m.

Guy Tensile Strength

Calculation of Tr:

According to S16-01 and S37-01 steel design codes, VisualDesign calculates the tensile strength as follows:

$Tr = \phi AFy$ with ϕ equal to 0.6

Remark This calculation is only applied for a "Guy" usage. Select this usage in the "Usage" drop-down list box of **Member** tab (**Characteristics of the Member** dialog box).

Analysing a Guyed Tower

To attain convergence when analysing a guyed tower, do the following:

- The guy sections must be of the GS (guy strand) or BS (bridge strand) types and the "Guy" usage must be selected in the **Member Characteristics** dialog box.

- Select the [(D) Guy] type of dead load to these members because the load factor for guy dead load is different from the structure one.

- Select tension /compression axial end conditions for guy members.

- Guys must be modeled continuous (+----+) except at ends (support and attached member) where they can be hinged.

- Guys must be split into smaller members using the function **Multiple Split.** The option *Automatic splitting* must be activated. When clicking OK, VisualDesign will split selected guys into small members according to the guy diameters.

- The number of required iterations may be up to 100 in order to attain convergence for this type of structure. Enter 100 in the **Analysis** tab of **Project Configuration**. Activate a non-linear type of analysis.

Steel Specification

Steel Specifications Spreadsheet

This spreadsheet is completed if the steel specifications generator has been used. Refer to chapter 10 *Steel design* for more details.

	Number	Code	Type of analysis	Optimization	Group of sections	Shape	Material
6							
5	S37-Design_2L	CAN/CSA-S37-01	Design	Area	Canada	2EL	350G/W/WT/AT
6	S37-Design_L	CAN/CSA-S37-01	Design	Area	Canada	L (b=d)	350G/W/WT/AT

Specifications must be assigned to members that will be design or verify. The type of analysis (design or verification) and design code or Standard must be specified, among others.

Design Specifications

A steel material, a group of sections and a type of shape must be selected in each specification. Therefore, if different shapes are composing the structure, specifications must be added in the spreadsheet because each type of shape must possess its specification.

Interaction can be specified in this spreadsheet. Two options are available, namely *Compression/Tension* or *Standard*, meaning Compression & Bending.

If many types of steel shapes are composing the structure, simply add specifications (insert lines at the end of the spreadsheet), select the steel shape and assign the appropriate specifications to members.

Adding specifications

Insert lines at the end of the spreadsheet (or copy/paste a line) and give a name to each one. Select the type of analysis, design code, a group of sections and a steel shape. Then, assign specifications to members. During optimization, VisualDesign will select the specified type of shape among the sections that are listed in the group of sections.

Member Design Groups

A specification must be assigned to each design group. The same specification can be assigned to many design groups. Refer to the Design Groups spreadsheet.

Specifications used for a verification

The selection of a shape, material and group of section is not required. VisualDesign will verify assigned shapes according to the chosen code or Standard.

During modeling

For each member that will be designed, select either a design group or a specification in the **Steel Design** tab (**Member Characteristics** dialog box). For each member that needs to be verified, select a check specification.

Look at the table below to know the definition of parameters that are included in this spreadsheet.

Group: Structural data

Column	Description	Editing
Specification ID	Automatically calculated	No
Number	16 alphanumerical characters	Single click
Code	Choose the standard that will be used for the design or verification of the structure. For bridge Evaluation, choose code S6-00.	Double-click
Type of Analysis	Indicate in which case the specification will be used for: Design or Verification. If you chose code S6-00, option "Bridge Evaluation" will be available.	Double-click
Optimization	Design: Choose a type of shape optimization for the design: Area, Inertia or Height of the section. Verification: Not required.	Double-click
Group of sections	Design: Choose the group of sections that can be used for the design. Verification: Not required.	Double-click
Shape	Design: Specify the steel shape that will be assigned to selected members. VisualDesign will search for appropriate shapes among the selected group of sections. Verification: Not required.	Double-click
Material	Design only: Select the steel material that will be assigned to optimized sections.	Double-click
HSS t (design)	If a HSS shape is specified as the design shape, you must select its design thickness in the list box: 1.0t CASC/Can), 0.9t (CISC/US) or 0.93t (AISC/US).	Double-click
Category	Cold-Formed Section only: Specify a category. This name must correspond to the one entered in the cold-formed section spreadsheet with respect to lower/upper-case. This column is not useful for other shapes.	Single click

Column	Description	Editing
Max. class	Design only: Specify the maximum class for optimized shapes.	Double-click
Max. height Mx	Design only: Specify the maximum height for optimized sections.	Single click
Min. height Mx	Design only: Specify the minimum height for optimized sections.	Single click
Max. width My	Design only: Specify the maximum width for optimized sections.	Single click
Min. width My	Design only: Specify the minimum width for optimized sections.	Single click
Maximum Capacity Factor	Design only: The design is done with respect to this given percentage of capacity. For example, if a value is inferior to 100%, it means that you are more severe than the chosen code.	Single click
Interaction	Design with S37-01 only: The "Compression-Tension" option is used to verify and design tower or antenna members according to compression and tension forces only (without bending). The "Standard" option (with bending) is automatically selected for all other users	Double-click

Design or verification of Bolted Connections

Column	Description	Editing
Analysis of Connections	Choose the type of analysis that will be done for connections: None, Verification or Design.	Double-click
Type of Connection	Type of bolted connection: Bearing or Slip-Resistant.	Double-click
Type of Hole	Specify if holes had been punched or drilled.	Double-click
Contact Surface	Specify the contact surface of connections: None, class A, class B or class C.	Double-click
Selection of Bolts	Choose the bolts that can be used for the design of bolted connections.	Double-click
Description	Give a description of this specification.	Single click

Assign Specifications to Members

Select one member or more, which will have the same bolted connections, and open the **Member Characteristics** dialog box.

- Activate design criteria in the **Member** tab and specify other parameters.

- Go to the **Steel Design** tab and assign a specification to the selected members.

- Finally, select the **Bolted Connection** tab and assign connection models to member(s) end nodes.

Selection of Bolts in a Steel Specification

Go to the **Steel Specifications** spreadsheet and double-click in the "Selection of Bolts" column. Choose as many types of bolts that can be used for the design of connections. VisualDesign will choose among that list. A dialog box will appear on the screen. Expand the roots and check the boxes corresponding to the type of bolts that you want VisualDesign to consider in the design.

If there are bolts missing in the list, add them in the **Bolts** spreadsheet (**Common** menu at heading **Bolts/Bolts**).

See also
Steel specifications spreadsheet
Bolts spreadsheet

Design Groups

Grouping Selected Elements

You must specify steel or timber specifications before grouping members.

This tool, available in **Structure** menu at **Groups** heading, allows creating design groups for members to get a more practical design.

Procedure

- Select members that you wish to group together (keep the [Ctrl] key down while you select members with your mouse);

- Select **Group Members** under **Structure/Groups** menu or use shortcut keys **Ctrl+G**.

- A dialog box will appear. Give a name to this group and choose the steel or timber specification for this design group.

- Press OK.
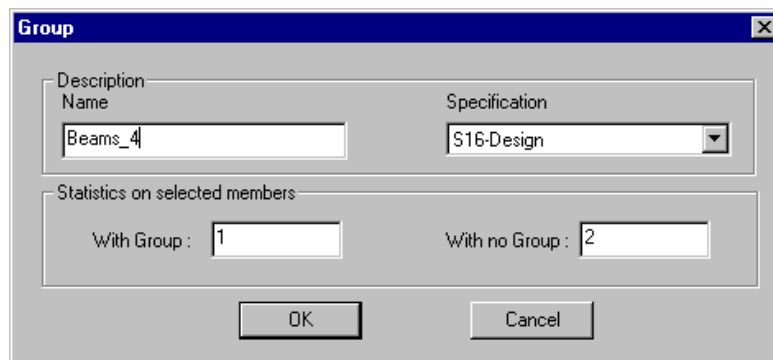
Design groups are automatically assigned to members in the **Steel Design** tab and they are listed in the **Steel Design Groups** spreadsheet (**Structure / Groups / Steel Members**.)

See also
Displaying Member Design Group
Steel Design
Steel Member Groups spreadsheet
Automatic Grouping of Elements

Grouping Members Automatically

This tool groups members having a symmetrical layout. Specifications must be assigned to members before grouping them. Members will be grouped together if they are assigned to the same specification.

Procedure:

- Select **Structure/Groups / Auto Group**. This dialog box will appear on screen:

- Members can be grouped according to one of these options:

  - Grouping continuous members. These members must be rectilinear and assigned to the same specification.

  - Grouping mirror members according to vertical planes. The user can specify the horizontal position of the mirror plane.

- The assigned specification is corresponding to the one that was assigned to the first member composing this group. The group number corresponds to the number of the first member part of this group.

Steel Design Groups Spreadsheet

Group: Structural Data

Column	Description	Editing
ID of group	Automatically calculated	No
Number	16 alphanumerical characters defining this group of members.	Single click
Specification	Choose the steel specification that will be applied to this group.	Double-click

See also

Steel Specification spreadsheet
Steel Design Procedure

Viewing a Design Group

To look at a design group, do the following:

- Select the **Steel Design Groups** spreadsheet under **Structure/Groups**.

- Highlight the line that corresponds to the group that you want to look at.

- Click OK.

Members that are part of this group will be highlighted on screen.

If you do not like the way members are grouped, delete lines in the **Steel Design Groups** spreadsheet and use the **Group Members** function.

Loads Definition

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

Load Cases tab

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 Load Cases 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 that represent the accidental torsion effects.

Load Case Families

A family is useful when using the **Load Combination Generation Wizard** when some 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.

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.

Allowed families with CAN/CSA-S37-01 Standard:

- Wind, and Autowind;

- Ice, and Auto-ice.

Example

We have six load cases of the *Wind* type (W1, W2, ... W6). Usually, when using the **Load Combination Generator**, six load combinations of the type D + W will be generated, each including the dead load and one wind load case.

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.

CAN/CSA-S37-01 Load Cases

To select load cases that belong to 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

Type of Load	Description	Family (1)	Note
(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.

Wind tab (Loads Definition)

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 when ice loads are generated. 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

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

Column	Description	Editing
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

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
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

Column	Description	Editing
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

Wind on Panels
Steel tab (Project Configuration)
Defining and Applying Loads on Elements
Modifying Applied Loads
Deleting Loads Applied on Elements

Ice tab (Loads Definition)

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 the density of ice.	Single click
Member Usage	Select the member usage for which this load will be apply.	Double-click

See also

Automatic Generation of Ice Loads
Generating Ice Loads
Load Case Families
Defining and Applying Loads on Elements
Modifying Applied Loads
Deleting Loads Applied on Elements

Automatic Generation of Loads

Automatic Generation of Wind Loads

Method for Wind Calculation

The method that will be used for the calculation of wind load must be selected in the **Steel** tab of **Project Configuration** dialog box before selecting the generator. Available methods are according to standard *CAN/CSA-S37-01* or *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.

Generation of Loads

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 is automatically calculated. Only members can be loaded with wind. VisualDesign™ transfers wind loads as forces to 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 can be a *Wind* type of load, and a Cd of 1.0 is required, as per S37-01 Standard.

During the 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 software 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 for calculating wind pressure (*Steel* tab of **Project Configuration**), the dialog box will include the following parameters:

Wind Pressure according to CAN/CSA-S37-01:

Field	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:

Heading	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.

- Then, activate the "Load Case" mode on Activation toolbar and select the *Auto-Wind* load in the drop-down list box.

- Now, select the **Automatic Generation of Wind Loads** function under the **Loads** menu. You can modify parameters in the **Wind Load** dialog box.

See also
Automatic Generation of Wind Loads

Wind on Panels

This tool is useful to calculate wind force on a selected panel (or more), with or without ice coating on members, and with or without equipment attached on these members. It can be used before an analysis.

Restriction for the selection of panels

- Members must be in the same plane;

- The selection must include at least one *Upright* member;

- This *Upright* member must be part of a closed polygon.

Procedure

- Create a *Wind* type of load in the **Loads Definition** spreadsheet.

- Select the **Wind** tab and enter the wind parameters to be applied to panels. According to S37-01 Standard, Cd must be equal to 1.0 for wind applied on members. (For guys, Cd must be equal to 1.2 and the wind load must be of the "Auto-wind" type.)

- Activate the Load Case mode and select this *Wind* load case in the list box.

- Select a panel by clicking on members. To obtain wind forces considering the projected area with or without ice coating, or with or without equipments, go to **Loads** menu and select **Load Cases** / **Automatic Generation** / **Wind on Panels**.

- If equipment is attached to this panel, enter the corresponding bare flat area (Afa), round area (Ara), and radial ice area (Aia) in the spreadsheet. Parameters will be automatically recalculated.

- Press the "Add" button to add these loads in this load case.

Here is a description of fields included in this spreadsheet.

Group: Load case: (title)

Column	Description	Editing
ID	Automatically calculated	No
Node Number	Loaded Node.	No
Level	Level of this node relatively to this panel.	No
Coord. Y	Node coordinate in structural model.	No
qh	Site-specific wind pressure profile.	No
Pressure	Design wind pressure.	No
Af	Face area of flat members, bare.	No
Afa	Enter the face area (flat) of equipment, bare, if any.	Yes
Ar	Face area of round members, bare.	No
Ara	Enter the face area (round) of equipment, bare, if any.	Yes
Ai	Face area of radial ice on members.	No
Aia	Enter the face area of radial ice on equipment, if any.	Yes
As	Net projected area of this panel.	No
Rs	Strength ratio: As/Ag.	No
Kdf (1)	Factor multiplying Cdf and depending on solidity ratio Rs.	No
Cdf	Drag factor for flat members.	No
Kdr (1)	Factor multiplying Cdr and depending on solidity ratio Rs.	No
Cdr	Drag factor for round members.	No

Column	Description	Editing
Angle	Angle of wind applied on this panel.	No
F	Total wind load applied on this panel.	No
Fx	X component of wind force F	No
Fy	Y component of wind force F	No
Fz	Z component of wind force F	No

Note (1): Refer to clause 4.9.1.1 – CAN/CSA-S37-01 standard.

See also
Wind tab – Loads Definition spreadsheet

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:

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 design cycles.

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**. The **Ice Load** dialog box will appear on screen.
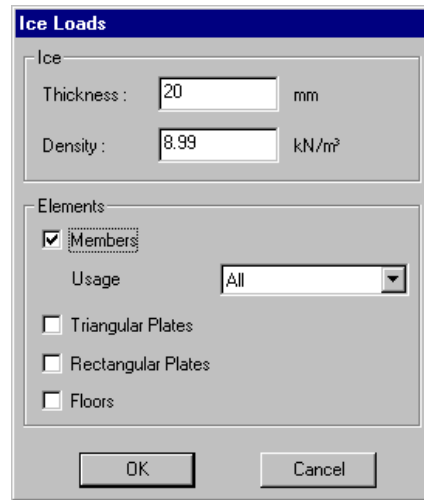
- Specify the ice thickness and the type of element on which ice coating will be applied. Click OK.

- 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 loads applied on members 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 with the equation below:

W = Density * Thickness * S

"S" is the area of ice surrounding the shape and is calculated with the equations presented below:

General

$$S = e (p + \pi e)$$
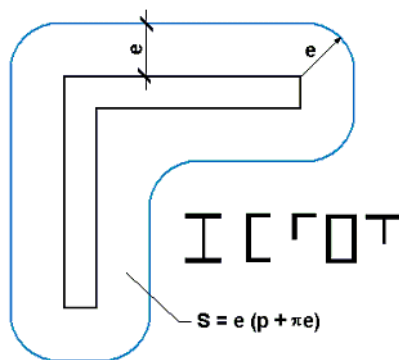
Round shapes
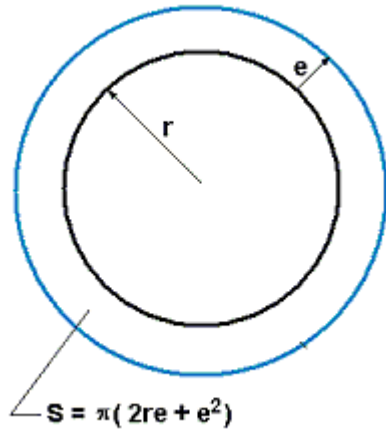
$$S = \pi(2re + e^2)$$

2L shapes
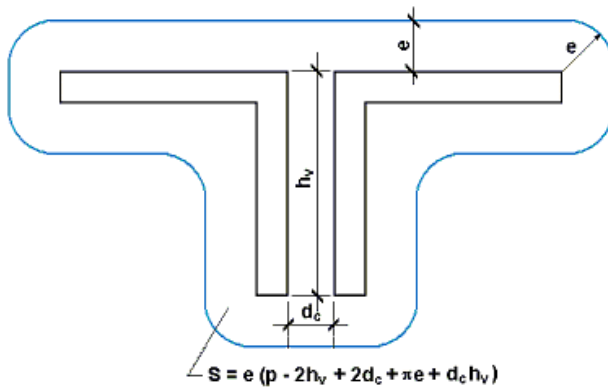
$$S = e (p - 2h_v + 2d_c + \pi e + d_c h_v)$$

See also

Generating Ice Loads
Loads Definition spreadsheet
Tower Design Module

Numerical Results

Steel Design Results Spreadsheet

The spreadsheet includes, among others, the name of shapes, critical load combination, code provision that controlled the design, members design load, bending, shear and axial resistances, net area and members stability. A message also informs you if the design is OK or not. If not, lines will be coloured in red in the spreadsheet. However, the colour will not appear at the printing of the spreadsheet.

As soon as the design is finished, the "Steel Design Results" mode ![icon] is automatically activated. You can look at results in many ways:

- Go to **Results** menu and select the heading **Structure Design /Steel.** If no members were selected, the spreadsheet will include all members;

- Double-click on a member to access the spreadsheet. Only the results for this member will be included in the spreadsheet;

- Click on many members while pressing down the [Ctrl] key and click on the **Properties** icon ![icon]. Only the selected members will be part of the results spreadsheet.

In the lower part of this spreadsheet, you will notice three buttons. They give access to internal forces results and to a print of a design brief (for the selected member in the spreadsheet). A print preview is also supplied for the design brief. To learn more, see the following topics:

Design Brief
Access to internal forces and deflections
Print Preview of Design Brief

Group: Design Results

Column	Description	Editing
Number	Section number	No
Group	Group to which the member belongs.	No
Section	Name of shape.	No
Load Combination Mf + Nf	Load combination that controlled the combined bending and compression forces in the member.	No

Column	Description	Editing
Design load Mf + Nf	Member design load considering the combined bending and compression forces. A value greater than 100 means that section capacity is too small.	No
Load Combination Shear	Load combination that controlled the shear force in the member.	No
Design load Shear	Member Design load considering the maximum shear force. A value greater than 100 means that the shear capacity is insufficient.	No
Clause	Clause that controlled the design	No
Class Bending Mx	Class considered in the calculation of Mx.	No
Class Bending My	Class considered in the calculation of My.	No
Class Web	Class considered in the calculation of web buckling resistance	No
Class Compression	Class considered in the calculation of compression strength.	No
Type of Mfx	Indicates if a positive or negative moment (strong axis) controls the design for flexion.	No
Mrx Lu=0	Resisting moment (strong axis) when Lu = 0	No
Mrx Lu>0	Resisting moment (strong axis) when Lu > 0	No
Lux	Unsupported length (strong axis)	No
w2x	Factor allowing for the increase of the resisting moment of a laterally unsupported segment (strong axis) caused by a moment gradient. Please refer to topic Elastic Lateral-Torsional Buckling.	No
Type of Mfy	Indicates if a positive or negative moment (weak axis) controls the design for flexion	No
Mry Luy=0	Resisting moment (weak axis) when Lu = 0	No
Mry Luy>0	Resisting moment (weak axis) when Lu > 0	No

Column	Description	Editing
Luy	Unsupported length on weak axis. This value is null for sections with single and double symmetry because there is no lateral buckling on weak axis. VisualDesign supposes that all sections strong axis for bending is around local x-axis.	No
w2y	Factor allowing for the increase of the resisting moment of a laterally unsupported segment (weak axis) caused by a moment gradient. Please refer to topic Elastic Lateral-Torsional Buckling.	No
Type of Nz	Indicate tension or compression.	No
Tension Tr	Tensile strength of the section (Tr min using net area).	No
Cr Resistance	Compressive resistance of the section. (Where $\lambda=0$).	No
Cr Stability	Compressive resistance according to KL/rx, KLy/ry, KL/rz.	No
KLx	Effective compression length on strong axis.	No
KLy	Effective compression length on weak axis.	No
KLz	Effective compression length according to orthogonal axis system.	No
KLt	Effective compression length for torsional buckling.	No
KL/rx	Slenderness ratio according to the strong axis	No
KL/ry	Slenderness ratio according to weak axis	No
KL/rz	Slenderness ratio according to orthogonal axis system. This ratio is calculated according to equations of Standard S37-01 for owners of the Tower and Antenna design module and it is chosen in the Steel Design tab for single steel angles.	No
KL/r max	Maximum slenderness ratio	No
T/C Ratio	Tension/Compression ratio corresponding to maximum design load for this member.	No

Column	Description	Editing
Net Area	Net Area of the section: gross area minus bolt holes and affected by the reduction coefficient.	No
Vrx	Shear resistance on weak axis.	No
Vry	Shear resistance on strong axis.	No
Trz	Resistance to torsion.	No
Message	Design result: Ok, Insufficient capacity, etc.	No
Deflection	Status of the deflection result: n/a, Ok < Allowable Deflection, Too much deflection.	No
Load Combination Mx	Load combination that governs the deflection criterion.	No
Lx	Calculated span between inflexion points	No
Deflection (Mx)	Deflection between two inflexion points	No
Lx/	Ration between Lx/Deflection (Mx)	No
Load Combination My	Load combination that governs the deflection criterion.	No
Ly	Calculated span between inflexion points	No
Deflection (My)	Deflection between two inflexion points	No
Ly/	Ratio between Ly/Deflection (My)	No
Stud (0:M+)	Number of required studs to transfer the shear stress. Number corresponding between the zero moment and the maximum positive moment	No
Stud (0:M-)	Number of required studs to transfer the shear stress. Number corresponding between the zero moment and the maximum negative moment	No

See also
Design Brief
Access to diagrams from the Design Results spreadsheet
Print Preview of Design Brief
Failure Modes - Steel Design Results (S37-01)

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
Orientation	Orientation of this reaction: local or global axes system.	No

Spring Supports Only

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
Soil Pressure z-direction	Soil pressure acting on this spring support towards direction z, considering its tributary surface.	No

Internal Forces and Deflections for Design Groups

Group: Envelope Results: (title)

Column	Description	Editing
Number	Group number (12 alphanumeric characters).	No
Shape	Chosen shape for this design group.	No
Value	Type of results: Max. or Min. An empty field indicates a standard analytical result.	No
Bending Mx	Min. or max. bending moment on strong axis for this design group.	No
Shear Vy	Min. or max. shear force on strong axis for this design group.	No
Bending My	Min. or max. bending moment on weak axis for this design group.	No
Shear Vx	Min. or max. shear force on weak axis for this design group.	No
Axial Nz	Min. or max. axial force for this design group (positive in tension).	No
Torsion Tz	Max or min. torsion force for this design group.	No
u (weak axis)	Max or min displacement on weak axis for this design group.	No
v (strong axis)	Max or min displacement on strong axis for this design group.	No
w (axial)	Max or min axial deformation for this design group (accurate solution only at member ends).	No

Bolted Connections Design Results

When the steel design is finished, go to **Results** menu and select heading **Structure Design / Bolted Connections.**

The **Bolted Connections Design Results** includes information for each member end designed connections (node i and j). Look at the table below to know the definition of each column:

Group: Design Results (title)

Column	Description	Editing
ID	Automatically calculated.	No
Node Number	Structure node number	No
Shape	Member shape	No
Node	Node i or j. Results are given for each node.	No
Connection Number	Connection number at this node.	No
Connection Model Node i or j	Connection model at this node. See topic Connection Models	No
Type of Analysis	Type of analysis: Verification or Design.	No
Nos. of bolts	Number of bolts for this connection.	No
Bolts	Type of bolts used for this connection.	No
Nos. of transverse lines	Number of transverse lines for this connection. See topic Bolts Layout.	No
Nos. of longitudinal lines	Number of longitudinal lines for this connection. See topic Bolts Layout	No
Bolts Layout	Bolts layout: In line, Staggered A, or Staggered B. See topic Bolts Layout	No
Plate ep	Longitudinal distance measured from free edge of plate to the first bolt in a row.	No
Plate t	Designed plate thickness.	No
Other section	For connection model 5: specify the other section used as the connector.	No
Member em	Longitudinal distance measured from free edge of steel angle to the first bolt in a row.	No
Member p	Longitudinal distance measured centre-to-centre between two bolts.	No
Member g1	Transverse distance measured from outside edge of bent angle to the nearest bolt row.	No

Column	Description	Editing
Member g2	Transverse distance measured centre-to-centre between two consecutive bolt rows.	No
Member e	Transverse distance measured from free edge of member to nearest bolt hole.	No
Member Br	Bearing resistance of the connected member according to clause 6.5.2.2 i) and ii).	No
Member Tr1	Member tensile strength according to the failure mechanism for bolts layout or bolt bearing.	No
Member Tr2	Member tensile strength considering shear lag and member net area, as specified in the Steel Design tab of Member Characteristics dialog box.	No
Member Tf	Maximum factored tension in member.	No
Member Failure	Failure mechanism and failure path. See topic Failure Mechanisms in Bolted Connections	No
Plate Br	Bearing resistance of the plate in this connection according to clause 6.5.2.2 i) and ii).	No
Bolts Vr	Shear resistance of bolts in this connection.	No
Load Combination	Most critical load combination used for the design of this bolted connection.	No
Nf	Axial force in this bolted connection.	No
% Design load	Design load of the bolted connection.	No
Message	Message from VisualDesign about the design of this connection.	No
Inconsistent Dimensions	If some dimensions do not follow the code requirement, they will be written here.	No

See also

Connection Models
Bolts Layout
Failure Mechanisms for Bolted connections
Failure Mechanisms for Members
Tower Load Combination Results
Tower Load Combination Results
Failure Modes - Steel Design Results
Steel Design Results

Failure Modes - Steel Design Results (CAN/CSA-S37-01)

This spreadsheet, which is accessible through **Results / Structure Design**, describes the failure modes for each member and corresponding bolted connections, for tension/compression interaction only. Members must be verified or designed using the CAN/CSA-S37-01 Standard for towers.

The spreadsheet is divided into two tabs: **Members** and **Bolted Connections**. Please consult the "Derogation" columns to know the failure modes for members and bolted connections.

Failure Modes

The following failure modes are considered in VisualDesign:

For members: Tension (considering the gross and net area of member), the maximum slenderness ratio (KL/r max), compression (pure compression and compression considering slenderness ratios for x, y, and z), and T/C ratios.

For bolted connections: Shear, tension (considering the gross area and net area 1 & 2), and compression.

Load Combinations and Envelopes

The spreadsheet gives results for a selected load combination or a selected envelope. For the latter case, the critical load combination is given for each failure mode.

The Member tab

Group: Design Results

Column	Description	Editing
ID	Member ID number. Automatically calculated.	No
Number	Number of this member.	No
Shape	Steel shape assigned to this member.	No
KL/r maximum specified	Maximum effective slenderness ratio, as specified in the steel specification.	No
KL/r maximum obtained	Maximum effective slenderness ratio obtained for this member.	No
KL/r Derogation	Check box [x]: The calculated effective slenderness ratio exceeds the maximum specified.	No

Column	Description	Editing
Design load maximum specified	Maximum design load, as specified in the steel specification.	No
Design load maximum	The calculated maximum design load for this member.	No
Design load Derogation	Check box [x]: The calculated design load exceeds the maximum specified design load.	No
KL/rx KL/r max.	Maximum effective slenderness ratio calculated on x-axis.	No
KL/rx Load Combination	Name of the load combination that controlled the design load of the member, considering the maximum effective slenderness ratio on x-axis.	No
KL/rx Ratio T/C	The calculated T/C ratio, considering the maximum effective slenderness ratio on x-axis.	No
KL/rx Derogation	Check box [x]: The maximum effective slenderness ratio on x-axis exceeds the maximum specified.	No
KL/ry KL/r max.	Maximum effective slenderness ratio calculated on y-axis.	No
KL/ry Load Combination	Name of the load combination that controlled the design load of the member, considering the maximum effective slenderness ratio on y-axis.	No
KL/ry Rapport T/C	The calculated T/C ratio, considering the maximum effective slenderness ratio on y-axis.	No
KL/ry Derogation	Check box [x]: The maximum effective slenderness ratio on y-axis exceeds the maximum specified.	No
KL/rz KL/r max.	Maximum effective slenderness ratio calculated on z-axis.	No
KL/rz Load Combination	Name of the load combination that controlled the design load of the member, considering the maximum effective slenderness ratio on z-axis.	No
KL/rz Rapport T/C	The calculated T/C ratio, considering the maximum effective slenderness ratio on z-axis.	No
KL/rz Dérogation	Check box [x]: The maximum effective slenderness ratio on z-axis exceeds the maximum specified.	No

Column	Description	Editing
Tf max	Maximum tension force calculated for this member.	No
Tf max Load Combination	Name of the load combination that created the maximum tension force in this member.	No
Cf max	Maximum compression force calculated for this member.	No
Cf max Load Combination	Name of the load combination that created the maximum compression force in this member.	No
Tension Gross area Resistance	The tensile strength of the member, considering the gross area.	No
Tension Gross area Design load	The tensile design load of the member, considering the gross area.	No
Tension Gross area Derogation	Check box [x]: The tensile strength of this member is insufficient, considering its gross area.	No
Tension Net area Resistance	The tensile strength of the member, considering the net area.	No
Tension Net area Design load	The tensile design load of the member, considering the net area.	No
Tension Net area Derogation	Check box [x]: The tensile strength of this member is insufficient, considering its net area.	No
Compression Pure Resistance	Pure compression strength of this member.	No
Compression Pure Design load	The member design load regarding pure compression.	No
Compression Pure Derogation	Check box [x]: The pure compression strength of this member is insufficient.	No

Column	Description	Editing
Compression Slender x Resistance	The member compression strength, considering the maximum effective slenderness ratio on x-axis.	No
Compression Slender x KL/rx	The maximum effective slenderness ratio on x-axis, used to calculate the member compression strength.	No
Compression Slender x Load Combination	Name of the load combination that created the maximum compression force in this member, considering the maximum effective slenderness ratio on x-axis.	No
Compression Slender x Ratio T/C	The calculated ratio T/C, considering the maximum compression force and the maximum effective slenderness ratio on x-axis.	No
Compression Slender x Design load	The member design load regarding the compression force and the maximum effective slenderness ratio on x-axis.	No
Compression Slender x Derogation	Check box [x]: The compression strength of this member is insufficient regarding the maximum effective slenderness ratio on x-axis.	No
Compression Slender y Resistance	The member compression strength, considering the maximum effective slenderness ratio on y-axis.	No
Compression Slender y KL/ry	The maximum effective slenderness ratio on y-axis, used to calculate the member compression strength.	No
Compression Slender y Load Combination	Name of the load combination that created the maximum compression force in this member, considering the maximum effective slenderness ratio on y-axis.	No
Compression Slender y Ratio T/C	The calculated ratio T/C, considering the maximum compression force and the maximum effective slenderness ratio on y-axis.	No
Compression Slender y Design load	The member design load regarding the compression force and the maximum effective slenderness ratio on y-axis.	No
Compression Slender y Derogation	Check box [x]: The compression strength of this member is insufficient regarding the maximum effective slenderness ratio on y-axis.	No

Column	Description	Editing
Compression Slender z Resistance	The member compression strength, considering the maximum effective slenderness ratio on z-axis.	No
Compression Slender z KL/rz	The maximum effective slenderness ratio on z-axis, used to calculate the member compression strength.	No
Compression Slender z Load Combination	Name of the load combination that created the maximum compression force in this member, considering the maximum effective slenderness ratio on z-axis.	No
Compression Slender z Ratio T/C	The calculated ratio T/C, considering the maximum compression force and the maximum effective slenderness ratio on z-axis.	No
Compression Slender z Design load	The member design load regarding the compression force and the maximum effective slenderness ratio on z-axis.	No
Compression Slender z Derogation	Check box [x]: The compression strength of this member is insufficient regarding the maximum effective slenderness ratio on z-axis.	No

The Bolted Connections Tab

Group: Design Results

Column	Description	Editing
ID	ID number of this bolted connection. Automatically calculated.	No
Number	Number of this bolted connection.	No
Design Load Derogation	Check box [x]: The calculated design load for this bolted connection exceeds the maximum specified.	No
Tf max	The maximum calculated tension force acting on this bolted connection.	No
Tf max Load Combination	Name of the load combination that created the maximum tension force in this bolted connection.	No
Cf max	The maximum calculated compression force acting on this bolted connection.	No

Column	Description	Editing
Cf max Load Combination	Name of the load combination that created the maximum compression force in this bolted connection.	No
Connection i/j Shear Resistance	Calculated shear resistance of this bolted connection.	No
Connection i/j Shear Design Load	Calculated design load regarding the maximum shear force obtained for this bolted connection.	No
Connection i/j Shear Derogation	Check box [x]: The shear strength of this bolted connection is insufficient.	No
Connection i/j Support Tension Resistance	Calculated tensile resistance of this bolted connection, considering the gross area of the member.	No
Connection i/j Support Tension Design Load	Calculated design load of this bolted connection, regarding the maximum tension force obtained, considering the gross area of the member.	No
Connection i/j Support Tension Derogation	Check box [x]: The tensile strength of this bolted connection is insufficient.	No
Connection i/j Support Compression Resistance	The calculated compression strength of this bolted connection.	No
Connection i/j Support Compression Design Load	The calculated design load of this bolted connection regarding the maximum compression force obtained.	No
Connection i/j Support Compression Derogation	Check box [x]: The compression strength of this bolted connection is insufficient.	No
Connection i/j Tension Net Area 1 Resistance	The calculated tensile strength of this bolted connection, considering the net area 1 of the member.	No

Column	Description	Editing
Connection i/j Tension Net Area 1 Design Load	The calculated design load regarding the maximum tension force in this bolted connection, considering the net area 1 of the member.	No
Connection i/j Tension Net Area 1 Derogation	Check box [x]: The tensile strength of this bolted connection is insufficient, considering the net area 1 of the member.	No
Connection i/j Tension Net Area 2 Resistance	The calculated tensile strength of this bolted connection, considering the net area 2 of the member.	No
Connection i/j Tension Net Area 2 Design Load	The calculated design load regarding the maximum tension force in this bolted connection, considering the net area 2 of the member.	No
Connection i/j Tension Net Area 2 Derogation	Check box [x]: The tensile strength of this bolted connection is insufficient, considering the net area 2 of the member.	No

Graphical Results

Tower Results

Go to menu **Results/ Load Combinations** or **Results/ Envelopes** and choose heading **Tower Results**.

The **Tower Results** dialog box allows you to look at the tower behaviour in a global manner, for a chosen load combination or envelope, and to print the displayed diagram in this dialog box.

Select one of the following results among the drop-down list box that is located in the upper part of the dialog box:

- Twist Angle;

- Tilt Angle;

- Total angle;

- Average Deflection in the x direction;

- Average Deflection in the z direction;

- Total Average Deflection;

- Bending moments (Mx, Mz) in Mast;

- Shear forces (Vx, Vz) in Mast;

- Internal Forces in Upright Members and Compression Strength;

- Internal Forces in Secondary Members and Compression Strength;

- Internal Forces in Horizontal Secondary Members and Compression Strength;

- Upright Member Design Loads;

- Diagonal Member Design Loads;

- Horizontal Member Design Loads.

- Results according to an initial condition

Results according to an initial condition

Results may be calculated from an initial condition that was chosen by the user in this dialog box. Therefore, displayed values will be withdrawn or added, depending on this initial condition and calculated values. This applies for the following results: Twist and tilt angles, total angle, average x and y displacement and total average displacement.
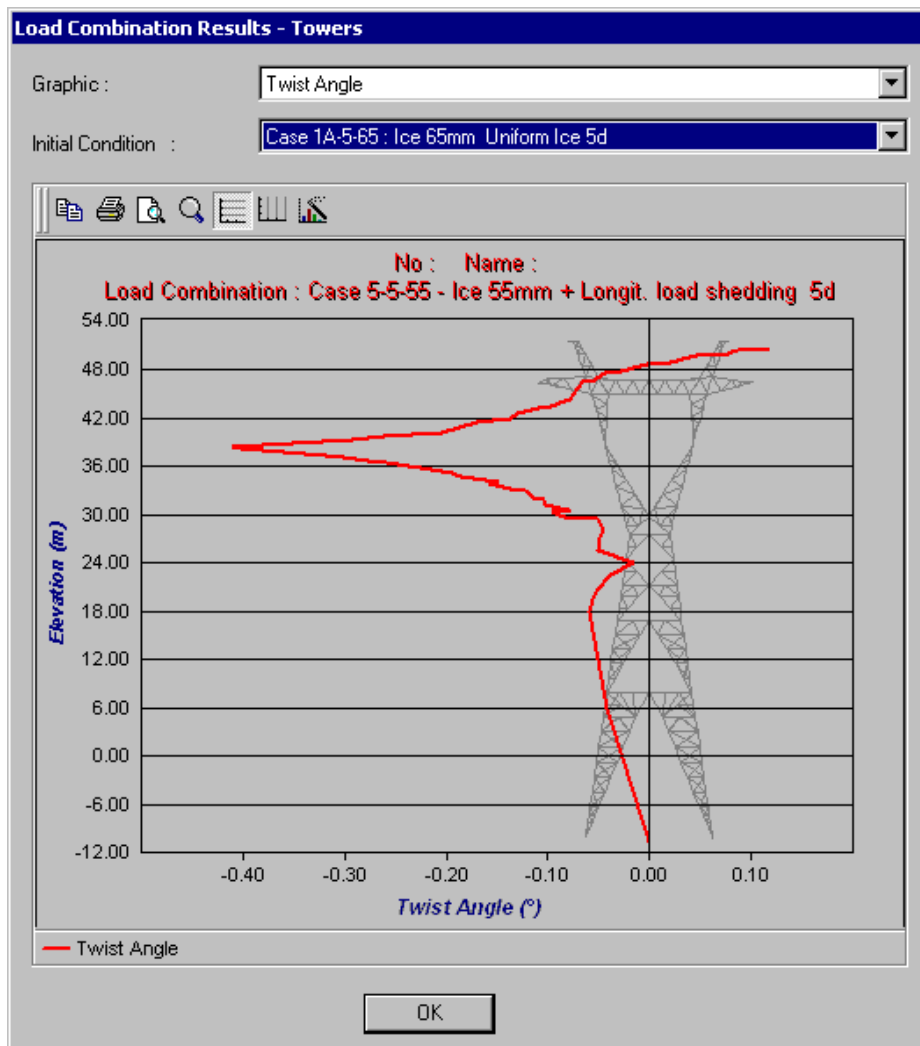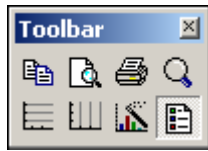
Graph Toolbar

A toolbar is also supplied to let you modify the display of diagrams. This toolbar comes from *ChartFX* and you will find its description in the following topic ChartFX toolbar.

See also

Failure Modes - Steel Design Results

Graphs Toolbar (ChartFX)

When activating graphical results of prestressed concrete analysis (**Results/Graphs** menu), a toolbar is provided in all the results diagrams to help you managing diagrams (copy, print, view options, etc.).

You will find below, a description of functions represented by icons and also a description of dialog boxes that can be called up by pressing some icons.

Copy to clipboard:

Click on this icon to copy the diagram as a Bitmap, as a Metafile, as Text (data only) or as OLE Object.

Print Preview:

Click on this icon to call up the *Page setup* dialog box. Complete the dialog box and look at the diagram. The table below explains the parameters included in the dialog box.

Field	Description
Options	
Separate Legend	Check this box to move the legend outside the diagram box.
Use Printer Resolution	Check this box to use the printer resolution.
Force Colors	If your printer is black and white, the printer driver will match the chart colors to a specific grayscale pattern.
Margins (inches)	Define the left, top, right and bottom margins, in inches.
Orientation	
Portrait	Check this box to look at the diagram in the vertical way.
Landscape	Check this box to look at the diagram in the horizontal way.
Use by default	Check this box to use the above parameters by default.

Print: 🖨️

Press this icon to print the diagram. In the *Layout* tab, choose the *Landscape* orientation. This option must be selected even if you activated the *Landscape* orientation in the *Page Setup* dialog box (Print Preview function).

Zoom: 🔍

Click on this icon and, with your cursor, draw a window around the elements that you wish to zoom in. The icon will still be activated to let you zoom again. To go back at a normal view, click on the icon again.

Horizontal grid: 📄

Click on this icon to display a horizontal grid.

Vertical grid: 📊

Click on this icon to display a vertical grid.

Series Legend: 📋

Click on this icon to mask or display the series legend of the diagram.

Properties: 📈

Click on this icon to modify the diagrams view options through the **Chart FX 98 Properties** dialog box. It is composed of the ***General***, ***Series***, ***Axes*** and ***3D*** tabs. Look at the tables below to learn more about each tab.

The *General* tab: This dialog box allows you to set general settings in the chart, like Major Unit Gridlines and Colors for the chart.

Field	Description
Effects	
3D	Check this box to switch from 2D/3D views.
Cluster (Z-Axis)	When displaying a clustered chart, each series will have its own position in the z-axis. This means, if you have a 3 series chart, each data series will occupy one row of data and there will be 3 rows (z-axis clusters) in the chart.
Stacked	This function is not activated.
Axes styles	This option allows you to change the axis drawing style.

Field	Description
Grid lines	This option allows you to draw vertical and/or horizontal gridlines.
Colors	
Palette	This option allows you to change the entire chart palette. This will affect all elements in the chart.
Scheme	This option allows you to change the color scheme for markers.
Background	This option allows you to set a color for the background. This background is the box where the chart is enclosed.
Chart box	This option allows you to set a color for the chart box. This is the background where the markers are enclosed.
Title	This option allows you to set a Top Title for the chart.

The *Series* tab: This dialog box allows you to set specific settings, including visual attributes for series in the chart.

Field	Description
Combo Box	When « All series » is selected in the Combo Box, settings will apply to all series in the chart and the property set is different if a specific series is selected. Therefore, 2 different screen shots are included for this particular dialog.
Marker fill	This option allows the user to change colors for series.
Lines	
Same color as markers	This option is activated if a specific series is selected. It will match the color of lines to the one used for markers.
Custom	This option is activated if a specific series is selected. Change the line color, style and weight for this specific series. Look at the sample that is displayed in the dialog.
Color	Choose a line color for this specific series.
Style	Choose a line style for this specific series.

Field	Description
Weight	Choose a line weight for this specific series.
Markers	Markers represent points on the series.
Gallery:	This option allows you to change the chart type
Shape:	Choose a marker shape such as cylinders or cones for a *bar chart* or *cube chart*.
Show every:	Repetition of markers
Size:	Markers size
Show point labels	This option allows you to show/hide point markers on the series.
Visible	This option allows you to show/hide a particular series in the chart.
Sample	Look at the sample displayed in this box.

The *Axes* tab: This dialog allows you to manipulate axis settings, including Major Interval, Minor Intervals, Scales, Min, Max, and Gridlines & Tickmarks.

Field	Description
Combo Box	Choose the axis for which you want to look at parameters. To change parameters, press on the *Details…* button posted in the bottom of the dialog. Check the "Visible" box to show/hide a particular axis, including labels and tickmarks.
Major unit	This option allows you to set the Major interval for the selected axis. Check the "Show gridlines" box to show/hide gridlines on the major interval.
Tick mark type	Choose the tickmark type for the major interval.
Minor unit	This option allows you to set the Minor interval for the selected axis. . Check the "Show gridlines" box to show/hide gridlines on the minor interval.
Tick mark type	Choose the tickmark type for the minor interval
Details… Button	Press this button to access the **Axis Properties** dialog to change the selected axis properties. See explanation below.

The **Axis Properties** dialog: This dialog box includes the selected axis properties and is composed of the following tabs: *General*, *Scale*, *Labels* and *Grid lines*.

The *General* tab:

Field	Description
Selected Axis	
Major unit	Choose the unit that will define the major interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of major interval.
Minor unit	Choose the unit that will define the minor interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of minor interval.

The *Scale* tab: This dialog allows you to manipulate axis scales, including min, max, and logarithmic scales among others.

Field	Description
Selected Axis	
Minimum	This option allows you to set the minimum for the selected axis.
Maximum	This option allows you to set the maximum for the selected axis.
Scale unit	Set the scale unit for the selected axis.
Format	Set the axis format: None, Number, Currency, Scientific notation or Percentage.
Decimal places	Set the number of decimals for the selected axis.
Logarithmic scale	Set logarithmic or linear scales.
Zero line	Set the starting point at zero. For example, if you have a bar chart with a minimum value of –50, the starting point will be zero and you will have bars that go up and down, depending on their value.

The *Labels* tab: This dialog allows you to manipulate labels displayed on the axis, including Rotation, Fonts and Axis Title.

Field	Description
Selected Axis	
Orientation	Use the *Degrees* arrows to set the rotated angle for labels in the selected axis.
Show labels	Check this box to show the labels in the selected axis.
2 Levels	Check this box to show the selected axis labels on two levels.
Rotate with chart	Check this box to rotate labels with chart.
Title	Set the title for the selected axis.

The *Grid lines* tab: This dialog allows you to customize gridlines on both the major and minor intervals.

Field	Description
Selected axis	
Major gridlines	
Color:	Choose the colors for major gridlines in the selected axis.
Style:	Set the major gridline styles in the selected axis.
Weight:	Set the major gridline width in the selected axis
Minor gridlines	
Color:	Choose the colors for minor gridlines in the selected axis.
Style:	Set the minor gridline styles in the selected axis.
Weight:	Set the minor gridline width in the selected axis
Align with labels	Align labels with tickmarks and gridlines in the selected axis.
Interlaced	Set interlaced gridlines. The interlaced colors are achieved with the major grid lines color and the background of the chart.

COMPOSITE BEAMS

TABLE OF CONTENTS

Chapter 12 Composite Beams

General

Allowed Steel Shapes

I-Shape, S-Shape and WRF are allowed as composite sections.

A member must be specified as "composite" in the **Member** tab of **Member Characteristics** dialog box. Then, users must fill in the **Composite Beam** tab in the same dialog box. A concrete slab must also be created in the **Composite Slabs** spreadsheet beforehand.

Codes and Standards

The following codes are integrated into VisualDesign:

CAN/CSA-S6-00	Canadian Highway and Bridge Design Code
CAN/CSA-S16-01	CISC Handbook of Steel Construction
AASHTO-LRFD-1998	AASHTO LRFD Bridge Design Specifications (SI Units)
AISC-LRFD-1993	American Institute of Steel Construction
CAN/CSA-A23.3-95/04	CPCA Reinforced Concrete Design Handbook

Construction stages

Composite beams can be defined without construction stages (shored beams) for those not owning the Steel Design module.

For those possessing the Steel Design module, construction stages can be specified in the **Composite Beam** tab (**Project Configuration)**.

See also
The Composite Beam tab (Project Configuration)
Composite Beam tab of Member Dialog Box
Defining a Member as a Composite Beam
Composite Beam Calculation Method
Loads due to shrinkage
Temperature Gradient for Composite Beams
Steel Design Results
Number of required studs
Results - Forces and Stresses in Composite Beams
Graphic Results

Analysing Composite Beams without the Steel Design Module

For users that do not own the Steel Design module, they can model and analyse composite beams all the same. The *Composite Beam* option must be selected in the **Member** tab of **Member Characteristics** dialog box and the **Composite Beam** tab, included in the same dialog box, must be completed.

However, the **Composite Beam** tab of **Project Configuration** is not available. Consequently, construction stages cannot be activated and thus, the ratio "n", which represents the ratio E steel/E concrete will be equal to 1.0, as a default value, and will be used for the calculation of short-term deformations. Long-term deformations cannot be considered.

The Steel Design module allows specifying a ratio "n" for long-term deformations, in the **Composite Beam** tab of **Project Configuration** dialog box.

Shored Composite Beams - Steel Design

A shored composite beam (without construction stages) can be analysed considering some construction stages as well, if long-term deformations need to be studied.

Activate the **Composite Beam** tab of **Project Configuration** dialog box and use construction stages 6, 7, and/or 8. These stages correspond to effective composite effects. Then, specify a ratio "n" in the field that corresponds to long-term effects. Results will consider long-term deformations.

Calculation Method

Calculation of Positive Moment

Calculate the maximum compression that the slab can support (CconcMax) taking into account the steel reinforcement, if needed.

Calculate the maximum tension that the beam can support (TsteelMax).

If CconcMax $\geq$ TsteelMax

The neutral axis is located in the concrete slab ➔ **CASE 1** (Plastic design; the beam may be of class 3 or 4 because the whole beam is in tension.)

If CconcMax $<$ TsteelMax

The neutral axis is located in the steel shape, either in the flange or in web:

The position of the neutral axis is in the flange ➔ **CASE 2a** (Plastic design, the beam may be of class 3 or 4 because the whole web is in tension and the flange is laterally supported by the slab).

The neutral axis is located in a class 1 or 2 web ➔ **CASE 2b**.

The neutral axis is located in a class 3 or 4 web ➔ **CASE 2c**.

If the governing case is the shear force in studs ➔ **CASE 3a, 3b and 3c,** according to Case 2 criteria.

Calculation of Negative Moment

As the concrete in tension is neglected, the position of the neutral axis is always located in the web of the steel section.

If the web is of class 1 or 2 ➔ **CASE 4a**.

If the web is of class 3 or 4 ➔ **CASE 4b**.

Definition of variables:

be   = effective width of the slab

b1   = width of the top flange

b2   = width of the bottom flange

te   = effective thickness of the slab

t1   = thickness of the top flange

t2   = thickness of the bottom flange

qrr, qrs= maximum stress of a stud

Qr   = stress transmitted by the studs (Qr = Nb studs * (qrr or qrs))

Vh   = studs required shear strength to obtain a percentage of 100% of interaction

%Qr= shear connection percentage (relative to Vh, %Qr = Qr / Vh)

Ar   = total area of steel reinforcement in the slab

As   = total area of the steel beam

Φ, Φr, Φc   = according to code

Fy   = according to type of steel

f'c   = according to type of concrete

Cr   = Φr Ar Fy (constant)

CbMax = 0.85 Φc be te f'c (constant)

TaMax= Φ As Fy (constant)

COEFFICIENTS ARE:

S16-01: $\Phi = 0.90$ $\Phi c = 0.60$ $\Phi r = 0.90$ $\Phi sc = 0.80$

S6-00: $\Phi = 0.95$ $\Phi c = 0.70$ $\Phi r = 0.85$ $\Phi sc = 0.80$

CHDBC $\Phi = 0.95$ $\Phi c = 0.70$ $\Phi r = 0.90$ $\Phi sc = 0.85$

Important: Use the smallest of the following stresses:

TaMax   → The neutral axis is located in the concrete- CASE 1

CbMax + Cr → The neutral axis is located in the steel, governed by the concrete and reinforcement – CASE 2a, 2b or 2c

Qr   → The neutral axis is located in the steel, governed by the studs – CASE 3a, 3b or 3c

---

Each one of these cases is detailed in the following pages.

If minimum stress = TaMax

The neutral axis is located in the slab → CASE 1

Codes distinctive features:

S16-01:   %Qr ≥ 40%. If not, consider the steel section only.

              If the steel reinforcement is not considered → Cr = 0

S6-88:   Ok.

S6-00:   Ok.

NOTES:

This case may happen only if the shear connection percentage equals 100%.

Neglect the concrete in tension at bottom of the slab.
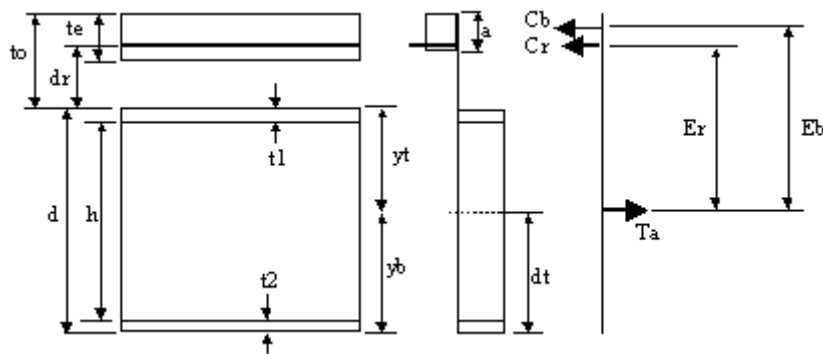
Where

Cb $= 0.85 \, \Phi c \, a \, be \, f'c$

Cr $= \Phi r \, Ar \, Fy$

Ta $= TaMax = \Phi \, As \, Fy$

a $= (Ta - Cr)/(0.85 \, \Phi c \, be \, f'c) \leq te$

dt $= yb$

Eb $= d - dt + to - (a/2)$

Er $= d - dt + dr$

and

$Mrc = Cb \, Eb + Cr \, Er$

If minimum stress = CbMax + Cr

The neutral axis is located in the steel, governed by the concrete and reinforcement →CASE 2

Cr $= \Phi r \, Ar \, Fy$

Cb $= CbMax = 0.85 \, \Phi c \, te \, be \, f'c$

Ca $= (TaMax - Cb - Cr)/2$

　　　　　(Because Ta = Cb + Cr + Ca and Ta = TaMax - Ca)

$CaMax = \Phi \, b1 \, t1 \, Fy$

Ca ≤ CaMax

The neutral axis is located in the flange of the steel section → CASE 2a

Codes distinctive features:

S16-01: %Qr ≥ 40%. If not, consider the steel section only.

If the steel reinforcement is not considered → Cr = 0

S6-88: Ok.

S6-00: %Qr = 100 %. If not, consider the steel section only.
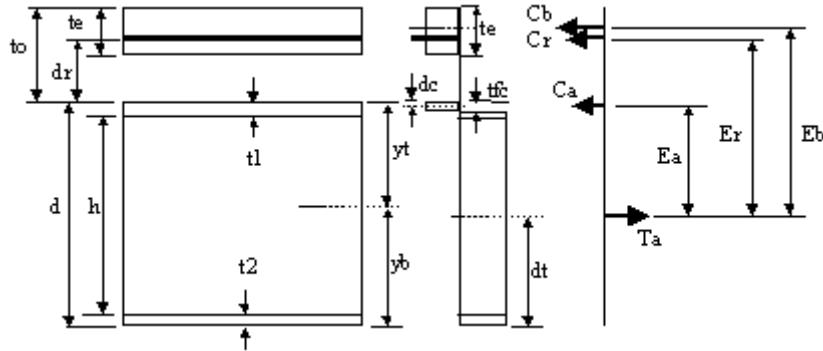
Where

yb = position of the section c.g. relative to the bottom of the section

tfc = Ca/(Φ b1 Fy)   (height of the compressed part)

dc = tfc/2   (position of c.g. of the compressed part)

Asc = tfc b1   (area of the compressed steel section)

As = total area of the steel section

dt = ((As yb) - (Asc(d - dc)))/(As - Asc)

Eb = d - dt + to - (te/2)

Er = d - dt + dr

Ea = d - dt - dc

and

Mrc= Cb Eb + Cr Er + Ca Ea

Ca > CaMax

The neutral axis is located in the web of the steel beam → CASE 2b or 2c.

Codes distinctive features:

S16-01: %Qr ≥ 40%. If not, consider the steel section only.

If the steel reinforcement is not considered → Cr = 0

S6-88: Criterion for hc/w ≤ 685/√(Fy), not for h/w ≤ 1700/√ (Fy)

S6-00:   Criterion for hc/w ≤ 850/√(Fy), not for h/w ≤ 1700/√ (Fy)

%Qr = 100 %. If not, consider the steel section only.

If (h/w) ≤ 1700/√(Fy) → Web of class 1 or 2 → CASE 2b
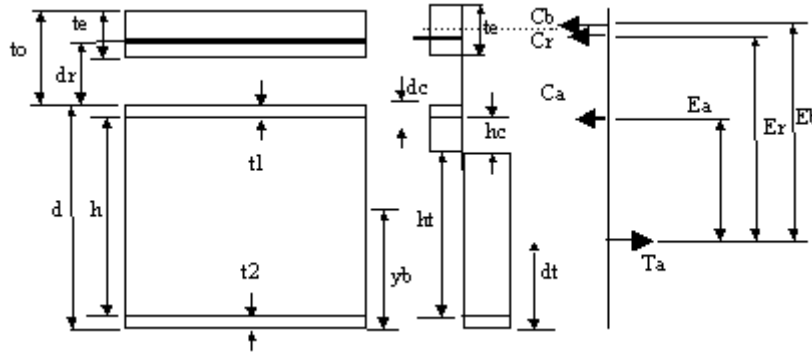
Where

Cb   = 0.85 Φc te be f'c

Cr   = Φr Ar Fy

Ca   = (TaMax - Cb - Cr)/2   (because Ta=Cb+Cr+Ca and Ta=TaMax-Ca)

hc   = (Ca - Φ b1 t1 Fy)/(Φw Fy)

Asc = (b1 t1) + (hc w)

As   = total area of the steel section

dc   = (b1 t1)(t1/2) + (hc w)(t1 + hc/2)]/Asc

dt   = (As yb) - (Asc (d-dc)))/(As - Asc)

Eb   = d - dt + to - (te/2)

Er   = d - dt + dr

Ea   = d - dt - dc

And

Mrc= Cb Eb + Cr Er + Ca Ea

If (hc/w) > 685/√(Fy) according to S6-00

or (hc/w) > 850/√(Fy) according to S6-00 → Web of class 3 or 4 → CASE 2c

Codes distinctive features:

S16-01:   This case is not covered. The code uses the steel section only.

S6-88:   Criterion for hc/w > 685/√(Fy), none for h/w > 1700/√(Fy)

          Calculation: Elastic for steel and plastic for concrete.

S6-00:   Criterion for hc/w > 850/√(Fy) , none for h/w > 1700/√(Fy)

          %Qr = 100 %. If not, consider the steel section only.

          Calculation: Plastic and removing the excess.

The following method (S6-00) is used for calculation:
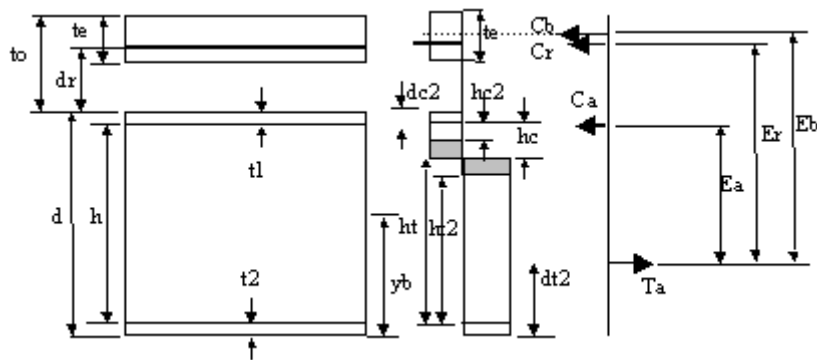
Where

Cb   = 0.85 Φc te be f'c

Cr   = Φr Ar Fy

CaT= (TaMax - Cb - Cr)/2 (because Ta=Cb+Cr+Ca and Ta=TaMax-Ca)

hc   = (CaT - Φ b1 t1 Fy)/(Φw Fy)

ht   = d - t1 - hc - t2

hc2 = 850w/√(Fy) [Code S6-00] or 685w/√(Fy) [Code S6]

hcm= hc - hc2

ht2 = ht - hcm

Asc2= (b1 t1) + (hc2 w)

Ast2= (b2 t2) + (ht2 w)

dc2 = [(b1 t1)(t1/2) + (hc2 w)(t1 + hc2/2)]/Asc2

dt2 = [(b2 t2)(t2/2) + (ht2 w)(t2 + ht2/2)]/Ast2

Ca   = Φ Asc2 Fy

Eb = d - dt2 + to - (te/2)

Er = d - dt2 + dr

Ea = d - dt2 - dc2

And

Mrc= Cb Eb + Cr Er + Ca Ea

If Minimum stress = Qr

The neutral axis is located in the steel section; governed by concrete and steel reinforcement → CASE 3

Qr = relating to the shear connection

Ca = (TaMax - Qr)/2 (because Ta = Qr + Ca and Ta = TaMax - Ca)

CaMax   = Φ b1 t1 Fy

Ca ≤ CaMax

The neutral axis is located in the web of the steel section → CASE 3a

Codes distinctive features:

S16-01:   %Qr ≥ 40 %. If not, consider the steel section only.

Qr acting at a/2, not at te/2.

a = thickness of the concrete transmitting Qr.

S6-88:   Ok.

S6-00:   This case is not covered.

%Qr shall be equal to 100%, steel section only.
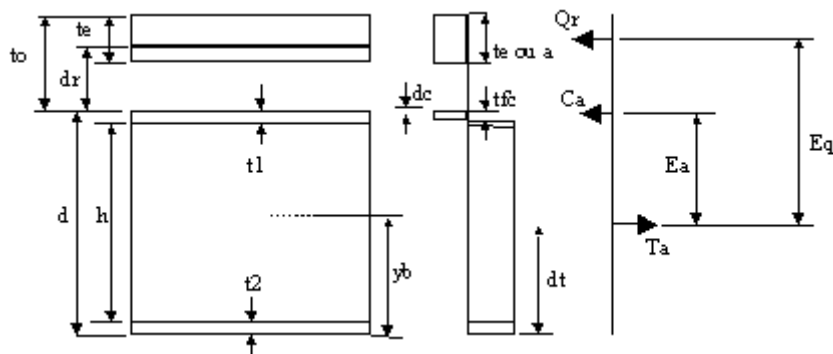
Where

tfc = Ca/(Φb1 Fy)   (thickness of the compressed part)

---

dc   = tfc/2   (position of c.g. of the compressed part)

Asc = tfc b1   (area of the compressed steel section)

As   = total area of the steel section

a   = Qr/(0.85 Φc be f'c)

dt   = ((As yb) - (Asc (d-dc)))/(As - Asc)

Eq   = d - dt + to - (te/2) [Code S6]   or d - dt + to - (a/2) [Code S16]

Ea   = d - dt - dc

and

Mrc= Qr Eq + Ca Ea

Ca > CaMax

The neutral axis is located in the web of the steel section → CASE 3b or 3c

If (h/w) ≤ 1700/√(Fy) according to S16.1

or (hc/w) ≤ 685/√(Fy) according to S6 → Web of class 1 or 2 → CASE 3b

Codes distinctive features:

S16-01:   %Qr ≥ 40 %. If not, consider the steel section only.

              Qr acting at a/2 instead of te/2

              a = thickness of concrete transmitting Qr.

S6-88:   Criterion for hc/w ≤ 685/√(Fy) or 905/√(Fy)

S6-00:   This case in not covered.

              %Qr shall be equal to 100. If not, consider the steel section only.
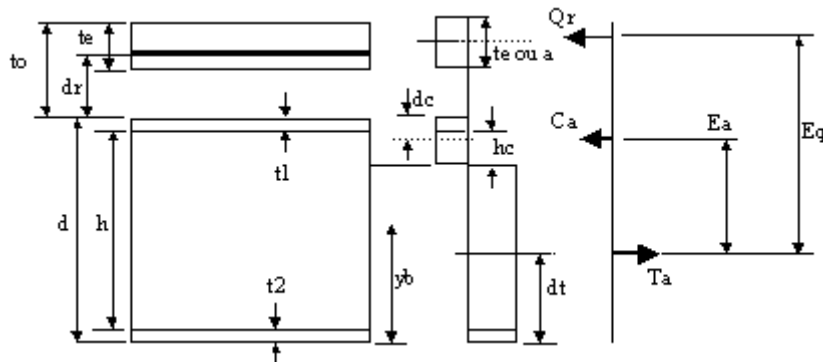
Where

Ca   = (TaMax - Qr)/2   (because Ta=Qr+Ca and Ta=TaMax-Ca)

hc   = (Ca - Φ b1 t1 Fy)/(ΦwFy)

Asc = (b1 t1) + (hc w)

As   = total area of the steel section

a   = Qr/(0.85 Φc be f'c)   (only for S16)

dc   = (b1 t1²/2) + (hc w)(t1 + hc/2)]/Acs

dt   = ((As yt) - (Asc (d-dc))/(As - Asc)

Eq   = d - dt + to - (te/2) [Code S6]   or d - dt + to - (a/2) [Code S16]

Ea   = d - dt - dc

And

Mrc= Qr Eq + Ca Ea

If (h/w) > 1700/$\sqrt{}$(Fy) [Code S16]

**or (hc/w) > 685/$\sqrt{}$(Fy) [Code S6] $\rightarrow$ Web of class 3 or 4 $\rightarrow$ CASE 3c**

Codes distinctive features:

S16-01:   This case is not covered; steel section only.

S6-88:   Criterion for hc/w > 685/$\sqrt{}$(Fy) or 905/$\sqrt{}$(Fy)

              Calculation: elastic for steel section and plastic for concrete

S6-00:   This case is not covered.

              %Qr shall be equal to 100%; steel section only.

The following method (S6-00) is used for calculation:
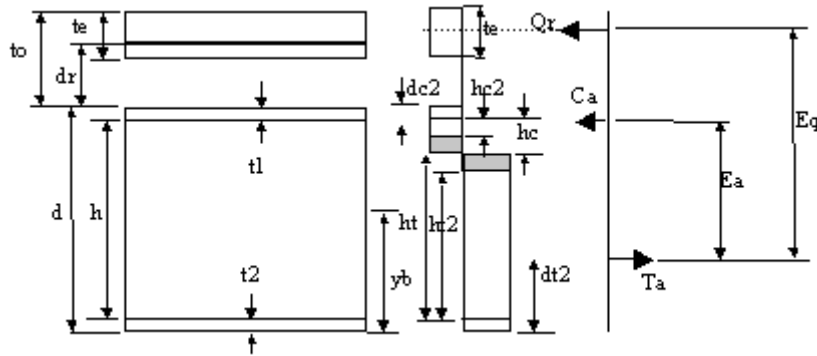
Where

CaT= (TaMax - Qr)/2   (because Ta=Cb+Cr+Ca and Ta=TaMax-Ca)

hc   = (CaT - Φ b1 t1 Fy)/(Φw Fy)

ht   = d - t1 - hc - t2

hc2 = 850w/$\sqrt{}$ (Fy) [Code S6-00], or 685w/$\sqrt{}$(Fy) [Code S6]

hcm= hc - hc2

ht2 = ht - hcm

Asc2= (b1 t1) + (hc2 w)

Ast2= (b2 t2) + (ht2 w)

dc2 = (b1 t1)(t1/2) + (hc2 w)(t1 + hc2/2)]/Asc2

dt2 = (b2 t2)(t2/2) + (ht2 w)(t2 + ht2/2)]/Ast2

Ca   = ΦAsc2 Fy

Eb   = d - dt2 + to - (te/2)

Er   = d - dt2 + dr

Ea   = d - dt2 - dc2

and

Mrc= Cb Eb + Cr Er + Ca Ea

Behaviour while in negative moment →CASE 4

If (hc/w) ≤ 685/$\sqrt{}$(Fy) [Code S6]

or (hc/w) ≤ 850/$\sqrt{}$ (Fy) [Code S6-00] → Web of class 1 or 2 → CASE 4a

Codes distinctive features:

S16-01:   This case is not covered; steel section only.

S6-88:   %Qr shall be equal to 100%. If not, consider the steel section only.

Criterion for hc/w ≤ 685/√(Fy)

S6-00:   %Qr shall be equal to 100%. If not, consider the steel section only.

hc/w ≤ 850/√(Fy)

The user must verify the stability of the bottom flange.
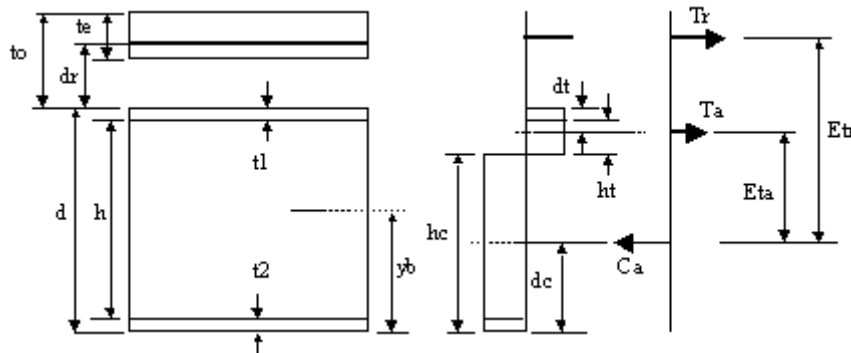
Where

Tr   = min(Φr Ar Fy, Qr)

Ta   = (TaMax - Tr) / 2   (because Ca = Ta + Tr and Ca = TaMax - Ta)

ht   = (Ta - Φ b1 t1 Fy)/(Φw Fy)

hc   = d - t1 - t2 - ht

Asc = (b1 t1) + (ht w)

As   = (b1 t1) + (h w) + (b2 t2)

dt   = ((b1 t1)(t1/2) + (ht w)(t1 + ht/2))/Asc

dc   = ((As yb) - (Asc (d-dt))/(As - Asc)

Etr = d - dc + dr

Eta = d - dc - dt

and

Mrc= Tr Etr + Ta Eta

If (hc/w) > 685/√(Fy) [Code S6]

or (hc/w) > 850/√ (Fy) [Code S6-00] → **Web of class 3 or 4** → **CASE 4b**

Codes distinctive features:

S16-01:   This case is not covered; steel section only.

S6-88:   This case is not covered; steel section only.

S6-00:   Class 3: elastic for steel; hc/w > 850/√(Fy),

Class 4: Flanges of class 1, 2 or 3, web of class 4 with transverse support.

          If b1 = b2 and (h/w) > 3150/√(Fy) OR

          If b1 <> b2 and (hc/w) > 1575/√(Fy) → Use the steel section only.

          Else, if (2 hc)/w > 1900/√(Fy)

          →Use Mr = Mr * [1.0 - (1/(300 + (1200 Acf/Aw))) (2 dc)]

                              ref.[10.10.4.3]

          Else, Mr = Mr.
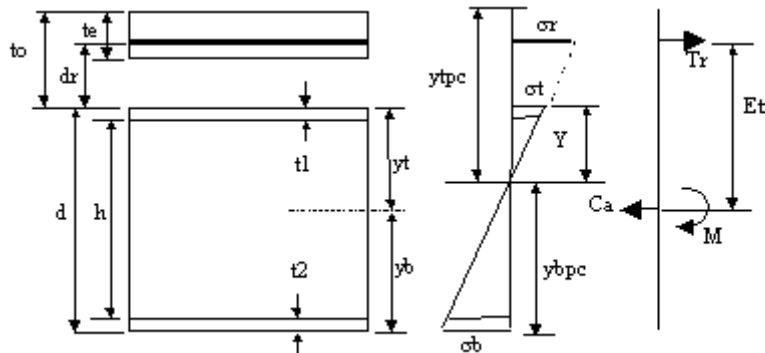
Where

ybpc= ((As yb) + (Ar (d+dr)))/(As + Ar)

Y   = d - ybpc

Can be limited by:

σr = Φr Fyr   (case 1) or

σt = Φ Fy   (case 2) or

σb = Φ Fy   (case 3)

Case 1:   σr = Φr Fyr

$$\sigma t = \Phi r \ Fyr \ (Y/ytpc)$$

$$\sigma b = \Phi r \ Fyr \ (ybpc/ytpc)$$

If ($\sigma t \leq \Phi$ Fy et $\sigma b \leq \Phi$ Fy) ➔ Case 1 is satisfying. If not, try case 2.

Case 2: $\sigma r = \Phi$ Fy (ytpc/Y)

$$\sigma t = \Phi \ Fy$$

$$\sigma b = \Phi \ Fy \ (ybpc/Y)$$

If ($\sigma r \leq \Phi r$ Fyr et $\sigma b \leq \Phi$ Fy) ➔ Case 2 is satisfying. If not, use case 3.

Case 3: $\sigma r = \Phi$ Fy (ytpc/ybpc)

$$\sigma t = \Phi \ Fy \ (Y/ybpc)$$

$$\sigma b = \Phi \ Fy$$

Tr   = Ar σr

Ca   = Tr   (sum of forces equals 0)

Etr   = yt + dr

M   = (σb - (Ar σr)/As) (Is/ybpc)   (because σb= (Ca/As) - (M ybpc/Is))

Mr   = Tr Etr + M

Project Configuration

Composite Beam tab

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

Activate the "Project with steel/concrete composite beams" box in the upper part of the dialog box in order to activate construction stages.

You need to define construction stages if you need to:

- Design or verify composite beams that are <u>not shored</u>;

- Consider the casting sequences for concrete slabs (rectangular and triangular plates);

- Obtain accurate deflections for long-term deformations considering the ratio "n".

Results will depend on these construction stages.

CivilDesign Inc. Engineering Software

Ratio "n" (E steel/E concrete)

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

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

Ratio of modulus E and G

Non-effective stiffness / E original stiffness: This ratio represents the difference between the stiffness of liquid and solid concrete for the analysis of construction stages.

Bridge Design – Fatigue in Studs

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<12m
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

Longitudinal Members	Span Length, L= 12 m	Span Length, L<12m
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

Modeling Composite Members

Create a Composite Slab

Go to **Structure / Composite Slabs** and insert a line. Give a name or number to this slab and fill in parameters. It is important to accurately define rebars steel grade and concrete material.

SLAB WITHOUT STEEL DECK

For a flat slab without a steel deck, choose "nil" for the deck.

SLAB DIRECTION

The slab direction represents the orientation of main reinforcement (ribs) with respect to the member direction.

SLAB REINFORCEMENT

Two layers of reinforcing bars are considered: one at the top and the second at the bottom of the slab. They are perpendicular to the member.

The Composite Beam tab

If you specified a composite beam in the "Composition" field of **Member** dialog box**,** you must select this tab in order to complete the following parameters:
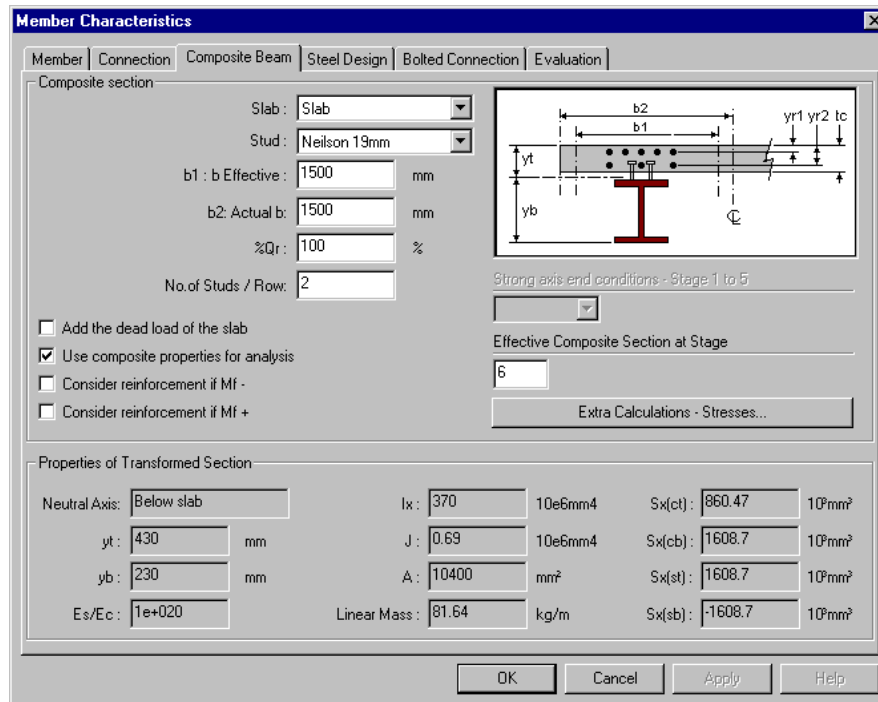
Field	Description
Composite Section	
Slab	Choose a type of slab previously defined in the Composite Slabs spreadsheet.
Stud	Choose a type of stud previously defined in the Studs spreadsheet.
Effective b (b1).	Effective width of the composite beam
Actual b (b2).	Actual width of the composite beam
%Qr	Fraction of full shear connection (ex: 100% , 70%, 40%)
Nos. Studs/Row	Specify the number of studs per row to consider fatigue in studs for a bridge design.
Linear mass	Linear mass calculated by the program
Add slab dead load	Check this box to add the slab dead load to the dead load of the structure.
Composite Properties	Check this box to consider composite section properties for analysis.
With Reinf. If Mfx -	Check this box to consider the steel reinforcement in the slab located at negative moment. As a result, the position of neutral axis will be modified.
With Reinf. If Mfx +	Check this box to consider the steel reinforcement in the slab located at positive moment. As a result, the position of neutral axis will be modified.
End Conditions on strong axis for stages 1 to 5	Prestressed concrete only: Select beam end conditions relative to strong axis during construction stages 1 to 5. VisualDesign will use the appropriate stiffness matrix when transferring to a statically indeterminate structure.
Composite section is active at Stage:	Steel/concrete composite beam: If you completed the **Composite Beam** tab of **Project Configuration**, specify the construction stage where the composite section should be effective. VisualDesign will use the transformed properties.
Extra Calculations - Stresses	Press this button to activate a dialog box allowing you to calculate stresses located at four points on the steel section. Refer to topic Extra Calculations
Transformed Section Properties	Shaded fields indicate the transformed section properties. Refer to the image below to locate the neutral axis.

Field	Description
Neutral Axis	Position of neutral axis in the composite section.
yt	Distance from neutral axis to top fibre.
yb	Distance from neutral axis to bottom fibre.
Es/Ec	Ratio n: Modulus of elasticity of steel divided by modulus of elasticity of concrete
Ix	Inertia of transformed section, on strong axis.
J	Torsional constant of the transformed section.
Area	Area of the transformed section.
Linear Mass	Linear mass of transformed section.
Sx (ct)	Elastic section modulus of the top of concrete slab, for strong axis.
Sx (cb)	Elastic section modulus of the bottom of concrete slab, for strong axis.
Sx (st)	Elastic section modulus at the top of steel shape, for strong axis.
Sx (sb)	Elastic section modulus at the bottom of steel shape, for strong axis.

See also

Composite Beam tab (Project Configuration)
Composite Concrete Slab
Properties of the Section
Defining Members as Composite
Slabs spreadsheet
Studs spreadsheet
Composite Prestressed Concrete Beam
Calculation of Forces and Sx for Composite Beams

Definition of Parameters

Effective b

This parameter represents the slab effective width that is considered in the strength calculations. This value is calculated in accordance with the different codes and with respect to the spacing of beams. The user does the calculation.

Actual b

This parameter represents the slab real width for calculating dead load of the slab. It is required if option *Add dead load of slab* is activated. However, the user must enter a uniform load representing the slab dead load.

Shear connectors

"%Qr" is the shear connection percentage. It shall have a value between 25% and 100%. A section having a percentage less than 25% cannot be considered as composite. The percentage represents the actual number of studs over the optimum number of studs for this type of beam. It is recommended to use 100% at all time. Do not forget that S6-00 does not allow a %Qr less than 100%. If less than 100% is used, only the strength of the steel section alone will be considered.

Options:

Add the slab dead load: You can activate this box if there no construction stages are defined.

Use composite properties for analysis: If you check this box, composite effects will be considered for analysis. The calculation of composite inertia, Ix, will be done this way:

According to the Canadian Code:

$Ie = Is + 0.85(\%qr/100)*0.25 (It - Is)$ (See the code S16-01 at section [17.3.1] for more details.)

According to the American Code:

$Ix = Is + sqrt(\%Qr) * (It - Is)$

Note. For a concrete-concrete composite section (prestress concrete), the standard transformed inertia is used.

Consider Rebars if Mf +: Check this box to consider the slab reinforcement for the calculation of the composite beam positive bending strength. Consequently, the neutral axis will be adjusted and so will be the section-transformed properties.

Consider Rebars if Mf -: Check this box to consider the slab reinforcement for the calculation of the composite beam negative bending strength. Consequently, the neutral axis will be adjusted and so will be the section-transformed properties. If you activate this option, we recommend putting studs to carry the shear forces.

The program will not consider the member as composite if no slab has been specified.

The Filled HSS tab

If you specified a filled HSS in the "Composition" field of **Member** dialog box**,** you must select this tab in order to complete the following parameters:
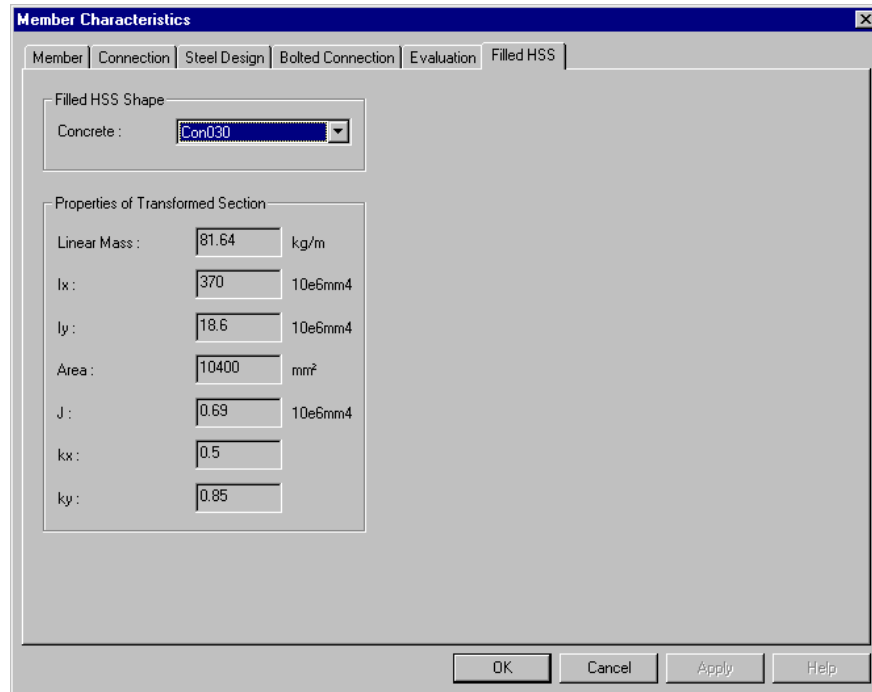
Field	Description
Filled HSS Shape	
Concrete	Choose the concrete material that fills the section.
Transformed Section Properties	Linear Mass, Ix, Iy, Area and J correspond to the properties of the transformed section. kx, ky parameters are used in the calculation of shear energy.

Procedure (with the Steel Design module)

To define members as composite sections, do as follow:

- If the project is done with respect to construction stages, go to **Project Configuration** and select the **Composite Beam** tab. Then, activate the *Steel/Concrete Composite Beam Project* check box and select appropriate construction stages listed below.

   If composite beams are shored up (with no construction stages), do not activate the *Composite Beam Project* check box.

- Go to the **Common** menu and consult data for studs, rebar steel grades, steel decks (if required) and concrete material in the appropriate spreadsheets.

- Select the **Slabs** spreadsheet under **Structure** menu and create a slab.

- Activate the Member icon on the Elements toolbar and select members that you wish to define as composite.

- Click the "Properties" icon to open the **Member Characteristics** dialog box.

- Open the Shape Selection tree by pressing icon and choose a steel shape. Specify a steel material in the "Material" drop-down list box.

- Select option "Composite beam" in the "Composition" field.

- Then, select the **Composite Beam** tab. Select the slab, stud and other parameters. Don't forget to specify the construction stage at which composite effects will be activated.

- It is recommended to align composite members at the "top", through the **Connection** tab (**Member Characteristics** dialog box), to avoid developing inappropriate bending moments that can be induced created by eccentricities.

- Define construction stage loads in the **Loads Definition** spreadsheet, if the composite beam is not shored.

- Define construction stage load combinations in the **Load Combinations** spreadsheet, if the composite beam is not shored. Enter the construction stage number for each construction stage load combination. Create other "usual" load combinations with the Wizard.

- Run a non-linear static analysis if you do not own the Steel Design module.

- If you possess the Steel Design module, go to the **Steel Design** tab, select a steel specification and fill in design parameters and criteria.

- Launch a steel design.

Procedure (Static analysis)

- Go to the **Common** menu and consult data for studs, rebar steel grades, steel decks (if required) and concrete material in the appropriate spreadsheets.

- Select the **Slabs** spreadsheet under **Structure** menu and create a slab.

- Activate the Member icon on the Elements toolbar and select members that you wish to define as composite.

- Click the "Properties" icon to open the **Member Characteristics** dialog box.

- In the **Member** tab, select a steel shape (I, S, and WRF) and specify a steel material in the "Material" drop-down list box. Select option "Composite beam" in the "Composition" field.

- Then, select the **Composite Beam** tab. Select the slab, stud and other parameters. Don't forget to specify the construction stage at which composite effects will be activated.

- It is recommended to align composite members at the "top", through the **Connection** tab (**Member Characteristics** dialog box), to avoid developing inappropriate bending moments that can be induced created by eccentricities.

Members Spreadsheets

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
Slab	Choose a slab (previously defined in the 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
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

Column	Description	Editing
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
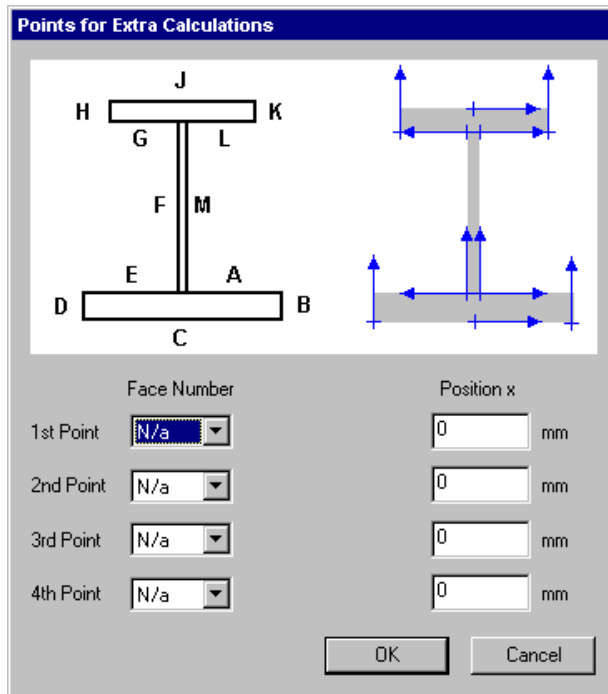
See also

Composite Beam tab (Project Configuration)
Members Spreadsheet
Defining Members as Composite Sections
Composite Prestressed Concrete Beam
Calculation of Forces and Sx for M+ and M-

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 that possess 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

Filled HSS 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
Infilling Material	Choose the type of concrete filling the HSS.	Double-click
Linear Mass	Linear mass of the transformed section.	No
Ix	Inertia of transformed section, for strong axis.	No
Iy	Inertia of transformed section, for weak axis.	No
Area	Area of transformed section.	No
J	Torsional constant of transformed section	No
kx	Parameter used in the calculation of deflection considering shear energy, for strong axis.	No
ky	Parameter used in the calculation of deflection considering shear energy, for weak axis.	No

See also
Deflection including shear energy

Loads & Load Combinations

Thermal Gradient and Shrinkage Effects

A thermal gradient can be applied to a composite slab. It will be applied to the top and bottom of the slab.

PROCEDURE:

Select **Loads** / **Load Cases** / **Members** / **Temperature**. Enter the temperature variation at the top and bottom of the concrete slab. Then, select option *Thermal Gradient* in column "Type of Application".

VisualDesign's approach

VisualDesign does the following steps for considering thermal gradient and shrinkage effects in steel/concrete composite beams:

Step 1:

Deflections σw and θx are calculated with the transformed properties of the composite beam without considering any sliding effects (theoretical composite beam).
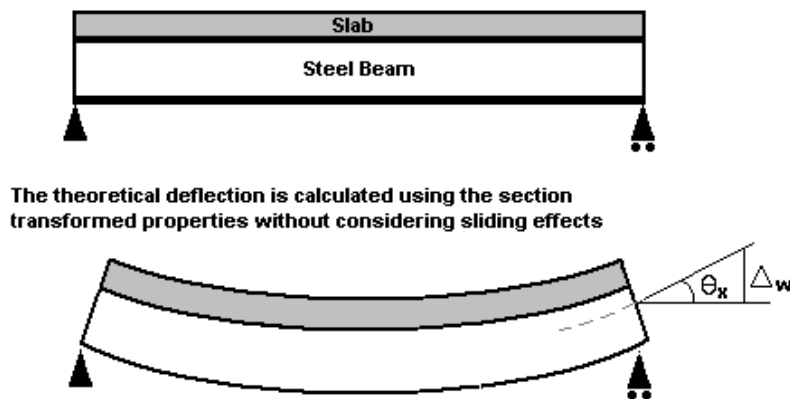
Step 2:

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.

VisualDesign applies the theoretical deflection to the steel beam only and calculates the corresponding stresses
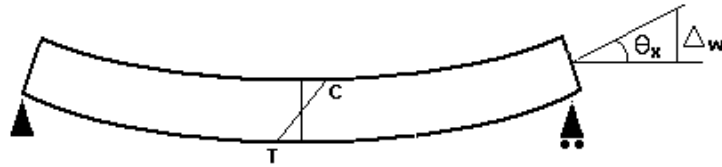
Then, equivalent forces are applied to the composite beam and sliding effects are considered
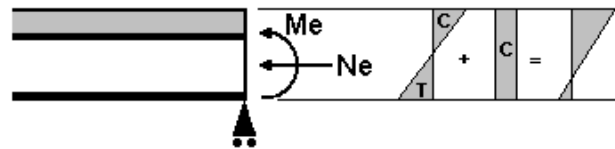
Balanced Effects

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**).

Corrections are made to stresses in the sections due to balanced effects

$$\sigma = \varepsilon_r * E_b$$
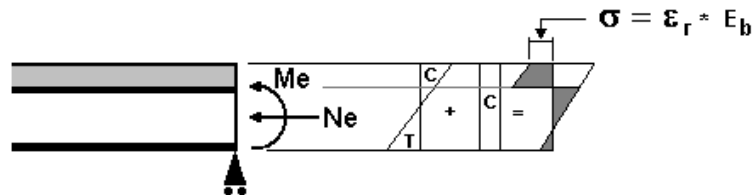
See also
Defining a composite section
Temperature Variations on Members

Load due to Shrinkage

Select this spreadsheet (**Loads** / **Load Cases** / **Members**) and enter the deformation due to shrinkage. It 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.

Group: Load case: (title)

Column	Description	Editing
ID	Automatically calculated	No
Shrinkage 10E-6	Deformation due to shrinkage. It is applied to the concrete slab of composite sections.	Single click

See also
Steel-concrete Composite Beam

Composite Beam with Construction Stages

Load titles and types

Define load titles and types in the **Load Definition** spreadsheet. Each construction stage load case must be defined in the spreadsheet (Slab dead load, additional dead load, bitumen, formwork, etc.). The software automatically cumulates these load cases during the analysis of construction stages.

Load Combinations

Each construction stage must be defined as a load combination and must include the load case that is corresponding to this stage only. These load combinations must have a *Construction Stage* status to be allowed to look at graphic results.

Load Combination Status

Load combinations status must be specified as *Construction Stage, Deflection* or *Fatigue* to be allowed to look at graphic results for composite beams. Refer to Chapter 4 for more details.

Automatic Generation of Load Combinations

When construction stage load combinations are completed, use the **Load Combination Generator** to generate usual load combinations, as per selected code. Do not forget to activate the option "Add generated load combinations to the existing ones" in the **General Options** page of the **Generator**.

Usual load combinations are applied to the structure at the end of construction stages and composite effects are considered in the analysis.

Composite Beam without Construction Stages

Load titles and types

Define load titles and types, as usual, in the **Load Definition** spreadsheet. It is not necessary to differentiate dead load cases.

Load Combinations

Define usual load combinations.

Analysis

Extra Calculations - Composite Beams

You can get up to four stress calculations for the selected composite beam. The procedure is as follows:

- Activate the Structure mode and double-click on a composite beam.

- Go to the **Composite Beam** tab of **Member Characteristics** dialog box and press the "Extra Calculations…" button.

- In the dialog box, you can define up to 4 points on the steel beam with respect to the direction of blue arrows.
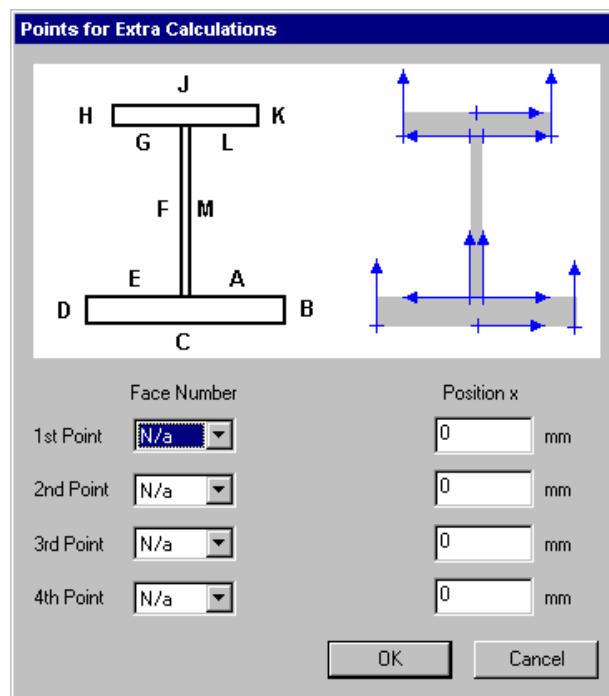
- Click OK.

- Launch a design.

- Once that the design is completed, look at results by selecting **Results / Load Combinations / Composite Beams – Stresses**. In the dialog box, select the appropriate graph.

See also
Graphs – Stresses in Composite Beams

CivilDesign Inc. Engineering Software

Calculation of Forces and Sx for M+ and M-

VisualDesign calculates inflexion points near positive and negative bending moments for continuous beams.

These inflexion points must be calculated at serviceability limits states with permanent loads acting on the steel section only combined to additional permanent loads acting on the composite section (equivalent factor of 3n). The definition of ratio n = E steel/E concrete in located in the **Composite Beam** tab of **Project Configuration** dialog box.

We recommend that you split continuous beams at inflexion points (approximately) so that you can activate specific options for the design near positive and negative bending moments. Options are included in the **Composite Beam** tab of **Member Characteristics** dialog box.

Negative Bending Moments

The calculation of resistance at negative bending moments must be done considering the steel section only, for loads that are applied once that the composite effects are effective.

To do so, do not activate any option in the **Composite Beam** tab of **Member Characteristics** dialog box.

[_] Add the dead load of slab

[_] Use the transformed properties for analysis

[_] Use the slab reinforcement when Mfx >0.0

[_] Use the slab reinforcement when Mfx<0.0

The calculation of strength, fatigue and permanent stresses for a composite beam at negative bending moments will be done considering the steel shape only and the minimum envelope. Calculation of stresses is done using values of transformed Sx, as explain below.

Positive Bending Moments

The calculation of resistance at positive bending moments must be done considering the composite section, for loads that are applied once that the composite effects are effective.

To do so, activate the second option in the **Composite Beam** tab of **Member Characteristics** dialog box.

[_] Add the dead load of slab

[x] Use the transformed properties for analysis

[_] Use the slab reinforcement when Mfx >0.0

[_] Use the slab reinforcement when Mfx<0.0

The calculation of strength, fatigue and permanent stresses for a composite beam at positive bending moments will be done with the composite section and maximum envelope.

Value of Iy

VisualDesign always uses the value of Iy transformed, except before the casting of concrete.

Calculation of Sx

The evaluation of section modulus Sx at the top and bottom of the slab is based on *Picard & Beaulieu* theory, which considers the value of Sx at the bottom of the steel section as follows:

Se = Ss + 0.85(alpha)^0.25 (St-Ss)

In VisualDesign, we use this principle for all values of Sx:

Sti = It / yti

Ssi = Is / ysi

frQr = 0.85f * Qr^0.25;

Ie = Is + frQr * (It - Is);

Sei = Ssi + frQr * (Sti - Ssi)
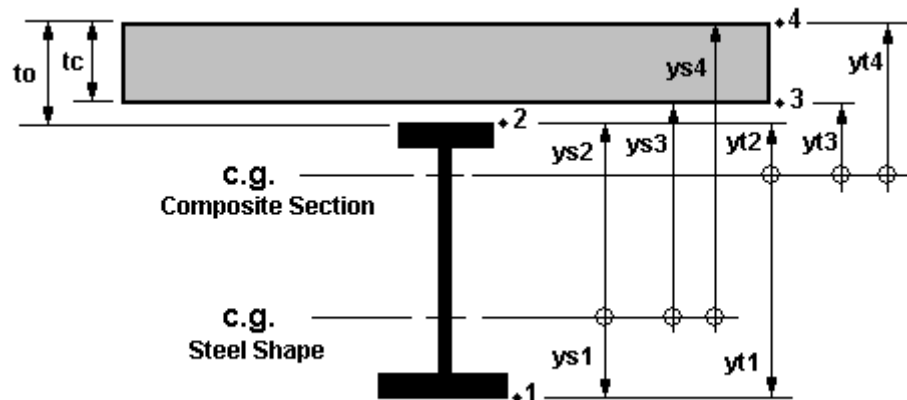
Stresses in the slab:

For positive bending moments in the slab, we use Se3 and Se4 values.

For negative bending moments in the slab, no stress is calculated.

The calculation of required number of studs for negative bending moment will take into account the reinforcement in the slab if the user activated the following option included the **Composite Beam** tab of **Member Characteristics** dialog box:

[x] Use the slab reinforcement when Mfx<0.0

See also
The Composite Beam tab - Members
Calculation Method for Composite Beams
Number of Required Studs

Calculation of Deflection for Composite Beams

Without the Steel Design Module

Deflections will be calculated using a ratio "n" equal to 1.0 and 3.0, respectively corresponding to short-term and long-term deformations.

With the Steel Design Module

A design of composite beams can be carried on according to construction stages defined in the **Composite Beam** tab of **Project Configuration**. Ratios "n" can be specified also in this tab and the transformed properties of sections will be considered for the calculation of short-term and long-term deformations

See also
The Composite Beam tab (Project Configuration)
Composite Beams Spreadsheets – Short-term and Long-term
Calculation of Long-term Deflection for a Shored Composite Beam

Composite Beams Results

Interpretation

There are two types of results for composite beams depending on the analysis with or without construction stages. Read carefully the following:

Composite beam without construction stages:

The *Internal stresses in members* spreadsheet (**Results / Load Combination** and **Results / Envelope**) includes stresses in the steel section only. It does not include stresses at the top and bottom of the slab. To get these results, select graphical results (**Results / Load Combination / Stresses in Composite beams).**

To obtain stresses in the slab AND in the steel shape, create *Service* load combinations or/and *Fatigue* load combinations and go to graphical results (**Results / Load Combination / Stresses in Composite beams)** when analysis will be completed.

Composite beam with construction stages:

Only graphical results are valid. (**Results / Load Combination / Stresses in Composite beams).**

N.B. Results included in the *Internal stresses in members* spreadsheet are not valid.

Steel Design Results

Consult this spreadsheet to know the number of required studs (**Results / Structure Design / Steel**).

See also
Internal stresses in members (min/max)
Stresses in Composite Beam
Number of required studs (Steel design results)

Design Results at Construction Stages

The design brief for steel/concrete composite beams can be obtained for load combinations that correspond to construction stages.

To do so, go to the **Load Combinations** spreadsheet and duplicate construction stage load combinations. Modify their statuses to *Ultimate* or *Service* (any status other than *Construction Stage*), but keep the construction stage number.

When the steel design will be completed, the Design Results activation mode will be on. You will notice that the word *Standard* will appear in the Title Selection box on Activation toolbar. Select a load combination among the list box. Select a composite beam and press the **Properties** icon to open the **Steel Design Results** spreadsheet.

Consult the Design Brief or print it.

See also

Interpreting Composite Beams Results

Load Combinations for Composite Beams

The Composite Beam tab (Project Configuration)

Calculation of Deflections for Composite Beams (General)

Composite Beams Spreadsheets – Short-term and Long-term

Number of Required Studs
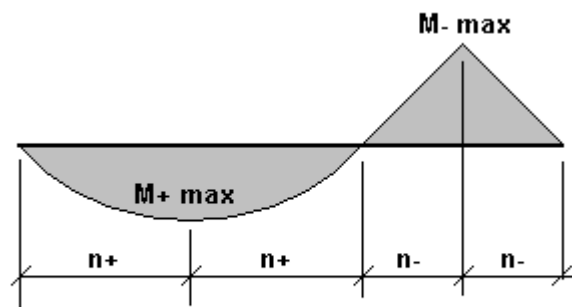
Number of Required Studs

Inside the **Steel Design Results** spreadsheet, you will find the required number of studs to transfer the shear force. According to codes, the number of studs shall be between M+ max and M=0 (n+ on diagram) and between M-max and M=0 (n- on diagram).

You will find this information in the **Steel Design Results** spreadsheet, namely, Studs (0:M+) and Studs (0:M-).

See also

Members Spreadsheet

The Member Dialog box

Defining a Member as a Composite Beam

Composite Beam Calculation Method

Steel Design Results Spreadsheet

Forces and Stresses in Composite Beams

Stresses in Composite Beams

This function allows displaying results in the form of graphs, for serviceability load combinations with or without construction stages.

Select a composite beam and go to **Results / Load Combination / Stresses – Composite Beams**.

Available graphs are:

- Stresses in composite beam (1);

- Stresses in composite beam (Other points) (2);

- Bending moments and resistance;

- Shear forces and resistance;

- Deflection;

- Stresses due to fatigue (3);

- Stresses due to fatigue (Other points) (2) (3);

- Spacing of Studs (4).

Note 1:

The following stresses will be displayed:

- Sigma_Ds: Stress at the top of slab;

- Sigma_Di: Stress at the bottom of slab;

- Sigma_Ss: Stress at the top of steel section;

- Sigma_Si: Stress at the bottom of steel section;

Note 2:

Stresses will be displayed for other points on the steel shape, as it was specified in the **Composite Beam** tab of **Member** dialog box, using the **Extra Calculation** tool. Refer to Extra Calculation.

Note 3:

VisualDesign can display such a graph if there is at least one *Fatigue* load combination that includes one results envelope. If there is no envelope, VisualDesign needs two *Fatigue* load combinations. This graph represents a variation of stresses.

Note 4:

This graph is available for a design using moving load analysis.

Diagram Values:

Double-click in the legend box to move it outside the diagram box.

On a diagram, place your cursor on a point line and coordinates will be displayed near your cursor.

Use the **Graphs** toolbar functions that are posted in every diagram box. To learn about these functions, see the topic Graphs toolbar.

Procedure:

- Select a composite member;

- Go to **Results / Load Combinations/ Stresses – Composite beams**.

- The **Graphs** dialog box will be displayed on screen.

  - Select a graph in the list box;

  - Select a load combination;

  - According to the selected graph, check or uncheck the boxes in the activated fields of the dialog box;

  - Double-click on the legend to move it out of the displayed graph;

  - Look at graphic results. Point the cursor on a maximum and the value (coordinates) will be displayed.

  - Print the graph using the **Graphs** toolbar.

See also
Graphs Toolbar (ChartFX)
Steel Design Results Spreadsheet

Graphs Toolbar (ChartFX)

When activating graphical results of prestressed concrete analysis (**Results/Graphs** menu), a toolbar is provided in all the results diagrams to help you managing diagrams (copy, print, view options, etc.).

You will find below, a description of functions represented by icons and also a description of dialog boxes that can be called up by pressing some icons.

Copy to clipboard:

Click on this icon to copy the diagram as a Bitmap, as a Metafile, as Text (data only) or as OLE Object.

Print Preview:

Click on this icon to call up the *Page setup* dialog box. Complete the dialog box and look at the diagram. The table below explains the parameters included in the dialog box.

Field	Description
Options	
Separate Legend	Check this box to move the legend outside the diagram box.
Use Printer Resolution	Check this box to use the printer resolution.
Force Colors	If your printer is black and white, the printer driver will match the chart colors to a specific grayscale pattern.
Margins (inches)	Define the left, top, right and bottom margins, in inches.
Orientation	
Portrait	Check this box to look at the diagram in the vertical way.
Landscape	Check this box to look at the diagram in the horizontal way.
Use by default	Check this box to use the above parameters by default.

Print:

Press this icon to print the diagram. In the *Layout* tab, choose the Landscape orientation. This option must be selected even if you activated the Landscape orientation in the *Page Setup* dialog box (Print Preview function).

Zoom:

Click on this icon and, with your cursor, draw a window around the elements that you wish to zoom in. The icon will still be activated to let you zoom again. To go back at a normal view, click on the icon again.

Horizontal grid:

Click on this icon to display a horizontal grid.

Vertical grid:

Click on this icon to display a vertical grid.

Series Legend:

Click on this icon to mask or display the series legend of the diagram.

Properties:

Click on this icon to display a vertical grid.

Click on this icon to modify the diagrams view options through the **Chart FX 98 Properties** dialog box. It is composed of the *General*, *Series*, *Axes* and *3D* tabs. Look at the tables below to learn more about each tab.

The *General* tab: This dialog box allows you to set general settings in the chart, like Major Unit Gridlines and Colors for the chart.

Field	Description
Effects	
3D	Check this box to switch from 2D/3D views.
Cluster (Z-Axis)	When displaying a clustered chart, each series will have its own position in the z-axis. This means, if you have a 3 series chart, each data series will occupy one row of data and there will be 3 rows (z-axis clusters) in the chart.
Stacked	This function is not activated.
Axes styles	This option allows you to change the axis drawing style.
Grid lines	This option allows you to draw vertical and/or horizontal gridlines.
Colors	
Palette	This option allows you to change the entire chart palette. This will affect all elements in the chart.
Scheme	This option allows you to change the color scheme for markers.
Background	This option allows you to set a color for the background. This background is the box where the chart is enclosed.
Chart box	This option allows you to set a color for the chart box. This is the background where the markers are enclosed.
Title	This option allows you to set a Top Title for the chart.

The *Series* tab: This dialog box allows you to set specific settings, including visual attributes for series in the chart.

Field	Description
Combo Box	When "All series" is selected in the Combo Box, settings will apply to all series in the chart and the property set is different if a specific series is selected. Therefore, 2 different screen shots are included for this particular dialog.
Marker fill	This option allows the user to change colors for series.
Lines	
Same color as markers	This option is activated if a specific series is selected. It will match the color of lines to the one used for markers.
Custom	This option is activated if a specific series is selected. Change the line color, style and weight for this specific series. Look at the sample that is displayed in the dialog.
Color	Choose a line color for this specific series.
Style	Choose a line style for this specific series.
Weight	Choose a line weight for this specific series.
Markers	Markers represent points on the series.
Gallery:	This option allows you to change the chart type
Shape:	Choose a marker shape such as cylinders or cones for a *bar chart* or *cube chart*.
Show every:	Repetition of markers
Size:	Markers size
Show point labels	This option allows you to show/hide point markers on the series.
Visible	This option allows you to show/hide a particular series in the chart.
Sample	Look at the sample displayed in this box.

The *Axes* **tab:** This dialog allows you to manipulate axis settings, including Major Interval, Minor Intervals, Scales, Min, Max, and Gridlines & Tickmarks.

Field	Description
Combo Box	Choose the axis for which you want to look at parameters. To change parameters, press on the *Details…* button posted in the bottom of the dialog. Check the "Visible" box to show/hide a particular axis, including labels and tickmarks.
Major unit	This option allows you to set the Major interval for the selected axis. Check the "Show gridlines" box to show/hide gridlines on the major interval.
Tick mark type	Choose the tickmark type for the major interval.
Minor unit	This option allows you to set the Minor interval for the selected axis. . Check the "Show gridlines" box to show/hide gridlines on the minor interval.
Tick mark type	Choose the tickmark type for the minor interval
Details… Button	Press this button to access the **Axis Properties** dialog to change the selected axis properties. See explanation below.

The **Axis Properties** dialog: This dialog box includes the selected axis properties and is composed of the following tabs: *General*, *Scale*, *Labels* and *Grid lines*.

The *General* Tab:

Field	Description
Selected Axis	
Major unit	Choose the unit that will define the major interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of major interval.
Minor unit	Choose the unit that will define the minor interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of minor interval.

The *Scale* Tab: This dialog allows you to manipulate axis scales, including min, max, and logarithmic scales among others.

Field	Description
Selected Axis	
Minimum	This option allows you to set the minimum for the selected axis.
Maximum	This option allows you to set the maximum for the selected axis.
Scale unit	Set the scale unit for the selected axis.
Format	Set the axis format: None, Number, Currency, Scientific notation or Percentage.
Decimal places	Set the number of decimals for the selected axis.
Logarithmic scale	Set logarithmic or linear scales.
Zero line	Set the starting point at zero. For example, if you have a bar chart with a minimum value of −50, the starting point will be zero and you will have bars that go up and down, depending on their value.

The *Labels* Tab: This dialog allows you to manipulate labels displayed on the axis, including Rotation, Fonts and Axis Title.

Field	Description
Selected Axis	
Orientation	Use the *Degrees* arrows to set the rotated angle for labels in the selected axis.
Show labels	Check this box to show the labels in the selected axis.
2 Levels	Check this box to show the selected axis labels on two levels.
Rotate with chart	Check this box to rotate labels with chart.
Title	Set the title for the selected axis.

The *Grid lines* **Tab:** This dialog allows you to customize gridlines on both the major and minor intervals.

Field	Description
Selected axis	
Major gridlines	
Color:	Choose the colors for major gridlines in the selected axis.
Style:	Set the major gridline styles in the selected axis.
Weight:	Set the major gridline width in the selected axis
Minor gridlines	
Color:	Choose the colors for minor gridlines in the selected axis.
Style:	Set the minor gridline styles in the selected axis.
Weight:	Set the minor gridline width in the selected axis
Align with labels	Align labels with tickmarks and gridlines in the selected axis.
Interlaced	Set interlaced gridlines. The interlaced colors are achieved with the major grid lines color and the background of the chart.

REINFORCED CONCRETE DESIGN

TABLE OF CONTENTS

Chapter 13 Reinforced Concrete Design

General

Reinforced Concrete Design module

This module designs and verifies reinforced concrete structure, such as beams, columns, shear walls, and 2-way slabs. VisualDesign™ supplies the required lengths of main and transverse reinforcement for all concrete elements. Continuous beams, continuous columns and shear walls are called continuous systems. Actually, all continuous elements, except 2-way slabs, are automatically grouped to form vertical or horizontal continuous systems, which are represented by this icon on Elements toolbar.

The design is done according to the user-defined concrete specifications (design criteria, concrete cover, code, selection of main and transverse rebars). The module includes all types of rebars – standard imperial, metric, epoxy coated, meshes and FRP reinforcing bars, which are fibre reinforced polymer. Different steel grades and bending shapes are also included in the database.

VisualDesign™ calculates all required reinforcement to resist shear forces, axial forces and bending moments that are present in concrete elements. Furthermore, the program determines lengths of rebars and places opened or closed ties, hoops, with or without cross ties, etc. according to member dimensions and position of rebar. Finally, the user can modify his specifications and re-run a design.

Design of Beams, Columns and Shear Walls

VisualDesign™ stands apart from other reinforced concrete design programs because users have the power to edit rebar or create and verify their own design. Actually, with the **Stretch** and **Move** function, the user is allowed to stretch main rebars or move stirrup sequences if he wishes to modify the design or to create his own. In addition, the user can create his own patterns of transverse rebars and check his design. To help in editing rebars, cross-sections can be automatically generated or created at desired locations through the Cross-sections spreadsheet.

When the design is complete, the user has access to the *Rebar Placement* window which includes all functions needed to edit rebar location, dimensions, concrete element dimensions and to simultaneously verify the design with diagrams. View options are helpful to look at forces and strength diagrams. Resistance diagrams are simultaneously recalculated as the user edits rebars. The user can also look at displacements graphically, interaction curves for a column and dimensions for rebar placement, and finally, print all.

At the end of the project, the user can get a partial or complete bar list which includes rebar numbers, lengths, partial and total mass of reinforcement steel for all types of rebars used in his project.

Design of 2-Way Slabs

VisualDesign designs 2-Way slabs and places four layers of reinforcing bars according to the obtained stress/force contours. Openings are allowed in the slab. The display of calculated layers of bars (Mx+, Mx-, My+ et My-) is done through the **FE Results** tab of **View Options** dialog box. The rebar placement can be printed directly or imported as a DXF file

A powerful "mesher" is available to create a 2-way slab along with finite elements composed of triangular plates. Any slab geometry can be created and meshed with this tool (**Structure / Generator / FE Slab**). The user specifies the maximum areas of plates, the material and thickness along with vectors to orient finite elements. The slab surface is defined by selected nodes, located on the slab outline, and members surrounding openings (optional). The mathematical concept used in VisualDesign is called the *Convex Hull.*

A concrete specification is required and concrete covers are specified in the **Groups of plates – Surfaces** spreadsheet. When the design is over, stress/force contours can be displayed along with layers of rebars (one at a time of all together). A different colour is assigned to reinforcing bars in accordance with the size, and the pen weight varies with the size for a better display. The colour of rebars can be modified and disabled in respective rebars spreadsheets. Bar lists are available for each group of plates representing a 2-way slab. These results are located in the **Results / Bar lists** menu.

See also
Reinforced Concrete Design
Project Configuration
Rebar Placement Window
Rebar Placement Activation mode
Definition and Type of Continuous Systems
Design of Shear Walls
The Slab & Mesh Generator
Design of 2-Way Slabs
The Groups of Plates - Surfaces
The Groups of Plates - Shear Walls
Rigid Extensions
Concrete Specifications
Longitudinal Rebars
Transverse Rebars
Rebars Bending Shapes
Pattern of Transverse Rebars
Cross-sections
Stretch function
Move function
Bar List – Continuous Systems
Bar List – 2-way slabs
General Results Spreadsheet

Limitations of this Module

- The module does not calculate nor verify columns with combined tension-compression. However, bending-compression combination is verified.

- Torsion is not verified. The program does not calculate extra reinforcement required for torsion.

- Continuous and adjacent members that are part of a standard continuous system must be collinear and not exceed 1 degree angle between the member s' longitudinal axis. However, members that will be part of a continuous system with a varying geometry cannot exceed an angle of 65 degrees.

Important

VisualDesign™ considers continuous systems as primary lateral resisting systems (See Code A23.3). Therefore, it will put hooks at end of longitudinal reinforcing bars for development lengths, as required by the code. If you change hooks by straight bars, VisualDesign™ will not warn you that you no longer respect the code.

We strongly recommend that you launch a design of continuous systems first. After the design, study results carefully and then modify rebar placement. If you modify dimensions, do not forget to launch another design for the distribution of forces in elements will have changed.

Rebar Placement for Bending Moments

VisualDesign™ designs longitudinal rebars in a continuous system in two steps:

Design of reinforcement in positive bending zones;

Design of reinforcement in negative bending zones considering rebars that have been placed in the previous step.

At the launching of design, there is no bar placed in the continuous system. VisualDesign™ will start by placing the required rebars for positive bending moments (lower part of the section). Then, the program will place rebars for negative bending moments considering the rebars in the upper part.

When the window is open, all rebars are present in the elevation view and the calculation of Mr considers these rebars in all points

If VisualDesign™ cannot find a solution for the design; the chosen solution will respect the c/d ratio and will show the biggest Mr value.

Materials - Steel Grades & Reinforcing Bars

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 $\lambda=1,00$ for normal density $\lambda=0,85$ for structural semi-low-density concrete in which all of the fine aggregate is natural sand. $\lambda=0,75$ 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

Column	Description	Editing
Elt	Concrete modulus of elasticity (long term)	Single click
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.

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

CivilDesign inc. Engineering Software

Column	Description	Editing
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	Rebar number or name (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

Column	Description	Editing
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
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

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:
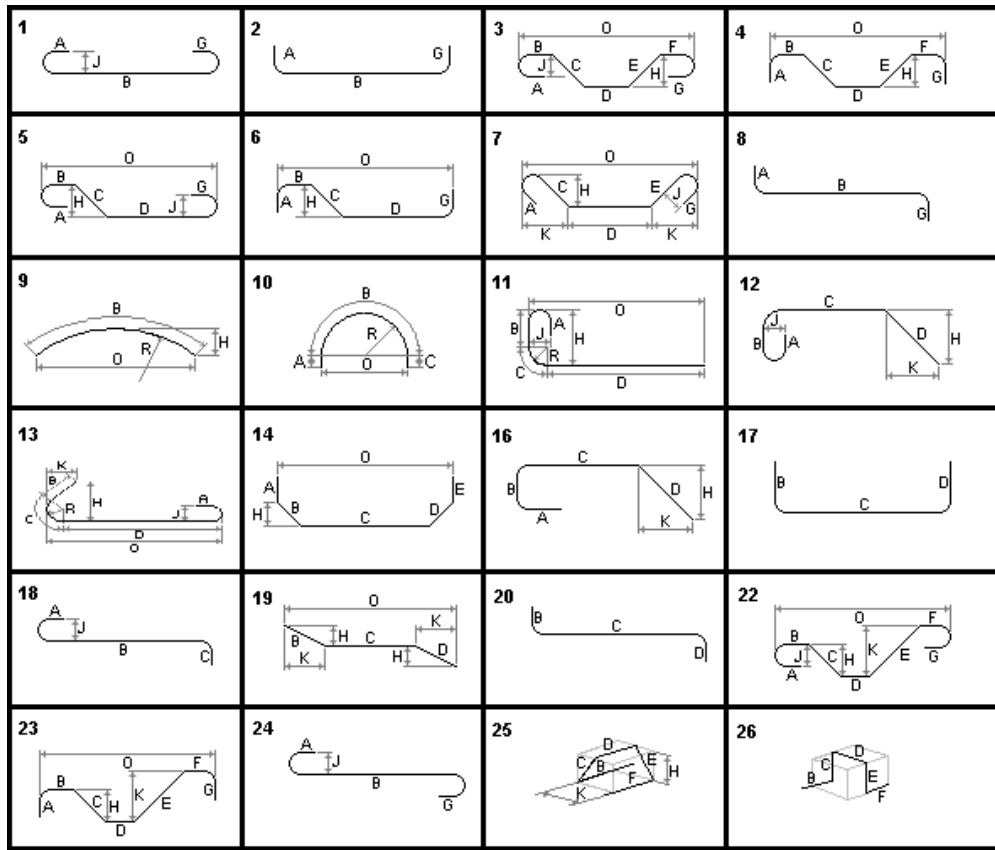
Types S1 to S11, SP included, represent bending shapes for transverse reinforcement bars for various concrete members. You will notice that the following bending shapes are open ties only:
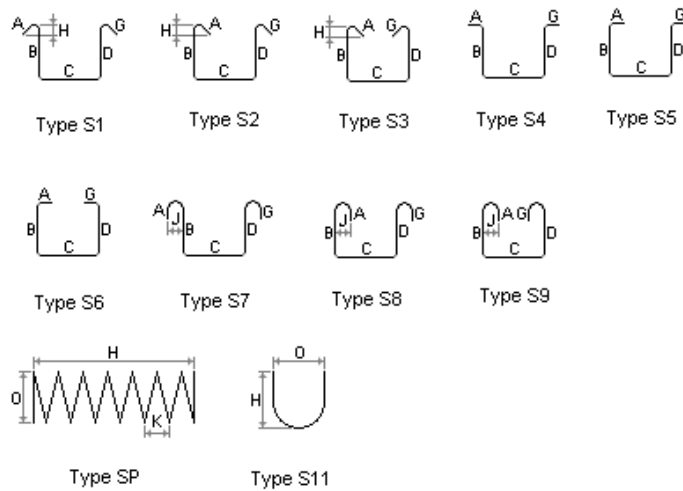
Types T1 to T9 are closed ties (including hoop reinforcements, cross-ties and seismic cross-tie) for transverse reinforcements that must carry only shear stresses present in concrete members. These bending shapes are used in seismic design. They are the following:
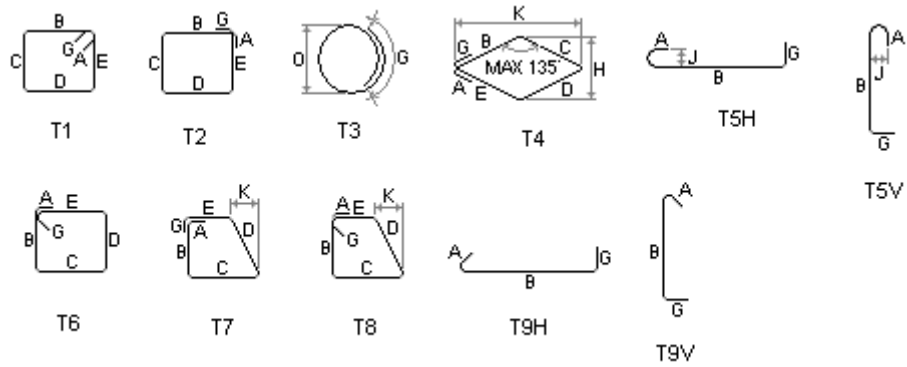
See also

Standard Reinforcing Bars
FRP Reinforcing Bars
Rebars Steel Grades

Project Configuration

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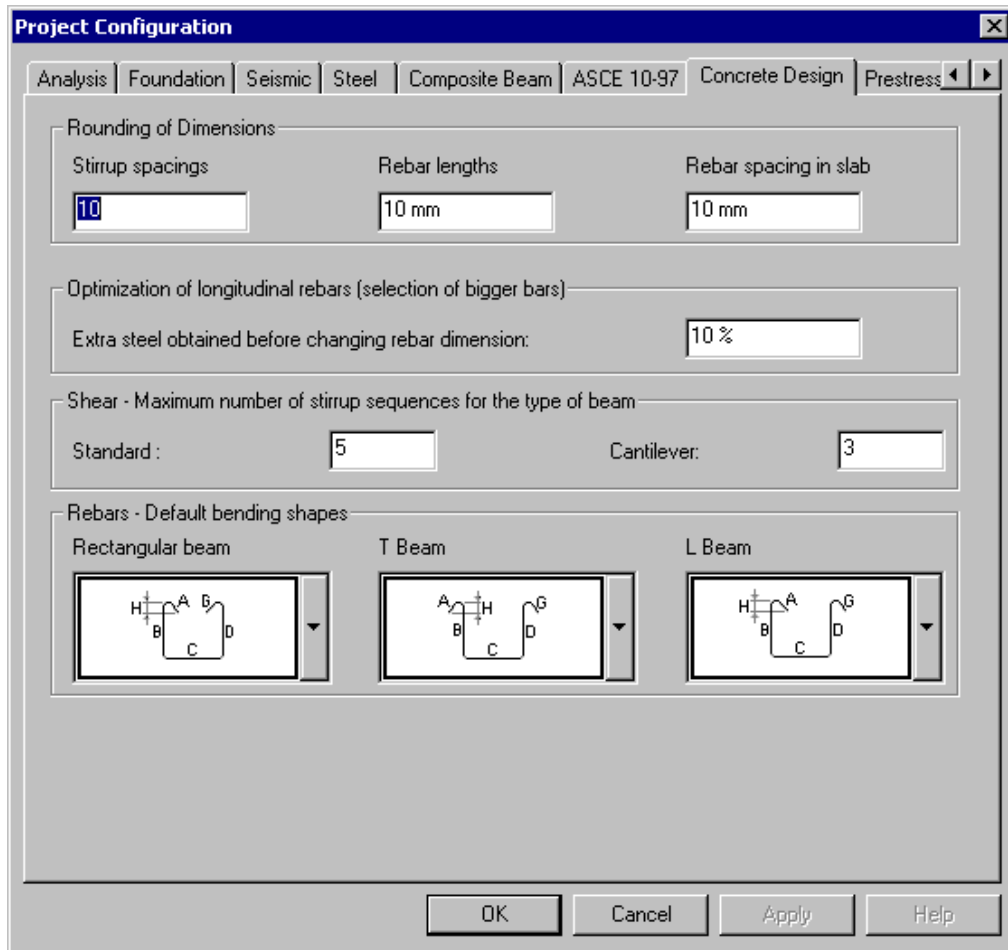
See the table below to know the definition of headings included in this tab.

Field	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.

Field	Description
Rebar spacing in slabs	Specify a rounding for the spacing of rebars in slabs.
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

Specifications

Concrete Specification Spreadsheet

This spreadsheet includes a list of pre-defined concrete specifications. Parameters can be modified and new specifications can be added at the bottom line of the spreadsheet.

Before proceeding with a concrete design of beams, columns, beam-slab or shear walls, a concrete specification must be assigned to each continuous system to be designed. This is done through the **Continuous Systems** spreadsheet. The concrete specification for 2-way slab design is selected in the **Groups of Plates – Surfaces** spreadsheet.

The spreadsheet is divided into tabs: **General**, **Beam-Column-Joist**, **Beam-slab**, **FE Slab**, and **Shear Wall**.

Generation modules (Culverts, and Abutments, Piers & Retaining Walls) automatically create concrete specifications and assign them to continuous systems.

The General tab

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Specification number (16 alphanumerical characters).	Single click
Code	Choose the building code that will be used for the design or verification of continuous systems.	Double-click
Type of Analysis	Indicate in which case the specification will be used for: Design or Verification. If you selected code S6-00, the option "Bridge Evaluation" will be available.	Double-click

Column	Description	Editing
Maximum Capacity Factor	The design is done with respect to this given percentage of capacity. For example, if a value is inferior to 100%, it means that you are more severe than the chosen code.	Single click
Calculation Method Mr/Vr (positive)	Choose an option for the calculation of main and transverse reinforcement for positive bending moment: *Maximize Mr* or *Maximize Vr*. Look at Note 1.	Double-click
Calculation Method Mr/Vr (negative)	Choose an option for the calculation of main and transverse reinforcement for negative bending moment: *Maximize Mr* or *Maximize Vr*.	Double-click
Epoxy coated bars	Indicate if bars are epoxy-coated by choosing option [x].	Double-click or Space bar
Optimization Main reinforcement	Select a type of optimization for main reinforcement. Refer to topic Optimization of main reinforcement	Double-click
Material Main reinforcement	Choose steel grade for main reinforcement.	Double-click
Selection of Main Reinforcement	Choose the main reinforcing bars that will be used for the design.	Double-click
Design method	In the drop-down list box, select the method that will be used for the reinforced concrete: *General* or *Simplified* Method.	Double-click
Calculation method for beta2	This factor is used for the calculation of cracking according to S6-00 standard only. Two methods are available: MTQ Modified and Standard. Refer to clause 8.12.3, at section Crack Control.	Double-click

Note 1: Maximize Mr and Maximize Vr

The first option, *Maximize Mr*, considers tensioned rebars at the top and bottom of the section. Consequently, the value "d" or "dv" can be smaller and Mr, greater. The second option considers tensioned rebars at the bottom only, so "d" or "dv" will be longer and Mr, smaller. This design will maximize the shear resistance of the member.

Optimization of Main Reinforcement

In the **General** tab, the user must select a type of optimization for main reinforcement.

Many options are available in the list box. Look at the table below to know each one of them:

Option	Optimization
None	No optimization will be done. This option must be selected for prestressed concrete design because cables are placed as longitudinal reinforcement.
Weight	This option is mostly used for the design of beam and column main reinforcement. Optimization is based on the weight of rebars. VisualDesign will change to bigger rebars if the weight is 10% higher than required, as specified in the Concrete Design tab of Project Configuration.
Weight, M+	This option, based on the weight of rebars, will optimize the main reinforcement for positive bending moments only.
Weight, M-	This option, based on the weight of rebars, will optimize the main reinforcement for negative bending moments only.
Constant "s" (M+ &M-)	This option allows getting a design of main reinforcement for positive and negative bending moments with a constant spacing between rebars or a multiple of this spacing.

The Beam/Column/Joist tab

Group: Structural Data

Column	Description	Editing
Number	Specification number (16 alphanumerical characters).	No
Maximum no. of layers in tension	Specify the maximum number of layers for rebars in tension.	Single click
Maximum no. of layers in compression	Specify the maximum number of layers for rebars in compression.	Single click
Optimization Transverse reinforcement	Select a type of optimization for transverse rebars (stirrups): None or according to the weight of rebars.	Double-click
Transverse reinforcement Material	Select a steel grade for stirrups.	Double-click

Column	Description	Editing
Selection of Transverse Rebars	Choose the transverse reinforcing bar dimensions that will be used for the design.	Double-click
Structure	This column is composed of options *Buried-Prefab, Buried-standard* or *Standard*. Buried culverts are designed according to clauses 7.8.8 to 7.8.12, at section *Buried Structure* of Code S6-00. A different resistance factor is applied to prefab structures.	Double click
Supports for stirrups	This default bar will be used as supports for stirrups in beam corners if none have been placed during the design.	Double-click

The Beam-slab tab

Group: Structural Data

Column	Description	Editing
Number	Specification number (16 alphanumerical characters).	No
Reinforcement Spacing	Specify the spacing of main reinforcement for positive and negative moment. The bar diameter will be chosen according to this configuration.	Single click
Reinforcement Minimum Spacing	Enter the minimum spacing of main reinforcement.	Single click
Extension of rebars	You are allowed to extend rebars or not. Double-click and choose one of the following options among the drop-down list box: No extension, All Rebars, 1 out of 2, 1 out of 3 or 1 out of 4.	Double click
Mesh Transverse Spacing	Specify the transverse spacing between bars for meshes. This distance is the transverse bars development length for welded meshes.	Single click
Structure	This column is composed of options *Buried-Prefab, Buried-standard* or *Standard*. Buried culverts are designed according to clauses 7.8.8 to 7.8.12, at section *Buried Structure* of Code S6-00. A different resistance factor is applied to prefab structures.	Double click

The FE Slab tab - Concrete Specifications

Complete this tab and select this specification in the groups of plates (surfaces) spreadsheet before launching the concrete design of 2-way slabs.

Group: Structural Data

Column	Description	Editing
Number	Specification number (16 alphanumerical characters).	No
Temperature reinforcement	Select the rebar dimension for temperature reinforcement in the slab.	Double-click

See also

Group of plates - Surfaces

The Shear Wall tab- Concrete Specifications

Group: Structural Data

Column	Description	Editing
Number	Specification number (16 alphanumerical characters).	No
Optimization Transverse reinforcement	Select a type of optimization for transverse reinforcement: *None* or *Weight of rebars*.	Double-click
Transverse reinforcement Material	Select a steel grade for stirrups.	Double-click
Selection of Transverse Rebars	Choose the transverse reinforcing bar dimensions that will be used for the design.	Double-click
Rebars Spacing Local x-dir.	Spacing of longitudinal rebars placed along the length of the wall for bending resistance. These rebars are located in columns (corners and junction of components) and resist to bending. The shear wall local axis system is shown below. This spacing corresponds to parameter "sx" on the image below.	Single click

Column	Description	Editing
Rebars Min. Spacing Local y-dir	Minimum spacing for longitudinal rebars placed perpendicular to the length of the wall. These rebars are located in columns (corners and junction of components) and resist to bending. The shear wall local axis system is shown below. This spacing corresponds to parameter "sy" on the image below.	Single click
Max. number Rebars Local y-dir.	Maximum number of longitudinal rebars placed perpendicular to the length of the wall. These rebars are located in columns (corners and junction of components) and resist to bending.	Single click
Skin reinforcement	This default rebar will be used as skin reinforcement in the shear wall.	Double-click

Local Axes of a Shear Wall
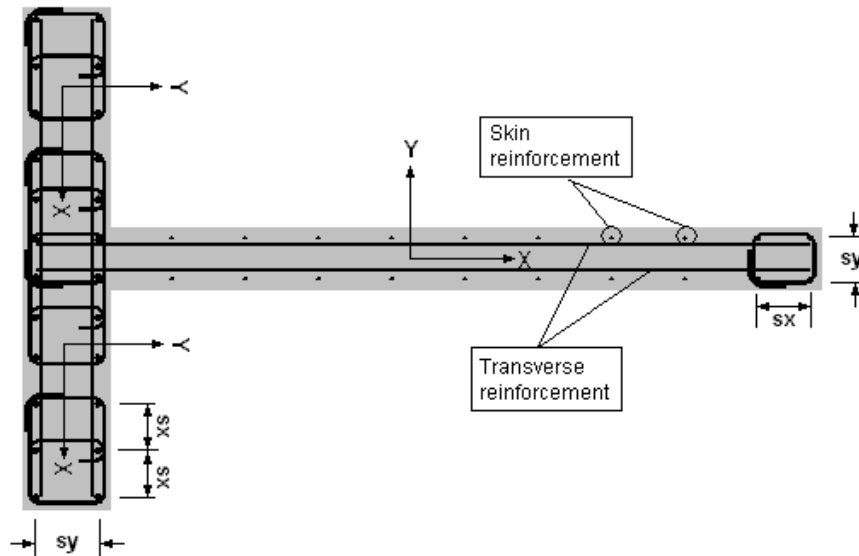
See also

The *Shear Wall* type of Continuous System
Ductility of Shear Walls
Modeling and Designing a Shear Wall
Calculation of Required Transverse Reinforcement in Shear Wall

Selection of Reinforcement in a Concrete Specification

Select main rebars or meshes and transverse rebars in the concrete specification, by double-clicking in each following cells: "Selection of main reinforcement" (**General** tab) and "Selection of transverse reinforcement" (**Beam/Column/Joist** tab). The appearing dialog box includes a selection tree of standard imperial and metric rebars, meshes and FRP bars. Activate boxes corresponding to standard rebars, meshes or FRP bars that you want to include as choices for the design. If meshes were chosen, specify rebar spacing.

Pre-defined reinforcement are included in spreadsheets, which are accessible in the **Common / Reinforcement** menu. The name of imperial rebars must begin with the symbol #.

Concrete Members

Modeling a Concrete Member

- While in the Structure activation mode, select a rectangular, circular, T, or L section and choose a concrete material in the **Member** tab (**Member Characteristics** dialog box).

- Activate design criteria in the **Member** tab and complete the **Concrete Design** tab in the same dialog box.

- Make sure that the local axis systems of members that will be forming a continuous system are pointing in the same direction. Otherwise, VisualDesign will not consider these members as continuous.

See also
Member Dialog Box
Modeling a Member with Variable Heights
Modeling Valid Rigid Extensions

Rigid Extensions

Rigid extensions are required at the left and right of a support for continuous members (beams). They are also required at the junction of transverse beams, and where a concrete section changes. VisualDesign needs this information to calculate the required development lengths of reinforcing bars.

If a building is composed of beams and columns, rigid extensions can be automatically calculated with the function **Calculation of Rigid Extensions** (**Structure / Tools**). However, if members are very thick, this tool can generate rigid extensions that are too long. If this is the case, we recommend that you define rigid extensions yourself through the **Connection** tab (**Member Characteristics** dialog box).

Distributed Loads on Floors and Rigid Extensions

VisualDesign computes the load on each rigid extension.

Distributed Load on Floor
kPa
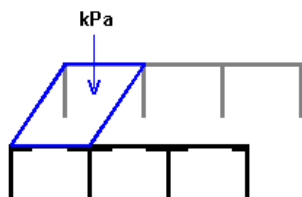
VisualDesign will automatically transform the transferred load on each rigid extension.

VisualDesign will calculate equivalent loads as follows:

Equivalent Loads on Rigid Extension
Calculated by VisualDesign
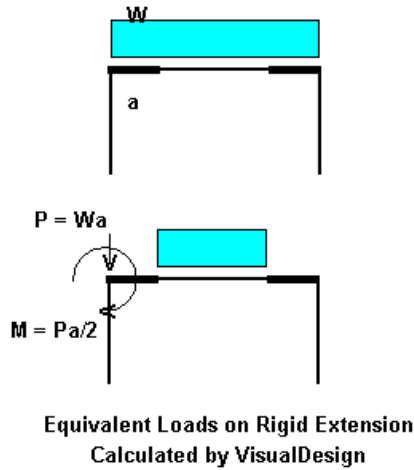
See also
Automatic Calculation of Rigid Extension
Modeling Valid Rigid Extensions

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.
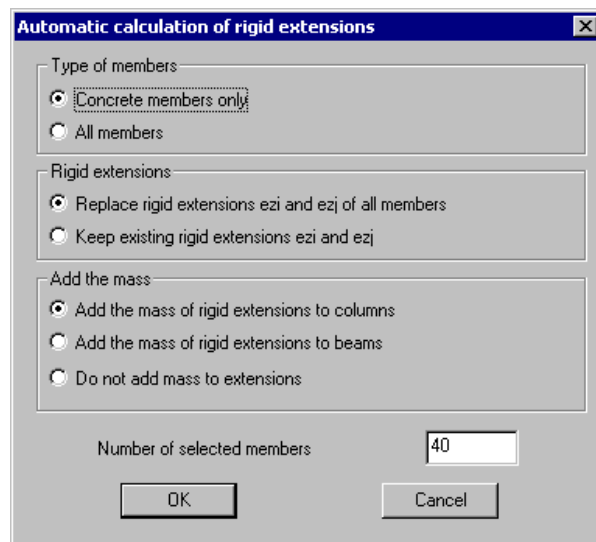
Connection tab

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**).
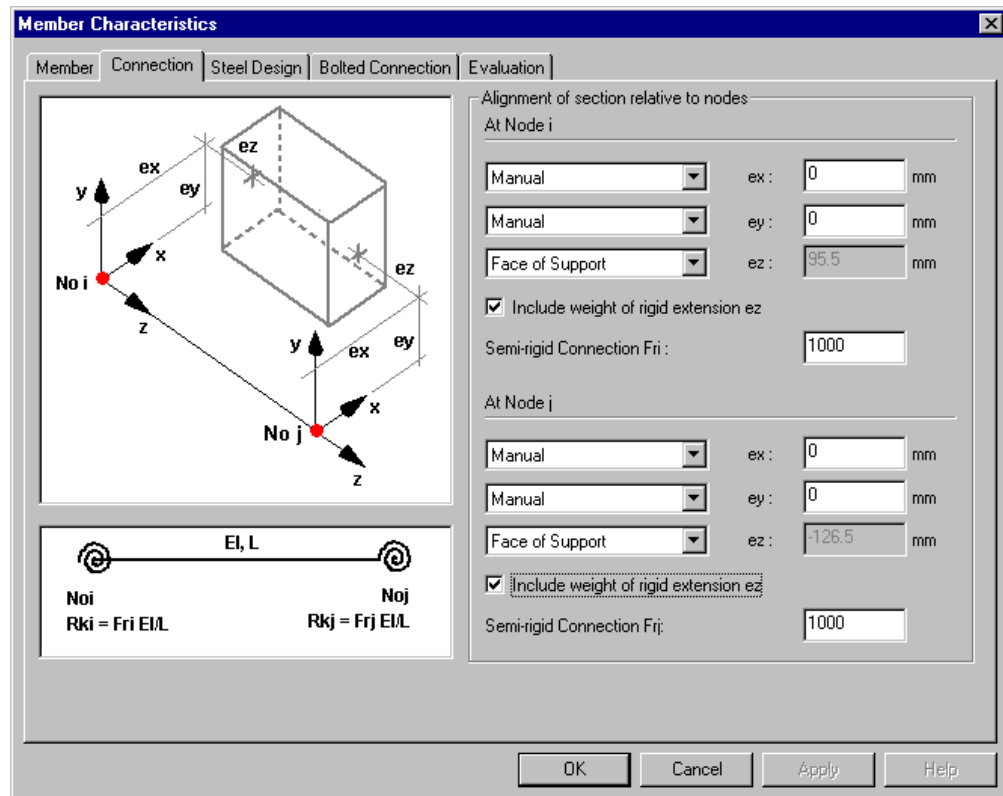
Alignment "ex"

Choose an alignment from the drop-down list box: "Manual", "Left" or "Right".

- 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.

- By choosing "Left", the section will be positioned left of the node and the value of eccentricity "ex" will be automatically calculated and written in this field. This value will be negative and equal to half of the section width.
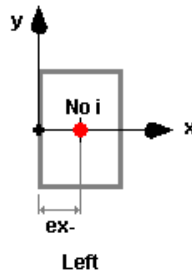
- By choosing "Right", the section will be positioned at the right of the node and the value of eccentricity "ex" will be automatically calculated and written in this field. This value will be positive and equal to half of the section width.
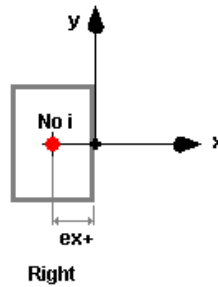
Alignment "ey":

Choose an alignment from the drop-down list box: "Manual", "Above" or "Below".

- 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.

- By choosing "Above", the section will be positioned above the node and the value of eccentricity "ey" will be automatically calculated and written in this field. This value will be positive and equal to half of the section height.

- By choosing "Below", the section will be positioned below the node and the value of eccentricity "ey" will be automatically calculated and written in this field. This value will be negative and equal to half of the section height.

Rigid Extensions "ez"

IMPORTANT: All concrete members and prestressed concrete members must have rigid extensions because VisualDesign needs them to calculate the required rebars development lengths.

Specify the length of rigid extension in the drop-down list box "Manual" or select option "Free edge".

- 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.

- If you choose "Free edge", the end of the section will be positioned at the face of the support. As an example, the end of a beam will be positioned at the face of the column.

Weight of Rigid Extensions:

Include the mass of the rigid extension "ez" at node i and j for columns, <u>OR</u> beams, by checking this box.

See also
Automatic Calculation of Rigid Extensions
Semi-rigid connections
Modeling Valid Rigid Extensions
Modeling a concrete beam with variable geometry

Modeling Valid Rigid Extensions

Rigid extensions must be modeled at <u>each side</u> of a support, for continuous members. This means that if a rigid extension is modeled at node j of member M1, a rigid extension must <u>also</u> be modeled at the node i of contiguous member (M2). Look at the examples below:

Valid Rigid Extensions:
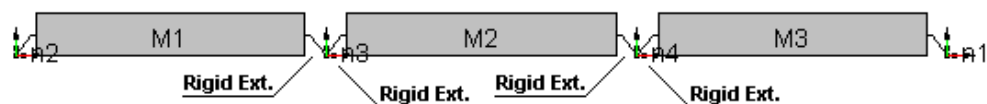
Invalid Rigid Extensions:
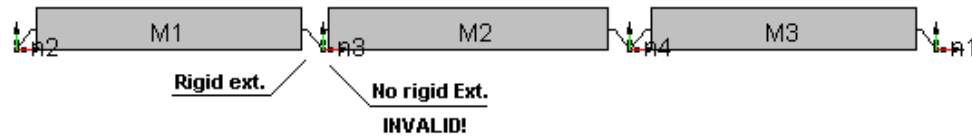
Notes:

In order not to over-evaluate the mass of your structure, we recommend including the mass of rigid extensions for columns only. Make sure that you do not include both columns and beams.

We recommend not splitting rigid extensions with the **Split** functions.

See also
Automatic Calculation of Rigid Extensions
Semi-rigid connections
Modeling a concrete beam with variable geometry

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:
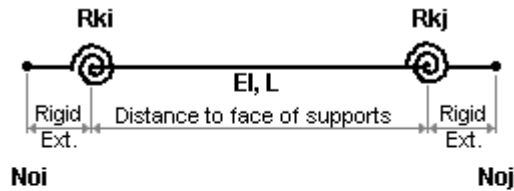
Where

$$Rk_i = Fr_i \ EI/L$$

$$Rk_j = Fr_j \ EI/L$$

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.
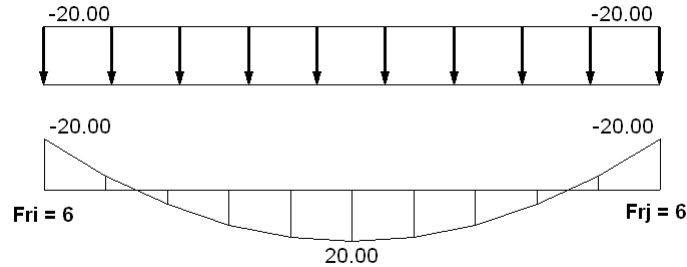
Note:

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.

See also
Connection tab (Member Characteristics dialog box)

Concrete Design tab

This tab is only available for users owning the *Reinforced Concrete Design* module. This tab appears if design criteria are activated in the **Member Characteristics** dialog box and if a concrete material is assigned to the member.

The table below gives a description of fields included in the **Concrete Design** tab of the **Member** dialog box.

Field	Description
Height of Section	You have to complete this section for a member that will be part of a continuous system with a variable geometry.
Node i	Height of section at node i for a continuous system with a variable geometry.
Node j	Height of section at node j for a continuous system with a variable geometry.

Field	Description
Design near Maximum Forces	
Consider Vy	Choose an option: Consider Vy at *d* from face of support (Simplified method) or *dv* (General method) or consider Vy at face of support.
Limit Proportioning of longitudinal reinforcement to Mf/dv max.	Choose an option among the following: *No*, *For Mx+*, *For Mx-* or *Both Mx+ Mx-*. Refer to clause 8.9.3.10.1 of CAN/CSA-S6-00 Standard or clause 11.4.9.1 of CAN/CSA-A23.3-95 Standard.
Characteristics	
Continuous non rectilinear reinforcement for variable geometry	Check the box if you want VisualDesign to place rebars in such a way to fit the varying surface of concrete elements, for a member that will be part of a continuous system with a varying geometry.
Prefabricated	Activate this option if the member is prefabricated.
Backfilled Structure – backfilled side	
Top of section	Check the box if the top of the section is backfilled, for the placement of temperature reinforcement according to article 7.8.11.2 of CAN/CSA-S6-00 Standard. Display the member local axis system.
Bottom of section	Check the box if the bottom of the section is backfilled, for the placement of temperature reinforcement according to article 7.8.11.2 of CAN/CSA-S6-00 Standard. Display the member local axis system. Display the member local axis system.

See also
Member Dialog Box
Modeling a Member with Variable Heights

Modeling a Continuous System with a Variable Geometry

Follow the steps to model continuous members with variable geometry. The longitudinal axis of contiguous members with a variable geometry cannot exceed an angle of 65 degrees.

Look at the example below:

Split the Member

- Create a member.

- Select the member and use function **Multiple Split** on Split toolbar. Split the member into smaller parts.

Assign a Concrete Section to each Member

- Click on a member and open the **Member Characteristics** dialog box..

- Assign a concrete section in the **Member** tab and activate design criteria.

- Then, select the **Concrete Design** tab and enter the section height at node i and j. Look at the example below.

Align Members and Model Rigid Extensions

- Once that all sections are defined, select them all and go to the **Connection** tab (**Member Characteristics** dialog box). The axis passing through all members must be aligned either at the top or at bottom, according to the shape of final beam that you want to model. In our example, the alignment "ey" must be set to "Above" for all members.
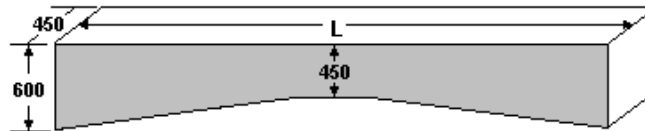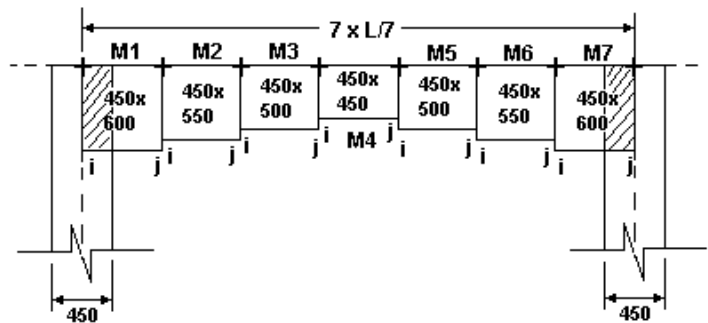
- Now, rigid extensions must be added to member ends that are connected to column. Rigid extensions are corresponding to ez (**Connection** tab).

- Double click on member M1 and choose option Free Face in the "ez" list box for node i only. Check the box "Include weight of rigid extension".

- Double click on member M7 and choose option Free Face in the "ez" list box for node j only. Check the box "Include weight of rigid extension"
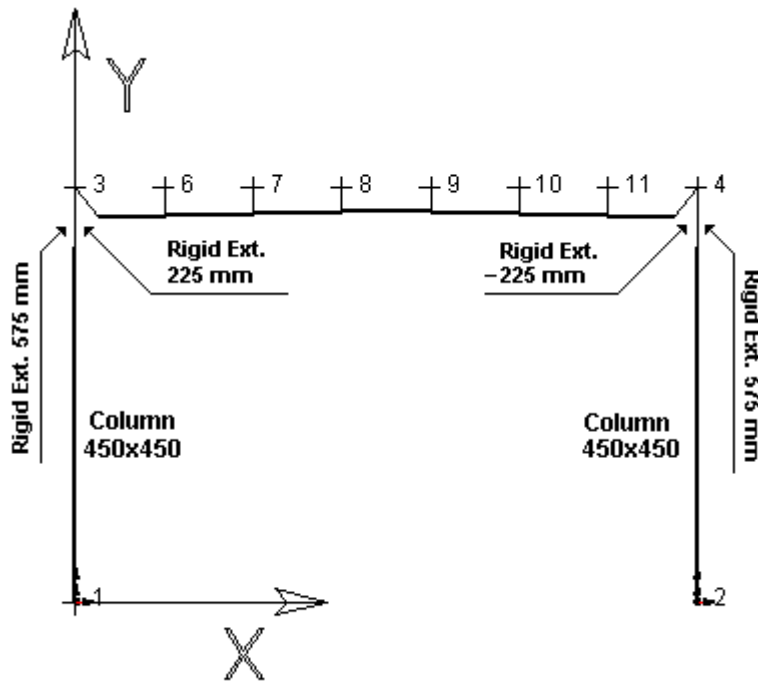
For the Design

- Select all members and open the **Member Characteristics** dialog box. Select the **Concrete Design** tab and check the box "Continuous non rectilinear reinforcement for a variable geometry".

- Activate the continuous system icon and double-click on the beam. Select the *Beam/Column* option in the "Type" column of **Continuous Systems** spreadsheet.

See also

Continuous Systems spreadsheet
Concrete Design tab (Member dialog box)
Summary - Design of miscellaneous continuous systems
The *Beam/Column* continuous system with variable geometry

Members Spreadsheets

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 or steel).

Other tabs can be the following if you own all other modules.

Concrete Design
Steel Design
Timber Design
Bolted Connections
Evaluation

The Members Spreadsheet

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

Column	Description	Editing
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
Usage	Select the member usage among the drop-down list box. See topic Member Usage.	Double-click
Dead load	Dead load case that will automatically consider the member self-weight in load combinations.	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 axial stiffness	Specify the effective axial stiffness of this member. Otherwise, enter a value of 1.0.	Single click

Column	Description	Editing
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

See also
Orthogonal and Major/Minor Axis System
Member with a Linear Behaviour

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

Column	Description	Editing
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
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

See also

Members Spreadsheet
Connection tab
Automatic Calculation of Rigid Extensions

Concrete Design Spreadsheet

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	Number (12 alphanumerical characters) used to describe the concrete member.	No
Height of Section Node i	For a member that will be part of a continuous system with a variable geometry, enter the height of section at node i.	Single click
Height of Section Node j	For a member that will be part of a continuous system with a variable geometry, enter the height of section at node j.	Single click
Consider Vy	Choose an option: Consider Vy at *d* from face of support (Simplified method) or *dv* (General method) or consider Vy at face of support	Double-click
Limit Proportioning of longitudinal reinforcement to Mf/dv max.	Choose an option among the following: *No*, *For Mx+*, *For Mx-* or *Both Mx+ Mx-*. Refer to clause 8.9.3.10.1 of CAN/CSA-S6-00 Standard or Clause 11.4.9.1 of CAN/CSA-A23.3-95 Standard.	Double-click
Continuous non-rectilinear reinforcement	Select option [x], if you want VisualDesign to create a polyline and place rebars in such a way to fit the varying surface of concrete elements, for a member that will be part of a continuous system with a variable geometry.	Double-click or Space bar
Prefabricated	Activate this option [x] if this member is prefabricated.	Double-click or Space bar
Backfill over top of section	Activate this option [x] if there is backfill over the top of the section, for the placement of temperature reinforcement according to article 7.8.11.2 of CAN/CSA-S6-00 Standard.	Double-click or Space bar
Backfill over bottom of section	Activate this option [x] if there is backfill over the bottom of the section, for the placement of temperature reinforcement according to article 7.8.11.2 of CAN/CSA-S6-00 Standard.	Double-click or Space bar
Order	Shaded field that informs the user of the member order among the continuous system.	No

Column	Description	Editing
Continuous System	Shaded field that informs the user of the continuous system number in which the member belongs.	No

See also

Continuous System
Continuous Systems Spreadsheet
Rebar Placement window

Continuous Systems - General

Definition of Continuous Systems

The "Continuous System" icon of Elements Toolbar

The continuous system concept in VisualDesign must be well mastered because design features of the **Reinforced Concrete Design** and **Prestressed Concrete Design** modules are based on continuous systems properties and types.

Standard Continuous Systems

A *Standard* continuous system is composed of co-linear (same plane) continuous members having a constant section along their lengths. An angle of 1 degree between adjacent members' longitudinal axis cannot be exceeded. A continuous system can be vertical (many columns) or horizontal (continuous beams).

There are four types of standard continuous systems: *Beam/Column*, *Slab*, *Shear Wall*, and *Joist*. The type must be specified in the **Continuous Systems** spreadsheet. In addition, the design can be done considering bending only or bending/compression.

See
The *Beam/Column* type of Continuous System
The *Beam-slab* of Continuous System
The *Shear Wall* type of Continuous System
The *Joist* type of Continuous System

Continuous Systems with Variable Geometry

A continuous system having a variable geometry is composed of continuous members that have different sections. The angle between members' longitudinal axis must not exceed 65 degrees. This type of continuous system is used to generate culverts.

The modeling of such members is done through the **Member Characteristics** dialog box, as you will see further on.

A continuous system with a variable geometry can be of two types: *Beam/Column* and *Slab*.

Local Axis System for Continuous System

The continuous system local axis is dependent upon the members' local axis systems that are composing the continuous system.

Members must be oriented in the same direction to be part of the same continuous system. Display the members' local axis system through the **View Options** to make sure that members' local z-axis are pointing in the same directions.

Creation of Continuous Systems

Continuous systems are automatically created if the user activates the design criteria in the **Member** tab of **Member Characteristics** dialog box. The program gives a name for each continuous system, such as S_1, S_2, S_3, etc. and writes data in the **Continuous Systems** spreadsheet available in the **Structure** menu.

The **Continuous Systems** spreadsheet can be selected in the Structure mode only. Double-click on a continuous system and the spreadsheet will include information for this continuous system. Select all continuous systems and click on the **Properties** icon: the spreadsheet will include information for all continuous systems.

See also
Member tab (Member Characteristics dialog box)
Summary - Design and Types of Continuous Systems
Continuous System Properties and Restrictions

Continuous System Properties and Restrictions

Members' rigid extensions are required because VisualDesign needs them to calculate appropriate rebar development lengths.

A continuous system is composed of continuous members that must be in the same plane. Continuous systems may be either vertical (columns) or horizontal (beams).

A continuous system is unique. This means that all the elements composing a continuous system must have the same beta angle and the same eccentricities (ex, ey) and rigid extensions (ez) at member ends.

Continuous systems cannot be grouped together

N. B. The **Add** function of **Edit** menu cannot be used to add continuous systems. They are automatically created.

Beams and Columns

Rebar placement can be copy to a second continuous system by using the function **Save Rebar Placement As**, which is available in the **File** menu of *Rebar Placement* window.

See also

Automatic Calculation of Rigid Extensions
Continuous Systems Spreadsheet
Save Rebar Placement As

Continuous Systems Spreadsheet

This spreadsheet is accessible in the Structure activation mode and is located in the Structure menu. To open it, activate the continuous system icon and double click on a continuous system or select many and press the **Properties** icon.

Group: Structural Data

Column	Description	Editing
System ID	Automatically calculated	No
Number	Continuous system number (12 alphanumerical characters).	Single click
Specification	Select the concrete specification for the design in the drop-down list box.	Double-click
Type	Choose the type of continuous system. See topic Definition and Types of Continuous Systems	Double-click
Interaction	Choose between a bending or bending/compression interaction with respect to the type of continuous system.	Double-click
Description	Enter a description of the continuous system	Single click
Exposure (Top, Bottom, Left and Right)	A23.3-95: Choose the type of exposure: Interior, Exterior, Corrosive or Ground contact. VisualDesign will consider the required concrete cover at the design. S6-00: only the option *Manual* is available.	Double-click

Column	Description	Editing
Cover (Top, Bottom, Left and Right)	A23.3-95: Net concrete cover from exterior face of main reinforcement. If you enter a value here and you specified a type of exposure, VisualDesign will consider the most critical case between the two data.	Single click
	S6-00: enter the net concrete cover calculated from exterior face of stirrups.	
	N.B. The left and right sides are determined from the member positive local axes.	
Crack Control Top	Enter the parameter for crack control that will be applied to the top of the section, according to appropriate building code. Code A23.3-95: z Code S6-00:$\beta 2$	Single click
Crack Control Bottom	Enter the parameter for crack control that will be applied to the bottom of the section according to appropriate building code. Code A23.3-95: z Code S6-00:$\beta 2$	Single click
Seismic Design		
Primary Lateral Resisting System	If this continuous system is part of a primary lateral resisting system, choose option [x]. Bar lengths will be developed to reach the yield strength fy. See clause 12.11.2 (CNBC) and the note below*.	Double-click or Space bar
Ductility	Select the ductility of this continuous system among the following: Not ductile (R=1.5), Nominal ductility (R=2.0), or Ductile (R> 2).	Double-click
R '	For a ductile continuous system: This factor will multiply the obtained value for Vf max in order to get probable forces when R > 2, and nominal forces when R > 1.5. Default values are: 1.3 for nominal ductility, 1.7 for ductility, and 1.0 for a not ductile continuous system.	Single click

***Note**: For the moment, VisualDesign™ considers all continuous systems as lateral-force-resisting-systems. So, all bar lengths will be developed in order to reach the yield strength fy. However, if you delete hooks, VisualDesign will not warn you if you no longer respect the code provisions.

Continuous Systems - Types

The Beam/Column Type

The *Beam /Column* type is mostly used because it represents a single beam, continuous beams, or columns in a typical building.

VisualDesign will assign the *Beam/Column* type of continuous system to vertical and horizontal continuous elements. However, the program will initialize a bending/compression interaction for vertical elements and bending only, for horizontal elements.

A concrete specification must be assigned to each continuous system included in the **Continuous Systems** spreadsheet.

A concrete specification can be assigned to both vertical and horizontal continuous systems. However, if a design parameter is different (selection of rebars, optimization of rebars, concrete material or stirrup design), a second specification must be created and assigned to appropriate continuous systems.

Variable Geometry

This type of continuous system is automatically created when VisualDesign generates the structural model of an abutment or pier structure, which may include a beam with a variable geometry. Refer to the **Generation of Abutments, Piers, and Retaining Walls** module.

If you model yourself a beam that will be part of a continuous system with variable geometry, you must activate the design criteria and then, specify the member height at node i and j and finally, activate the option "Continuous Non rectilinear Reinforcement for variable geometry" in the **Concrete Design** tab of **Member Characteristics** dialog box. Otherwise, the program cannot recognize such members. VisualDesign will place rebars in such a way to fit the varying surface of concrete elements.

VisualDesign will then initialize the continuous system as a *Beam/Column* type in the **Continuous Systems** spreadsheet and the design of stirrups will be adjusted according to the height and width of each member with variable height. The design of main reinforcement considers bending only.

If the following option "Continuous Non rectilinear Reinforcement for variable geometry" is activated, VisualDesign builds a polyline with member end nodes in such a way that longitudinal rebars, for each member, are put end to end and become one object. The nodes that compose the polyline are listed in The Position tab and are accessible through the **Longitudinal Rebars** spreadsheet.

The Beam-slab Type

The Reinforced Concrete module does not design two-way slabs as ADOSS does. VisualDesign is much more accurate.

A *Beam-slab* of continuous system can be assigned to a flat slab or a footing. This type of continuous system is not automatically generated. The user must create a concrete rectangular section of 1m width and of thickness x (through the **Rectangular Sections** spreadsheet). VisualDesign will recognize this type of continuous system and will check the shear resistance.

Choose a bending interaction (**Continuous Systems** spreadsheet), for a slab without compression and specify no stirrup design in the concrete specification spreadsheet.

If the slab acts as a rigid diaphragm, the bending/compression interaction must be specified.

Variable Geometry

This type of continuous system is used to model concrete walls that may be a monolithic structure such as culverts. The design considers bending and compression in the slab and the main reinforcement is calculated per meter of width. There is no stirrup design.

VisualDesign when generating culvert walls and the foundation slab of piles, abutments, and retaining walls automatically creates the slab with a variable geometry. By default, VisualDesign uses a concrete section of 1000mm of width. The slab thickness can be modified through the "Height of Section" fields of **Concrete Design** tab in the **Member Characteristic**s dialog box.

If you model yourself a slab (or concrete walls) that will be part of a continuous system with variable geometry, you must activate the design criteria and then, specify the member height at node i and j and finally, activate the option "Continuous Non rectilinear Reinforcement for variable geometry" in the **Concrete Design** tab of **Member Characteristics** dialog box. Otherwise, the program cannot recognize such members. VisualDesign will place rebars in such a way to fit the varying surface of concrete elements.

If the following option "Continuous Non rectilinear Reinforcement for variable geometry" is activated, VisualDesign builds a polyline with member end nodes in such a way that longitudinal rebars, for each member, are put end to end and become one object. The nodes that are composing the polyline are listed in The Position tab spreadsheet, which is accessible through the **Longitudinal Rebars** spreadsheet.

The Joist Type

A *Joist* type of continuous system can be assigned to a T section, a V (filled) section or others, if it acts like a joist, as defined by the code. Joists concrete cover requirements are different from a standard beam. This type of continuous system is not automatically generated.

The design must be done according to a bonding interaction only, as for standard beams.

The Shear Wall Type

This type of continuous system is specific to the design of shear walls.

When plates are grouped together as "Section" type of plates, VisualDesign creates a continuous system that is composed of a fictitious column and does the design according to data included in the **Shear Wall** tab of the **Concrete Specifications** spreadsheet.

See also
Shear Wall tab (Concrete Specifications)
Modeling and Designing a Shear Wall
Ductility for Shear Walls
Groups of plates – Shear Walls

Summary - Design of Continuous Systems

Continuous System			Design		
Type	**Continuous Non rectilinear Rebars (1)**	**Interaction**	**Longitudinal Rebars**	**Stirrups (2)**	**Example**
Beam/Column (5)	No	Bending	Number	Yes or No	Beam with or without stirrups
Beam/Column	No	Bending / Compression	Number	Yes	Column
Beam/Column	Yes	Bending	Number	Yes or No	Beam with or without stirrups (3)
Beam-slab	No	Bending	/m of width	No	Typical slab or footing
Beam-slab	No	Bending / Compression	/m of width	No	Slab without stirrups
Beam-slab	Yes	Bending only or Bending / Compression	/m of width	No	Culvert walls (4) or typical footing
Shear Wall	No	Bending / Compression	Number	N/a	Shear Wall
Joist (5)	No	Bending	Number	Yes or No	Joist with or without stirrups

NOTES:

(1) This option is available in the **Concrete Design** tab of **Member Characteristics** dialog box.

(2) Stirrup design is specified in the concrete specifications.

(3) Refer to Generation of Abutments, Piles, and Retaining Walls module.

(4) Refer to Generation of Culverts module.

(5) The difference between the *Beam* (of Beam/Column type) and the *Joist* type is the concrete cover requirement.

Design of Beams and Columns

Analysis and Concrete Design

The "Analysis and Design" Icon of Tools toolbar

Press this icon to run an analysis and a design of concrete beams, columns, shear walls and 2-way slabs or select the function **Analysis and Design** under the **Analysis** menu.

If your structure is composed of steel beams and/or columns and contains concrete elements, the function first analyses all the structure and keeps the records (forces and displacements) in memory. Then, the program designs steel members according to the selected steel specifications. Finally, it designs concrete elements according to concrete specifications.

When the steel design is done, the "Design Results" activation mode will be activated to let you look at the results through the Steel Design Results spreadsheet, diagrams and design brief. In the same manner, when the concrete design will be done, the "Rebar Placement" activation mode will be activated. The *Rebar Placement* window is open by double-click a continuous system.

Beams, columns and shear walls

In the *Rebar Placement* window, **View Options** allow displaying force and resistance diagrams over the elevation view of the current continuous system. Placement of rebars is editable and resistances are calculated automatically. Then, if you want to look at numerical results, call up the **General Results** spreadsheet through the **Results** menu of the *Rebar Placement* window.

If you modified dimensions, do not forget to re-run an analysis and a design because the distribution of moment in your structure will be different.

Design of 2-way Slabs

The analysis and design function is also used to design 2-way slabs. Results are available through the **FE Results** tab of **View Options** dialog box. A bar list is supplied in the **Results / Bar List** menu.

Prestressed Concrete:

VisualDesign verifies and designs the prestressed element according to the cables that were placed by users.

Date and Time

VisualDesign now displays the date of analysis and design in the **Design** 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

Procedure - Concrete Design for Beams and Columns

1. Project Configuration – Concrete Design tab

- Select the **Project Configuration** dialog box from the **File** menu and complete the **Concrete Design** tab.

2. Concrete Specification and Selection of Rebars

- Select the **Concrete Specifications** spreadsheet **Structure** menu. Select a building code that will be applied to design concrete elements and modify default parameters, if needed. By default, some longitudinal and transverse rebars and stirrups are chosen for the design. To add or withdraw type of rebars, double-click in the cell.

3. Structure Modeling

4. Defining Concrete Members

- Select members and press the **Properties** icon. In the **Member** tab of the **Member Characteristics** dialog box, choose a concrete section in the "Shape" selection tree and choose a concrete material. Activate design criteria and continuous systems will be automatically created.

- Select the **Connection** tab and define rigid extensions at member end. (Or, select the tool **Calculation of Rigid Extensions** in the **Structure**/**Tools** menu.) Make sure that eccentricities ex and ey are the same for a given member. N. B. The *ez* rigid extensions may be positive or negative. Check the member local axis system. Align elements: eccentricity *ey* represents the axis passing through the element.

- Complete the **Concrete Design** tab.

5. Continuous Systems spreadsheet

- In the "Specification" column of **Continuous Systems** spreadsheet (**Structure** menu), choose the concrete specification that will be applied for design or verification. Define the type of exposure and concrete cover. Enter the cracking variable.

6. Define Loads, Load Combinations and Envelopes

If you are placing rebars yourself and checking forces and resistances diagrams on the screen, go to step 9b.

7. Launch Analysis and Design

8. Basic Results

- Go to **Results / Structure Design / Concrete** (in VisualDesign main window) and look at calculated percentage of design loads for all continuous systems in your project.

9. Open the Rebar Placement Window

- Activate the "Rebar Placement" mode on Activation toolbar and double-click on any continuous system. This will automatically open the *Rebar Placement* window.

- Select the **View Options** dialog box. In the *Rebar Placement* tab, check the boxes corresponding to the diagrams (results) that you want to display on your screen. They will be displayed above the beams elevation view (continuous system). Check also the roots *Dimensions* and *General* in the selection tree.

- Create cross-sections: select the **Cross-sections** spreadsheet in the **Rebar Placement** menu or use the tool **Automatic Generation of Cross-sections**.

- To display and check 3D interaction curves for columns, you must create cross-sections first. Then, check the "Mrx, Mry vs. Nz (3D)" box in the *Rebar Placement* tab of **View Options** dialog box. Zoom in by using functions **Zoom window** and **Zoom +**. To look at points included in a 3D interaction curve, click once on the curve and rotate it by using the keyboard arrows. To mask one or more interaction curves, select the **Cross-sections** spreadsheet and select "Yes" in the column "Mask interaction".

- Select the **General Results** spreadsheet under **Results** menu.

- To know the amount of reinforcement steel for the displayed continuous system, select "Bar List" in the **Rebar Placement** menu.

- To know the amount of reinforcement steel for the whole project, close the *Rebar Placement* window and go to **Results / Bill of Materials / Bar List**.

10a. Editing Rebars after a Design

- To modify rebars after a design, activate the **Cursor** mode. Click on a longitudinal or transverse rebar, on the elevation view or in a cross-section. Use the **Stretch** and **Move** function. Use the red target that will appear on your screen to help you to accurately move a rebar. The displayed diagrams (forces, resistances and deflection) will be automatically recalculated as you change and modify rebars.

- If you changed beams and columns dimensions, do not forget to launch a design again.

10b.   Placing Rebars and checking your Design

- Choose a concrete specification that is of the "Verification" type, when defining continuous systems (spreadsheet).

- Activate the "Rebar Placement" mode on Activation toolbar and double click on a continuous system to open the *Rebar Placement* window.

- Select the **View Options** dialog box. In the *Rebar Placement* tab, check the boxes corresponding to the diagrams (results) that you want to display on your screen. They will be displayed above the beams elevation view (continuous system). Check also the *Dimensions* and *General* roots in the selection tree. The displayed diagrams (forces, resistances and deflection) will be automatically recalculated once that rebars will be placed in the continuous system.

- Create cross-sections to help you placing rebars: select the **Cross-sections** spreadsheet in the **Rebar Placement** menu or use the tool **Automatic Generation of Cross-sections**.

- Select the **Longitudinal Rebars** spreadsheet in **Rebar Placement** menu and define the rebars (number, bend shape, location in the continuous system, etc.).

- Select the **Transverse Rebars** spreadsheet in **Rebar Placement** menu and define the stirrups (number, dimensions, shape, sequences, patterns, etc.).

- To edit rebar placement on your screen, see stage 9a.

- To look at various numerical results, select the **General Results** spreadsheet in the **Results** menu.

Design of 2-Way Slabs

Analysis and Design

The "Analysis and design" Icon of Tools toolbar

VisualDesign designs 2-way slabs composed of finite elements (triangular and/or rectangular plates). Openings are allowed. Rebars are placed according to obtained stress/force contours.

The Slab and Mesh Generator

The **Slab & Mesh Generator** is a powerful tool that creates the slab surface from selected nodes on the slab outline and meshes the slab with triangular plates. This tool is located in **Structure / Generator** / **FE Slab**. Any geometry can be created and meshed if selected nodes are coplanar. VisualDesign uses the Convex Hull, which is a mathematical concept, to create and mesh the surface.

The user specifies that maximum area of plates, the material and thickness along with direction vectors. The latter parameters are required because VisualDesign creates a group of plates for this generated slab. A specification and concrete covers must be specified for each slab in the **Group of Plates – Surfaces** spreadsheet, for each slab to be designed. Refer to Groups of plates - Surfaces.

Graphic Results

Layer of bars (Mx+, Mx-, My+, My-) can be displayed, at the user's choice, through the **FE Results** tab of **View Options** dialog box. A different colour is assigned to reinforcing bars in accordance with the size, and the pen weight varies with the size for a better display. The colour of rebars can be modified and disabled in respective rebars spreadsheets. Rebar Placement can be directly printed using the **Print / Graphic** function in **File** menu and it can be imported as a DXF file.

Numerical Results

Bar lists are available in the **Results / Bar lists** menu, for each group of plates representing a 2-way slab.

See also
The Slab & Mesh Generator
Groups of plates - Surfaces
Concrete Specification - The FE Slab tab
Limitations for 2-way slab design
Procedure for Designing 2-Way Slabs
Bar List – 2-Way Slabs

FE Results of View Options

Limitations for 2-Way Slab Design

For the moment, calculated reinforcing bars are not editable. This limitation includes rebar lengths, spacing and size.

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 nodes located on the slab outline and members (optional). The user specifies the maximum areas of plates, the material and thickness along with vectors to orient finite elements.

Specified Common
Local Axes
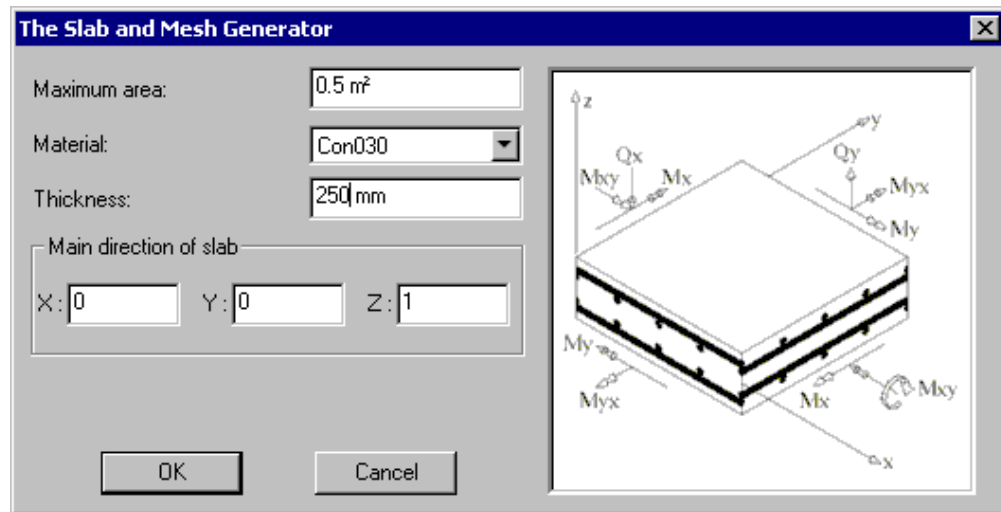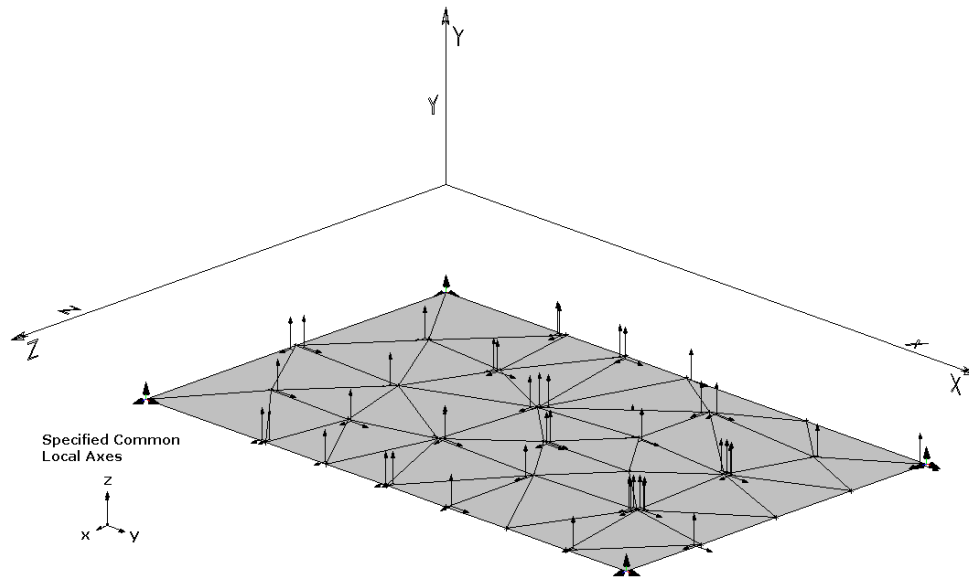
Orientation vectors are automatically initialized in the **Groups of plates –
Surfaces** spreadsheet, which is called up when the slab is created. The
direction of vectors depends on the order of selected nodes (clockwise or
counterclockwise).

	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	1.52	5.00	1.52	0.00	0.00	1.00	1.00	0.00	0.00

Groups of Plates Spreadsheet - Surfaces

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.

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.

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.
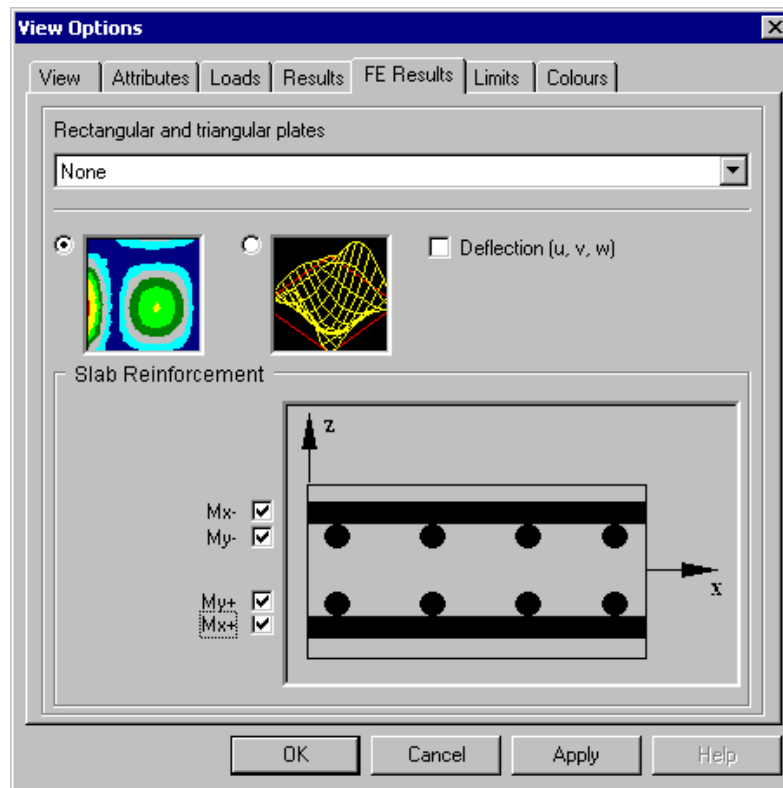
Force/Stress Contours

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
Groups of Plates - Surfaces
Scaling for Intervals
View Options
Interpreting Plates Analysis Results
Concrete Design of 2-Way Slabs

Bar List - 2-Way Slab Design

VisualDesign supplies the bar list for designed slabs. Total lengths and weights are calculated for each designed slab using the name of each group of plates (surfaces).

To reach this spreadsheet, go to **Results / Bar List / Groups of plates – Surfaces**.

Group: 2-Way Slab Design (title)

Column	Description	Editing
ID	Automatically calculated	No
Number	Name or number of the group of plates that was assigned to the designed slab.	No
Layer of Rebar	Layer of rebar: Mx+, Mx-, My+, My-.	No
Material	Rebar steel grade	No

Column	Description	Editing
Reinforcement	Type of rebar in this layer.	No
Quantity	Number of bars of this type in this layer	No
Surface density	Weight of this type of rebar over the total slab area.	No
Total weight	Total weight of this type of rebar in this layer.	No
Total length	Total length of this type of rebar in this layer.	No
Number of hooks	Number of hooks in this layer. Hooks are placed to obtain continuity with columns or walls, if present in the model.	No
Hooks A	Length A of hooks in this layer. See the image below.	No
Hooks B	Length B of hooks in this layer. This length is corresponding to the rebar development length + 300mm. See the image below	No

Layout of Rebars
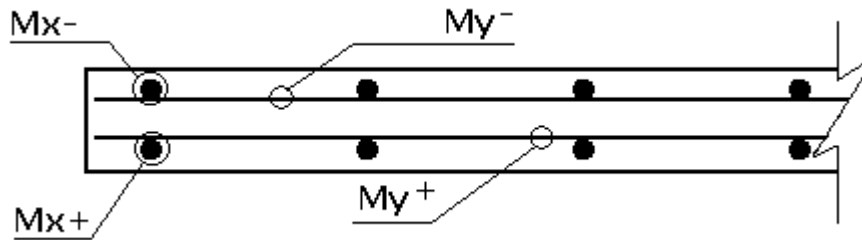
Hooks
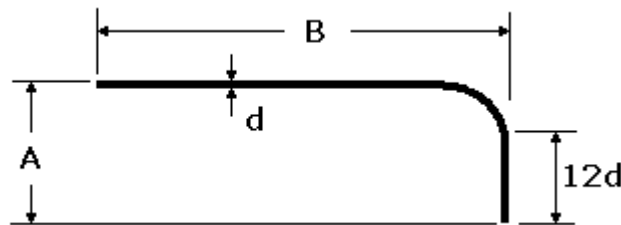
Procedure for Modeling and Designing 2-Way Slabs

REMARKS

- Model the structure but do not model plates for 2-way floors or walls.

- The meshing is done from selected coplanar nodes and members, if required.

- Openings: Model members that surround openings. The mesh generator will not generate plates that go through members. However, plates will be generated over the opening. They can be deleted afterwards

- If the slab has an irregular geometry, select nodes according to the geometry.

Procedure

- Activate the Structure mode.

- Open the Concrete Specifications spreadsheet (**Structure / Specifications / Concrete**):

  - Select the **FE Slab** tab and choose temperature reinforcement for the design of the slab.

- Select coplanar nodes and members, if required, and call up the Slab & FE Generator (**Structure / Generator / Slab & FE**).

  - Specify the maximum area of plates, the material and the thickness.

  - Specify the slab main direction, according to global axis system. This main direction must coincide with the main layers of rebar (layer 1 and 4). This data is used by VisualDesign to initialize the direction vectors for plates, in the **Groups of Plates – Surfaces** spreadsheet.

  - Click OK.

  - The **Groups of Plates – Surfaces** spreadsheet opens on screen. Verify if the direction vectors are OK. Select a concrete specification and specify the concrete covers. Close the spreadsheet.

- Use the generator for each slab to be created and meshed.

- Delete extra plates, if required.

- Align selected plates in the **Plate Characteristics** dialog box.

- Launch the concrete design by clicking this icon .

- Activate the Rebar Placement mode and select the name of a group of plates on Activation toolbar.

- Display graphic results through the **FE Results** tab of **View Options** dialog box.

  - Activate layers of rebars (Mx+, Mx-, My+ and/or My-) to visualize them. Disable the display of plate surface and outline through the **View** tab.

  - Activate a load combination and look at stress/force contours for a selected force or stress diagram.

  - Activate the "mesh" radio button and the *Deflection* box and select a force or stress diagram. Disable the display of nodes to have a better look.

- The colours of rebars can be modified in respective reinforcement spreadsheet.

- Print the rebar placement using the **Print Graphic** function of **File** menu.

- Import the rebar placement as a DXF file through the **File / Import** menu.

- Open the bar list (Results / Bar List / Slab).

N. B. Rebar sizes, lengths and spacing are not editable.

Seismic Design

Ductility of a (local) Lateral-Force-Resisting-System

Ref.: *Concrete Design Handbook*, CSA Standard A23.3-94, Clause 21 "Special Provisions for Seismic Design".

Definitions:

Base of Structure: Level at which earthquake motions are assumed to be imparted to a building. This level does not necessarily coincide with the ground level.

Critical Section: Section where a plastic hinge may start to form under earthquake loading.

Plastic Hinge: Region of a member where inelastic flexural curvatures occur. Its length extends at least *d* from a critical section.

Factored Load Effect: Effect of factored load combinations as specified in Clause 8.3 which include earthquake load effects as determined in accordance with Clause 4.1.9 of the *National Building Code of Canada 1995*.

Nominal Resistance: Resistance of a section calculated using axial loads Ps and Pn where applicable and the specified values of f'c, fy with φc and φs taken as 1.0.

Ps: Axial force at section resulting from specified dead load plus specified live load.

Pn: The earthquake-induced transfer force resulting from interaction between elements of a linked or coupled wall system and shall be taken as the sum of the end shears corresponding to the nominal flexural resistance in the coupling beams above the section.

Pp: The earthquake-induced transfer force resulting from interaction between elements of a linked or coupled wall system and shall be taken as the sum of the end shears corresponding to the probable flexural resistance in the coupling beams above the section.

Probable Moment Resistance: Moment resistance of a section calculated using axial loads Ps and Pp where applicable and 1.25fy as the stress in the tension reinforcing and the specified values of f'c and fs, with φc and φs taken as 1.0.

Ductile Coupled Wall: Ductile flexural wall system with coupling beams where at least 66% of the base overturning moment resisted by the wall system is carried by axial tension and compression forces resulting from shear in the coupling beams. This structural system qualifies for a force modification factor, R, of 4.0 in the *National Building Code of Canada*.

Ductile Partially Coupled Wall: Ductile flexural wall system with coupling beams where less than 66% of the base overturning moment resisted by the wall system is carried by axial tension and compression forces resulting from shears in the coupling beams. This structural system qualifies for a force modification factor, R, of 3.5 in the *National Building Code of Canada*.

Ductile Flexural Wall: Structural wall that resists seismic forces and dissipates energy through flexural yielding at one or more plastic hinges. This structural system qualifies for a force modification factor, R, of 3.5 in the *National Building Code of Canada*.

Hoop: Closed tie or continuously wound tie. A closed tie can be made up of several reinforcing elements with seismic hooks at each end. A continuously wound tie shall also have seismic hooks at each end.

Seismic Hook: Hook with at least a 135° bend with a six-bar-diameter extension (but not less than 100 mm) that engages the longitudinal reinforcement and is anchored in the confined core.

Seismic Cross Tie: Single bar having a seismic hook at one end and a hook not less than 90° with at least a six-bar-diameter extension at the other end. Hooks shall engage peripheral longitudinal bars. The 90° hooks of successive crossties engaging the same longitudinal bar shall be alternated end for end.

Summary

Objectives of Clause 21 (Standard A23.3-94)

R-values vary from 1.5 to 4.0 and are chosen according to the capacity of a system to absorb energy. Clause 21 applies to structures designed using a force modification factor, R greater than 1.5.

"Clause 21 contains special requirements for the design of reinforced concrete members of a structure for which the design forces, related to earthquake motions, have been determined on the basis of energy dissipation in the non-linear range of response."

"These structures shall be the subject of capacity design: energy-dissipating elements or mechanisms are chosen and suitably designed and detailed, and all other structural elements are then provided with sufficient reserve capacity to ensure that the chosen energy-dissipating mechanisms are maintained throughout the deformations that may occur."

So: Factored Resistance = Effect of factored loads (EFL)

"The structure is expected to go well into the inelastic range to absorb the energy from the earthquake event: The expected loads exceed the capacity of some members."

Capacity Design

"To avoid having to design all of the structure to be ductile in all of the possible failure modes the capacity design approach is adopted. Only some elements of the building and locations within these elements which will absorb the energy are designed with the necessary ductility."

"The capacity of the selected ductile energy absorber or absorbers is determined and then the non-ductile failure modes and non-yielding locations within the selected elements are designed for capacities higher than the effects of yield of the energy absorbing elements."

So:

Factored Resistance of Non-energy Absorbers	$\geqslant$	Effect of Development of Nominal or Probable Resistance of Energy Absorbers

"In addition the rest of the building considered to be not part of the lateral force resisting system must be capable of sustaining the deformations which will occur during the energy absorption process without failure and this is covered in Clause 21.8."

"This clause [21.8] is intended to give guidelines to help ensure that the parts of the structural system designed for gravity loading only will continue to function at the lateral displacements of the frame expected during the earthquake."

Some of General Requirements (see clause 21.2.):

"The linear and non-linear behaviour and interaction of all structural and non-structural members, which materially affect the response of the structure to earthquake motions, shall be considered in the analysis."

"The forces in the various components of a lateral load resisting system are usually determined by a linear elastic analysis using code equivalent static loads. The section properties used for the elastic model are modified gross section properties. Ex.: For beams that are part of a frame, the effective inertia, Ie, is equal to 0.4*Ig (See the table in the commentary N21.2.2.1.)"

- "The specified compressive strength, f'c, used in design shall not exceed 55 MPa." (Exceeding this limit, the concrete becomes brittle and cannot be considered as a ductile material.)

- Foundation: "The factored resistance of the foundation system and supports of frames or walls, or both, shall be sufficient to develop the nominal moment capacity of the frames or walls and the corresponding shears. Where the factored moment resistance of any wall or any frame significantly exceeds the required factored moment, the factored resistance of unanchored footings supporting those walls or frames need not exceed the maximum factored load effects determined with loads calculated using R equal to 1.3. Where frames or walls are supported by anchored footings or elements other than foundations, the factored resistance of these elements need not exceed the maximum factored load effects determined with loads calculated using R equal to 1.0."

- "Reinforcement resisting earthquake-induced forces in frame members and walls in lateral load resisting systems designed with force modification factors R greater than 2.0 shall be weldable grade in conformance with CSA Standard G30.18. Lateral load resisting systems designed with force modification factor of 2.0 or less shall comply with CSA Standard G30.18 but need not be weldable grade."

- "The reduction of lap splice length permitted by Clause 12 where the area of reinforcing provided exceeds that required shall not be permitted for members conforming to Clause 21." "Lateral force resisting elements are expected to develop their yield capacity and deform plastically under the action of seismic forces. Under these circumstances all reinforcement provided is expected to yield and must therefore be spliced for full tension capacity."

Ductile Frame Members Subjected to Flexure (for R greater than 1.5) (Clause 21.3)

"A ductile moment resisting frame must be capable of sustaining a series of oscillations into the inelastic range of response without critical decay in strength."

"When a plastic hinge region is deliberately relocated away from the column then hoop reinforcement must be provided within and adjacent to the plastic hinge region."

Ductile Frame Members Subjected to Flexure and Axial Load (for R greater than 1.5) (Clause 21.4)

"The energy dissipation necessary for a multi-storey frame to survive a severe earthquake should in general occur by the formation of ductile plastic hinges in beams. Plastic hinges in beams are capable of tolerating larger rotations than hinges in columns. Further, […] mechanisms involving beam hinges cause energy to be dissipated at many locations throughout the frame. An additional consideration is that extensive hinging in columns may critically reduce the gravity load carrying capacity of the structure."

"To achieve the desired beam hinging mechanism, the Standard specifies the following design approach:

STRONG COLUMN - WEAK BEAM

According to equation (21-1), Clause 21.4.2.2:

$$\Sigma \text{ Mrc greater or equal to } 1.1 \Sigma \text{ Mnb}$$

"Eq. (21-1) requires that the total factored resistance of the columns based on φc=o.6 and φs=0.85, must be at least 10% greater than the total nominal resistance, based on φc and φs = 1.0, of the beams framing into the joint."

"Frame members not satisfying Clause 21.4 shall be designed according to Clause 21.8 and shall not be considered part of the lateral force resisting system."

"Axial design loads in frame columns shall account for beams yielding at levels above the level being considered. The shears from the beams shall be those given by the method of Clause 21.7.2.1 using nominal rather than probable strengths. Allowance may be made for the reduction in accumulated beam shears with increasing numbers of stories."

"It is important to appreciate that during a severe earthquake some column hinging and some yielding of columns will occur even if the "strong column – weak beam" philosophy has been followed. For this reason columns need to be detailed for ductility in accordance with the requirement of Clause 21.4.4.7."

Ductile Flexural Walls (for R greater than 2)   (Art. 21.5):

"One possible approach for determining regions of potential plastic hinging is to compare the distribution of moments over the height of the wall corresponding to the development of the probable moment resistance(s) at locations of known plastic hinging (e.g. at the base of the wall or at abrupt changes of cross-section) with the distribution of the probable moment resistance of the wall over its height. In regions where the resistance considerably exceeds the moment demand it can be assumed that the plastic hinging cannot occur."

See Clause 21.5.7 titled "Ductility" and equation (21-5) to ensure ductility in a plastic hinge region of a wall.

Ductile Coupled and Partially Coupled Walls (Art. 21.5.8)

"Ductile coupled walls are walls with ductile coupling beams (headers) linking individual wall segments where the primary ductility and energy absorption is in the coupling beam plastic hinges. In order to classify as a coupled wall system with an R of 4.0 a majority of the overturning moment must be carried through axial loads in the wall elements induced by vertical shears in the coupling beams. If this is not the case then a significant proportion of the moment is carried by individual bending in the wall elements with an R of 3.5 as a standard ductile wall." (See N21.5.8.1).

"Coupled and partially coupled walls shall be connected by ductile coupling beams. The coupling beam requirements are the same whether a wall is coupled or partially coupled and are selected from two types depending on header proportions and the shear to be carried. The primary system is the diagonally reinforced coupling beam, however if the shears are small and the span to depth ratios large then beams similar to ductile frame beams may be used." (See N21.5.8.2).

"Walls at each end of a coupling beam shall be designed such that the factored moment resistance of the wall calculated about its centroid exceeds the moment at its centroid resulting from the nominal resistance of the coupling beams framing into the wall and the factored moment in the wall. The factored capacity shall be calculated using axial loads Ps and Pn." (Clause 21.5.8.3).

"In fact, […] in order to ensure that the plastic hinges form in the coupling beams (headers), not the walls, the wall at each end of the beam must be stronger than the beams framing into it."

"According to Clause 21.5.8.5, linked and coupled wall systems shall be designed with factored moment resistance greater than the overturning moment corresponding to the development of the nominal moment resistance of the coupling beams above the level under consideration. The factored capacity shall be calculated using axial loads Ps and Pn." (See N21.5.8.5)

See also
Overall Ductility of a Structure for Seismic Analysis According to NBCC 95

Participating Mass for Spectral Analysis

To get an accurate spectral analysis, building codes generally require that the mass participation to displacement of a structure in a given direction, be at least 90% of the structure dead weight plus 25% of the snow load.

To get this percentage, the number of calculated modes can be increased in the modal analysis or the dead weight that is distributed to supports can be modified in such a way to minimize this weight and to maximize the weight acting above supports.

The weight of a member is equally distributed at end nodes. Typically, half of column weight is distributed to support node in the first story of a building, so this weight is lost and is not considered in the participating mass of the structure. To reduce the distributed weight that goes to supports, split the columns at a level of about 0.5m above supports.

Shear Wall

The same principle applies to a shear wall that is composed of plates and bearing on support nodes. It would be interesting to specify the first row of nodes above supports as "Level Nodes" for the design of such wall. Refer to Node Characteristics dialog box.

See also

Calculation of Vf in a Seismic Design

All codes and standards require to check the shear resistance of concrete elements, ductile and with nominal ductility, considering Vf calculated with R=1.

Forces due to accidental torsion effects are also corrected according to this requirement (Vf|torsion calculated with R=1), if necessary.

This requirement applies as much to members as for shear walls.

Note: This correction of Vf applies only to continuous systems that are ductile and of nominal ductility.

For these two cases, the design will be done in order to obtain a value Vr that is greater than the value Vf, **for which a probable or nominal bending moment will develop**.

In VisualDesign, due to the structure of data, we do not know the values of probable and nominal forces when the seismic calculations and combination of forces are done. For this reason, we used a simplified approach, which is conservative, and created a factor R ', allowing us to calculate first the value of Vf using R=1. This factor R ' will multiply the calculated Vf (using a R=1) to obtain probable and nominal forces, which are bigger that Vf (R=1).

R ' is specified in the **Continuous Systems** spreadsheet.

Correcting the contribution of inelastic effects:

When calculating the seismic envelope, we add accidental torsion effects and inelastic effects to the spectrum response:

Forces|seismic = Forces|spectral + Forces|accidental torsion + Forces|inelastic effects

The contribution of displacements due to seismic effects must not be multiply by the factor R. Then, we get:

Displacements| seismic = R x Displacements| spectral + R x Displacements| accidental torsion + Displacements | inelastic effects

See also

Shear Wall

Ductility and Design of Shear Walls

The following flow charts represent the CAN/CSA-A23.3 code provisions that the software uses to design shear walls considering three ductility: Not ductile, nominal ductility, and ductile.

Shear Wall: Not Ductile
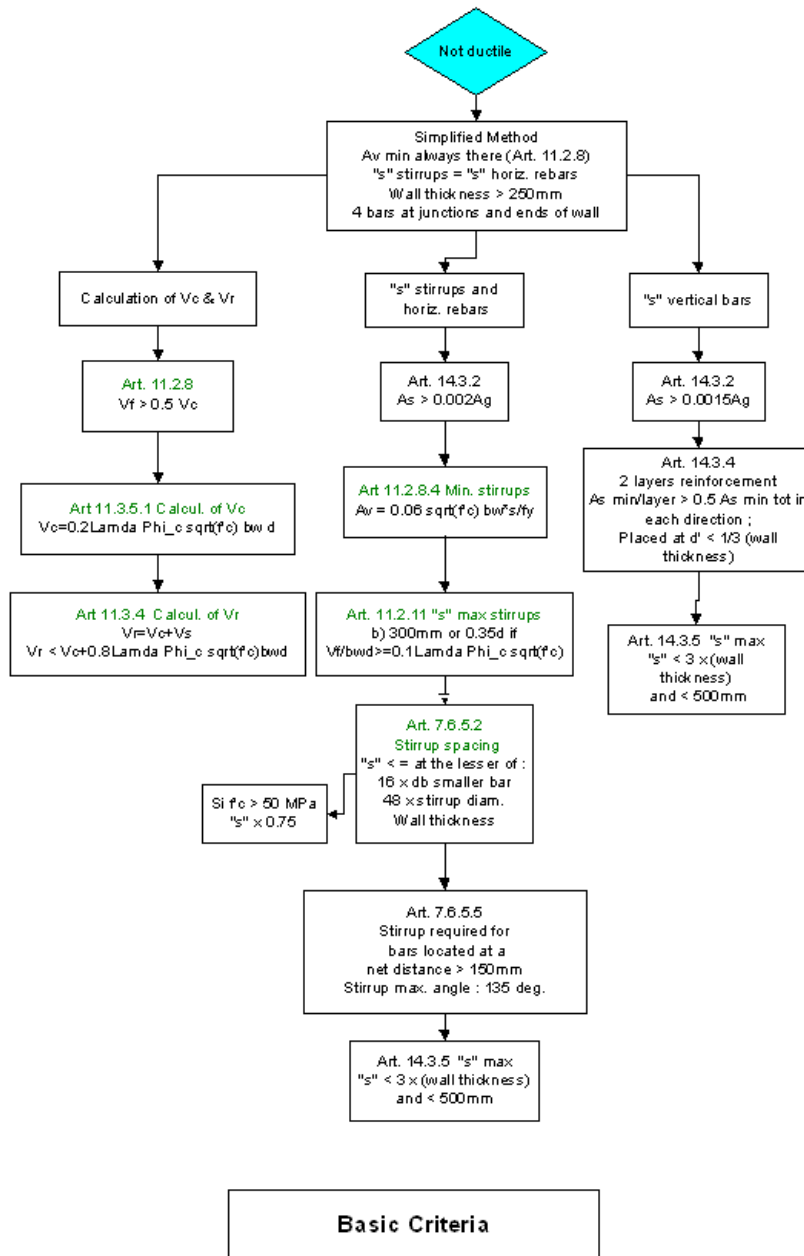
Shear Wall: Nominal Ductility

Shear Wall: Ductile

```
                          ╱╲
                         ╱   ╲
                        ╱ Ductile╲
                        ╲        ╱
                         ╲      ╱
                          ╲    ╱

                    Refer to Basic Criteria
                       and art. 21.5

   Calcul. of Vc & Vr      "s" stirrups and        "s" vertical bars
                            horiz. bars

     Art.21.7.2.3          Art. 21.5.5            Art. 21.5.5
     Calcul. of Vf         Rho min = 0.0025       Rho min = 0.0025
                           s max = 450mm          s max = 450mm

      Use Vf min           Art. 21.5.6.5          Art. 21.5.4
                           "s" stirrup            lap splice: 1.5xld
                           in conc. zones :       rho max = 0.06
                           See Art. 7.6           db < 1/10 (wall thickness)

  Art21.7.2.3 a)      Art.21.7.2.3 b)
  Vf calculated with R'   Vf calculated with
  where R' = Mp / Mf      R=1.0                   Art. 21.5.6.3
                                                  As min = 0.001 bwlw at
                           Art. 21.5.6.5 Hinge :  each corner
                           "s" stirrup < the smaller of :
                              6x db long. rebar
      Plastic Hinge    No      24x tie diam.      Art. 21.5.6.4 Hinge:
                               1/2 (wall thickness) As min = 0.002 bwlw at
   Yes                      or art. 21.5.7 if applic. each corner

    Art. N21.5.2.1
    Location of plastic                           Art. 21.5.7 Ductility
    hinge at Mrp +/-:                             For column:
         lw                                       rho_v min = 0.005
       > hw / 6      Art. 21.7.3. Vf due to seismic forces
       < 2 lw        Design according to
                     Clause 11, except :
                     Art. 21.7.3.2 Walls

  Art. 21.5.5.3         Art. 21.7.3.2
  2 curtains of reinforcement   a)Where vert. reinforc. is expected
  are required where               to yield : Vc = 0
  Vf > 0.2 Lamda Phi_c sqrt(f'c) Acv  b) Ductile walls: epsilon v = 0.015
                             c) Outside hinge = Cl. 11
                             d) dv = Cl. 11 but > 0.8 lw
                          e) vf < 0.15 phi_c f'c if eps_v <= 0.005
                           If eps_v > 0.005, vf= table 21-1.
                          f) Vert. reinforc. in hinge must extend
                          beyond a distance where yielding will
                            occur: distance d eff + ld
                             g) If hw/lw< 2.0 :
                            Rho_v >=(1+ 2Vc/Vs) * Rho_h
```

See also

Continuous Systems spreadsheet

Calculation of Required Transverse Reinforcement in Shear Wall

This topic explains our approach for calculating the required transverse reinforcement for shear resistance of a shear wall.

Levels:

VisualDesign automatically recognizes levels in a shear wall if horizontal elements are attached to the wall, such as horizontal plates and floors. However, if there are openings in the wall but no horizontal elements, the user must defined levels at the top and bottom of openings if he wants to obtain forces and particular designs at these levels. A sole node is required to define a level. Go to the **Node Characteristics** dialog box (by double-clicking on a node in the Structure activation mode) and activate the option "Level- Shear Wall".

A shear wall model could look like the image below, once that particular levels are all well defined by the user:
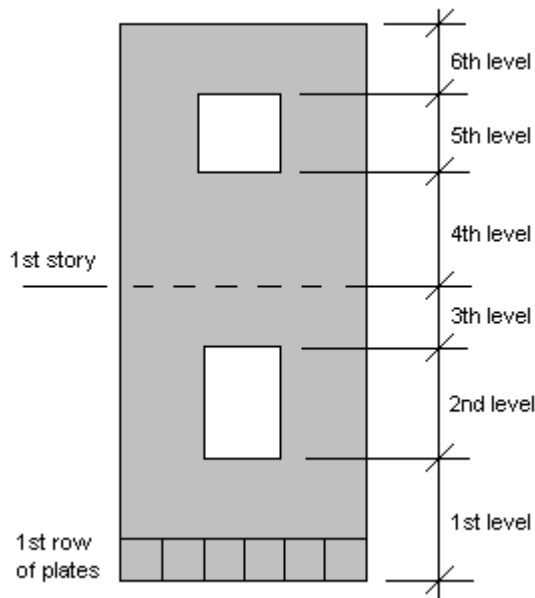
Calculation Method

For all Shear Walls (Ductile and Not ductile):

VisualDesign considers all plates located on a same level and calculates the average shear force, Vxy, per meter of wall length. As shown on the image below, VisualDesign considers each plate dimensions using a thickness "b" that corresponds to the wall thickness, and a width "d" equals to 1 meter. Therefore, the software must obtain Vf < Vr for all load combinations. From this calculated value, VisualDesign can place the required transverse reinforcement and spacing as per code provisions 14.3.3, and 21.5.5 (A23.3-94 standard).

The calculated spacing will be uniform for one level. It can vary from one level to another.

Studying the First Level
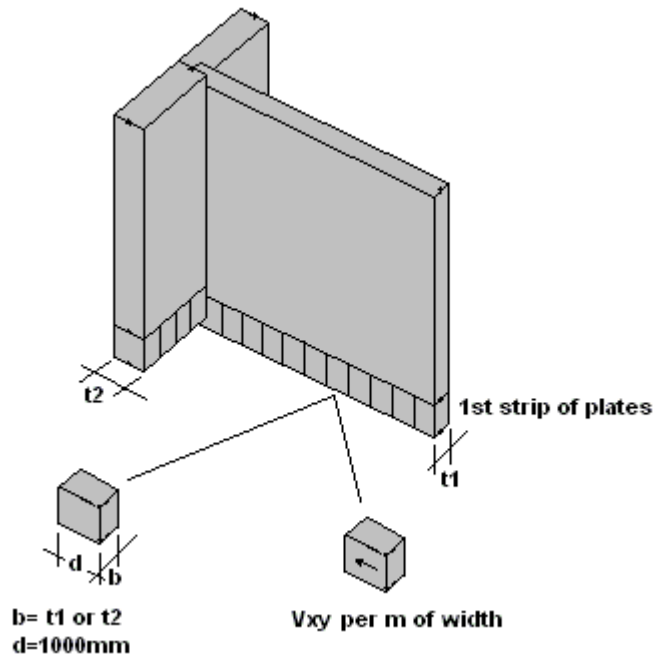
This calculation is conservative because VisualDesign considers the worst shear force along the wall, even if it varies along the length of the wall, for a particular level.

IMPORTANT Critical values that were used for the design are written in the **General Results** spreadsheet in the form of kN/meter of width.

If a shear wall is composed of walls having different thickness, as shown here, VisualDesign will take the worst design of each wall. The worst spacing will be chosen and apply to the considered level.
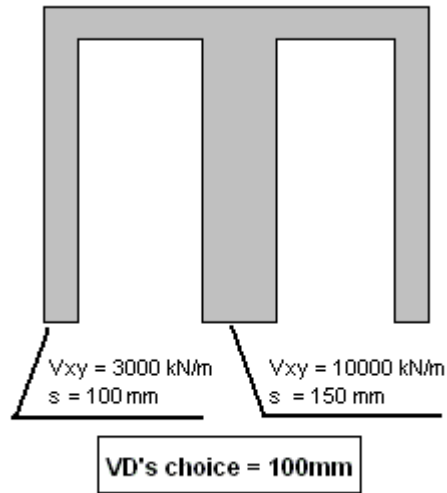
Ductile Shear Wall (R>2.0):

The location of the plastic hinge is based on the (A23.3-94) code provision 21.5.2.1. When the plastic hinge is located within a level, VisualDesign will consider all of this level as a plastic hinge and use the required rebars spacing as per code.

VisualDesign does not consider the case where rigid horizontal elements can create a plastic hinge in a shear wall.
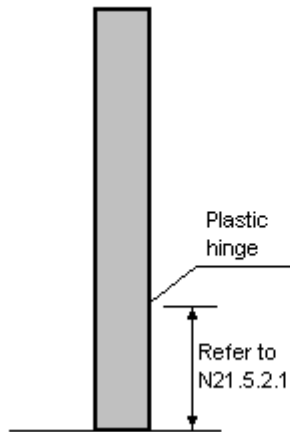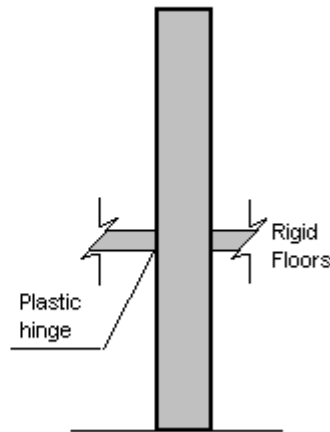
See also

The *Shear Wall* type of Continuous System
Modeling and Designing a Shear Wall
Concrete Specifications for a Shear Wall

Shear Wall and Temperature Effects

When temperature loads are applied to plates that are composing a shear wall, considerable shear forces will be present in plates located near supports, if they are blocked. To get round this problem, create a support node at the centroid of the shear wall; link all the base nodes to this support using very big members; Use the same material as the shear wall and apply temperature loads on these members.

See also
Procedures - Shear Wall
Temperature Loads on Rectangular Plates

Procedure to Model and Design Shear Walls

- Create the model with plates and split them with the **Split** function;

    - If you plan to run a dynamic analysis of the wall, specify one node above supports as a "Level - Seismic" (in the **Node Characteristics** dialog box) to increase the participating mass acting above them.

    - If openings are present in the shear wall, select a node above and below the opening and specify these nodes as "Level – Shear wall" in the **Node Characteristics** dialog box, to get results at these particular levels. You can use this option when no horizontal element is linked to the wall because VisualDesign automatically detects levels with those elements.

    - Model supports at the base of the wall.

- Orient the groups of finite elements using the **Groups of Plates – Shear Walls** spreadsheet. Usually, plates that are modeled in the same plane are grouped together (a footing and a vertical wall should be in different groups). VisualDesign will automatically create a continuous system, which is a fictitious column.

- Apply loads.

- Use the **Load Combination Generator** to define load combinations, and include the spectral envelopes.

- Go to the **Concrete Specifications** spreadsheet and fill in the required design parameters in the **Shear Wall** tab. Specify the minimum and maximum spacing of rebars in concentration zones.

- Open the **Continuous Systems** spreadsheet and assign the concrete specification to the shear wall.

- Define two main seismic directions (orthogonal) in the **Linear Seismic Directions** spreadsheet and fill in the required parameters.

- Launch the reinforced concrete design (**Analysis and Design** function). The modal and spectral analysis will be automatically launched during the design process.

- Double-click the fictitious column, representing the vertical continuous system, to open the *Rebar Placement* window.

- Generate cross sections.

- Open the **View Options** and check the *Dimensions* root. Open the *Column Diagrams* root and select forces and resistances diagrams.

Particularities:

VisualDesign does not design (sections) the shear wall itself. It designs reinforcement. You must model the shear wall with rectangular plates (finite elements) and then, the program will place reinforcing bars within according to loads and load combinations.

VisualDesign looks for horizontal elements (levels or stories) by searching for floors or horizontal plates that may be linked to the model. If there are no horizontal elements in the shear wall, you can specify levels for VisualDesign, and obtain results at these levels.

Specify a level or story in the shear wall

Select one node at a specific level and go to the **Node Characteristics** dialog box. Activate the option "Level – Shear wall".

Placing Rebars:

VisualDesign places main reinforcement in the ends of wall. For the design of a concrete cage, as for elevators, main reinforcement will be placed in corners. Secondary reinforcing bars are also supplied.

See also

Calculation of Required Transverse Reinforcement in Shear Wall
Design of Concrete Plates
Groups of Plates
Participating Mass
Node Characteristics Dialog Box

Reinforcement and Cross-sections

Main Reinforcement Spreadsheet

This spreadsheet describes the main reinforcement for the displayed continuous system, in *Rebar Placement* window. It can be accessed from elevation view by double clicking on a longitudinal rebar or by selecting many and clicking on function **Properties**.

If the option "Continuous non rectilinear reinforcement for a variable geometry" of **Concrete Design** tab has been activated for a continuous system with a variable geometry, VisualDesign will build a polyline with each member end nodes. Consequently, longitudinal rebars are put end to end and form a sole object. You will find nodes that are composing this polyline in the new spreadsheet entitled Position. It will appear next to the **Main Reinforcement** spreadsheet.

Group: Continuous System (title)

Column	Description	Editing
ID	Rebar ID, automatically calculated.	No
Number	Rebar number (16 alphanumerical characters).	Single click
Bar size	Type of main rebar.	Double-click
Bending Shape	Rebar bending shape.	Double-click
Flip Horiz.	Choose [x] to flip the rebar horizontally (mirror).	Double-click or Space bar
Flip Vertical	Choose [x] to flip the rebar vertically (mirror).	Double-click or Space bar
No. Rebars	Number of rebars in this layer for a standard continuous system (no variable height).	Single click
X start	Distance from outside diameter of the bar located at the far left (in the cross-section) to continuous system axis. See the cross-section below.	Single click
X end	Distance from outside diameter of the bar located at the far right (in the cross-section) to continuous system axis. See the cross-section below.	Single click
zi	In the figure shown below: Position of the bar relatively to origin of continuous system axis (from rebar outside diameter).	Single click

Column	Description	Editing
zj	In the figure shown below: Position of the bar cut-off, relatively to the same continuous system axis origin.	Single click
yi	Distance from centre of gravity of bars to continuous system axis, at the beginning of the section having a variable height.	Single click
yj	Distance from centre of gravity of bars to continuous system axis, at the end of the section having a variable height.	Single click
Left End	Left end of main rebar: No hook, With hook, Continuous or With sleeve.	Double-click
Right End	Right end of main rebar: No hook, With hook, Continuous or With sleeve.	Double-click
Column only:		
Type of Reinforcement	This column only appears if the continuous system is a column. Choose a "Standard" or "Circular" type of reinforcement.	Double-click
Radius	Radius used to place rebar in a circular or square column.	Single click
Start α_o	Initial angle used to place bars. This angle is relative to the local x-axis of the continuous system.	Single click
Gap in lap splice	Gap between two spliced bars where one of the rebar is bent.	Single click
Rotation β	Rotation angle of the bar, relative to the local x-axis of continuous system.	Single click
1st bend	Distance from rebar origin to the first bend.	Single click
2nd bend	Distance from rebar origin to the second bend	Single click
Bridge Evaluation module only		
Deterioration	**Bridge Evaluation Module only**: Enter the percentage of deterioration for this rebar. (0% represents no deterioration.)	Single click

Elevation of a continuous system:
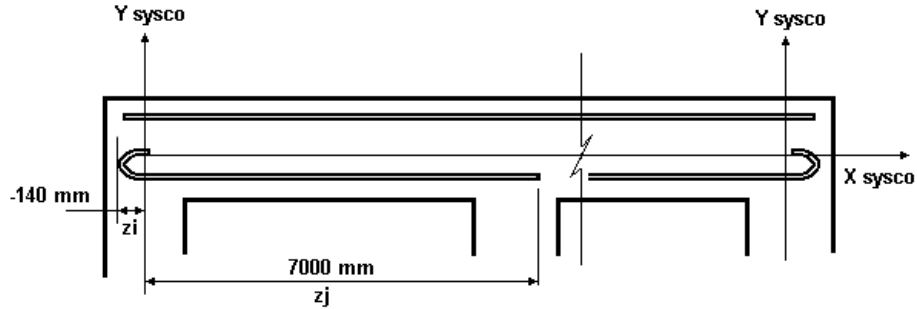
Beam cross-section:
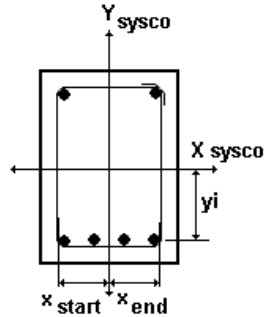
The yi distance is taken from the centre of gravity of rebars to continuous system axis.

The *x start* and *x en*d distances are measured from the rebar outside diameter (located at the ends of the bar layer) to the continuous system axis.

Algebraic sign!!! The distance *x start* will be negative if located left of the continuous system Y-local axis, as you can see on the figure below. The distance "yi" will also be negative.

Warning!!! The continuous system axis does not necessarily correspond to the member local axis system.
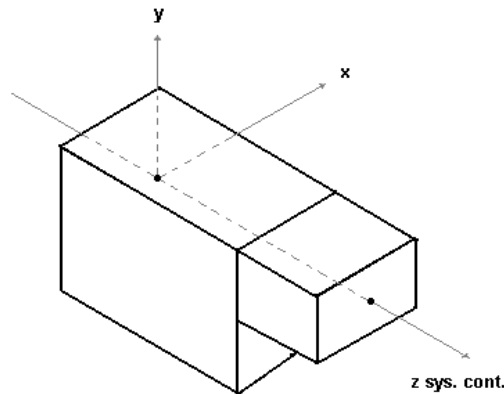
Note. VisualDesign™ will place default rebars (to provide support to the placement of stirrups) in the beam corners (top or bottom) if none have been calculated during the design of the continuous system (because none was needed). To specify the size default rebars, select the **Concrete Design** tab of **Project Configuration**.

See also

Rebar Placement Menu
Design of a continuous system
The Position tab (Longitudinal Rebars)
Rebars bending shapes
Cross-sections
Deterioration of rebars and cables

The Position tab (Main Reinforcement spreadsheet)

This spreadsheet appears next to the **Main reinforcement** spreadsheet if option "Continuous non rectilinear reinforcement for a variable geometry" has been activated in the **Concrete Design** tab of **Member Characteristics** dialog box, for a continuous system with a variable geometry (Beam-slab or Beam column types only).

VisualDesign will build a polyline with each member end nodes. Consequently, longitudinal rebars are put end to end and form a sole object. You will find nodes that are composing this polyline in this new spreadsheet, which appears next to the **Main reinforcement** spreadsheet.

Group: Continuous System (title)

Column	Description	Editing
ID	Automatically calculated	No
y	y-coordinate of node according to continuous system local axes.	Single click
zj	z-coordinate of node according to continuous system local axes.	Single click
End of Rebar	Description of the main rebar end: No hook, With hook, Continuous or With sleeve.	Double- click
Order	This column indicates the position of this node among the series of nodes in continuous reinforcement.	Single click

See also

The Main Reinforcement spreadsheet
A Beam/Column type of Continuous System with Variable Geometry
A Beam-slab of Continuous System with Variable Geometry
Culvert Design
Abutments, Piers, and Retaining Walls

Using the Main Reinforcement spreadsheet

You can access this spreadsheet in many ways:

- Activate the "Rebar Placement" mode on Activation toolbar.

- Double-click on any continuous system to open the *Rebar Placement* window. An elevation view of continuous system will be displayed on the screen.

- Select the **Main reinforcement** spreadsheet in the **Rebar Placement** menu. The spreadsheet will include all longitudinal rebars included in this continuous system.

OR

- Double-click on a longitudinal rebar on the elevation view or within a cross-section. The **Main reinforcement** spreadsheet will include information on this rebar only.

Transverse Reinforcement Spreadsheet

The **Transverse Reinforcement** spreadsheet is accessible from the **Rebar Placement** menu of *Rebar Placement* window. It is composed of two spreadsheets: **Transverse Reinforcement** and **Pattern**.

In the **Transverse Reinforcement** spreadsheet, you will find details such as the number of stirrups included in each sequence and the position of this sequence relative to the continuous system origin.

The **Patterns** spreadsheet includes detail about the bending shape(s) that composes a particular pattern of stirrup and its position within continuous system.

Section 2-2
6-30M
2-30M

Section 3-3
2-30M
2-30M

Section 4-4
2-10M
4-30M
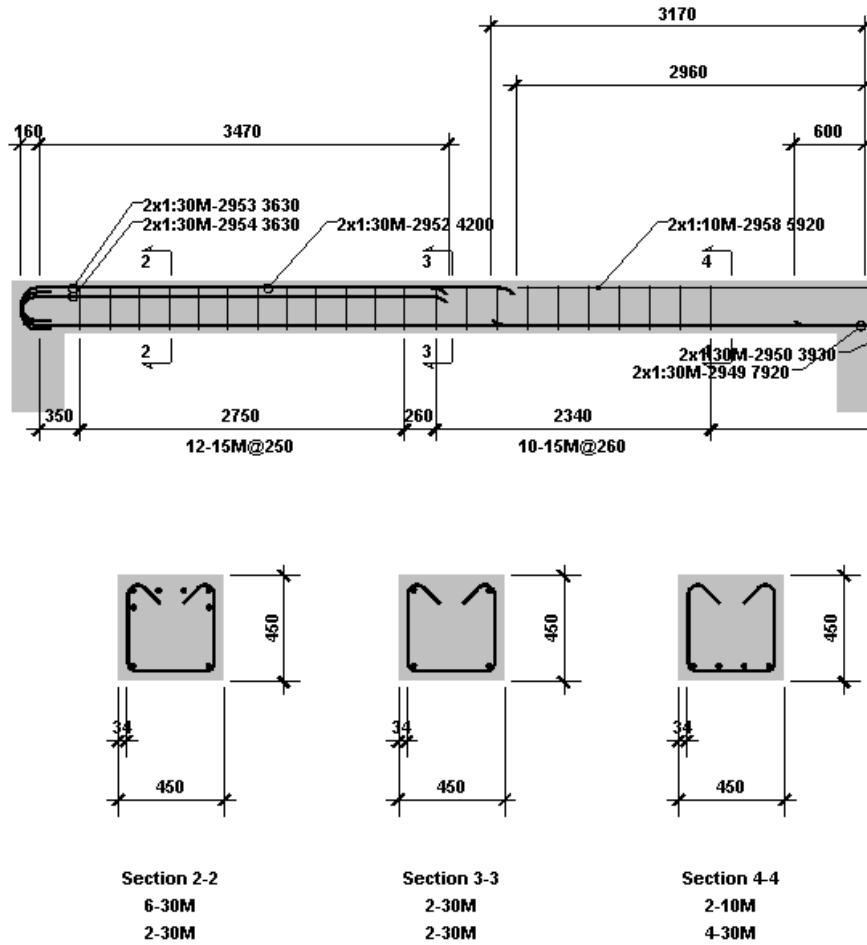
The first sequence of stirrups begins at 350mm from the continuous system origin and ends at 3100mm (350 + 2750). The number of patterns represents the number of stirrups in this sequence.

Warning! If you are creating stirrup sequences yourself, take care not to overlap them. Stirrup sequences are inclusive.

Group: Continuous System (title)

Column	Description	Editing
Rebar ID	Automatically calculated	No
Number	16 alphanumerical characters. Enter a number representing this sequence.	Single click
Reinforcement	Choose the size of rebar.	Double-click
Number of patterns	Enter the number of stirrups that composed the sequence.	Single click

Column	Description	Editing
Spacing s	Stirrup spacing in this sequence.	Single click
zi	Beginning of the sequence relative to continuous system origin.	Single click
zj	End of the sequence relative to continuous system origin.	Single click

Bridge Evaluation Module

Deterioration (strong axis)	Enter the percentage of deterioration according to strong axis for this rebar. (0% means no deterioration)	Single click
Deterioration (weak axis)	Enter the percentage of deterioration according to weak axis for this rebar. (0% means no deterioration)	Single click

See also
Rebar Placement Window
Cross-Sections
Patterns of Transverse Rebars Spreadsheet
Deterioration of Rebars and Cables
Copying a Stirrup Sequence along with Patterns

Using the Transverse Reinforcement spreadsheet

You can access this spreadsheet in many ways:

- Activate the "Rebar Placement" mode on Activation toolbar.

- Double-click on any continuous system to open the *Rebar Placement* window. An elevation view of continuous system will be displayed on the screen.

- Select the **Transverse Reinforcement** spreadsheet in the **Rebar Placement** menu. The spreadsheet will include all designed transverse rebars in this continuous system. Select the **Pattern** spreadsheet and look at the different patterns created by VisualDesign™.

OR

- Double-click on a stirrup sequence on the elevation view or within a cross-section. The **Transverse Reinforcement** spreadsheet will include information on this bar only.

See also
Transverse Rebars Spreadsheet
Stirrup and Tie Patterns

Patterns of Transverse Reinforcement

The **Pattern** tab is part of the **Transverse Reinforcement** spreadsheet, which is accessible through the **Rebar Placement** menu. The **Patterns** spreadsheet lists the bending shapes and ties that are composing each stirrup sequence.

You can also create them yourself by combining rebar-bending shapes (hoop, double closed ties, cross ties, etc.) to define your own stirrup patterns that you will be using for verification of reinforced concrete members.

If you are creating patterns yourself, you must define the position of each bending shape that will be part of a particular pattern relative to the continuous system local axis passing through the section.

Look at the cross-section below. Two bending shapes (identical) are used for this particular pattern.
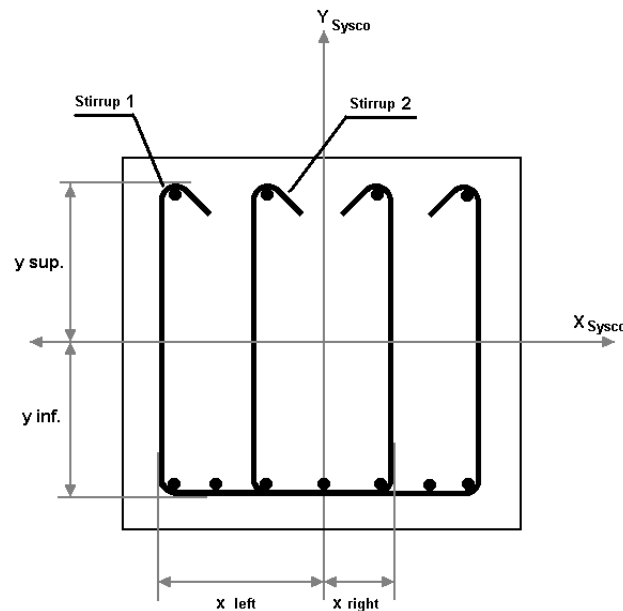
To define the pattern, the user must enter the following distances for each stirrup:

x left: Distance measured from the outside diameter of the stirrup leg that is located left of the continuous system Y-axis passing through the section. A negative value can be entered according to the figure below.

x right: Distance measured from the outside diameter of the stirrup leg located right of the continuous system Y-axis passing through the section.

y inferior: Distance measured from the outside diameter of the bar located below the continuous system X-axis passing through the section. A negative value can be entered according to the figure below.

y superior: Distance measured from the outside diameter of the bar located above the continuous system X-axis.

The point of origin of each stirrup is always located in the bottom left corner. The distances are measured from the outside diameter of the bar.

Group: Continuous System: (title), Transverse Reinforcement: (title)

Column	Description	Editing
ID	Automatically calculated	No
Number	Number of the bar pattern.	Single click
Flip horizontal	Choose [x] to obtain a horizontal mirror view of the bar.	Double-click or Space bar
Flip vertical	Choose [x] to obtain a vertical mirror view of the bar.	Double-click or Space bar
X left	Looking at the cross-section: Distance measured from the outside diameter of the stirrup leg that is located left of the continuous system Y-axis passing through the section. Negative or positive value.	Single click
Y inferior	Looking at the cross-section: Distance measured from the outside diameter of the bottom bar located below the continuous system X-axis passing through the section.	Single click
X right	Looking at the cross-section, distance measured from the outside diameter of the stirrup leg that is located right of the continuous system Y-axis passing through the section.	Single click
Y superior	Looking at the cross-section, distance measured from the outside diameter of the top bar located above the continuous system X-axis passing through the section.	Single click
Bending Shape	Bending shape used in this stirrup pattern.	Double-click
Angle	Shear wall: Position of stirrups in this continuous system. In shear walls, the layout of stirrups is horizontal at 180 deg. (In beams, stirrups are vertical at 0 deg.)	Single click

See also

Transverse Rebars Spreadsheet
Bending Shapes
Cross-Sections

Copying a Stirrup Sequence along with Patterns

- Open the **Transverse Reinforcement** spreadsheet in **Rebar Placement** menu.

- Select the line that corresponds to the stirrup you want to copy.

- Right click and choose function **Duplicate** in the spreadsheet's contextual menu.

Or

- Open the **Transverse Reinforcement** spreadsheet by double-clicking on a stirrup sequence on elevation view or on a stirrup in an appropriate cross-section.

- The spreadsheet includes data about the selected stirrup sequence.

- Select the lines and right click. Choose function **Duplicate** in the spreadsheet's contextual menu.

See also
Spreadsheets' Contextual Menu
Duplicate function

Cross-sections

This spreadsheet is useful to create and define cross-section locations along a continuous system. The position of the cross-section line of cut is determined from origin of continuous system. Cross-sections are helpful to visualize rebars and to edit them through cross-sections. In fact, you are allowed to modify the location and dimension of main rebars and stirrups.

The spreadsheet includes cross-sections' coordinates relative to the origin (z_i) of continuous system. Select the **Cross-sections** spreadsheet through the *Rebar Placement* window, under **Rebar Placement** menu. Look at the procedure below for the creation of a cross-section.

Section 2-2
6-30M
2-30M

Section 3-3
2-30M
2-30M

Section 4-4
2-10M
4-30M
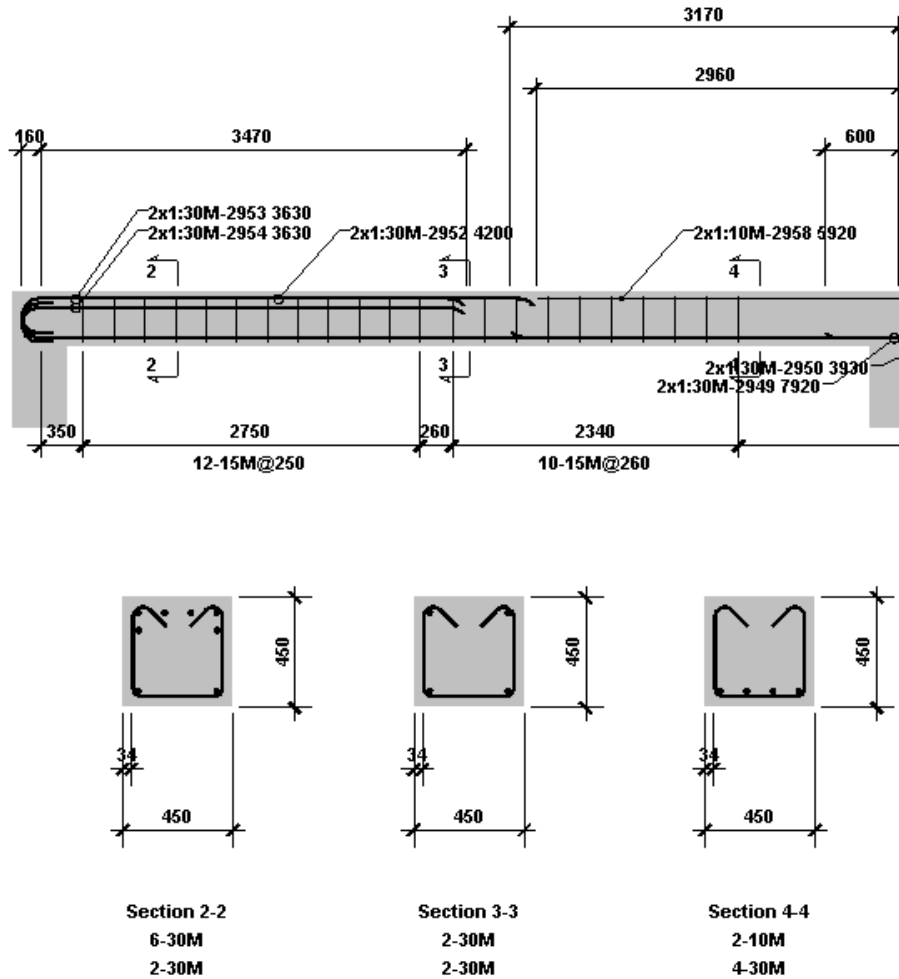
To create the cross-sections shown above, you used the function **Automatic Generation of Cross-Sections** (**Rebar Placement** menu). The generated cross-sections are always located on each side of supports and at mid-spans. To consult data, go to **Rebar Placement** menu and select **Cross-sections**.

6	Number	X Screen mm	Y Screen mm	Z ContSyst mm	Scale	Mask Interaction
1	2	1125	-2725	1125	2.00	[ ]
2	3	3500	-2725	3500	2.00	[ ]
3	4	5875	-2725	5875	2.00	[ ]

X Screen = Position of the displayed cross-section. We can see that the cross-section will be displayed at the same location as the line of cut on continuous system Z contsyst).

Y Screen = Vertical position of cross-section relatively to continuous system local y-axis. A negative value indicates that the cross-section will be displayed below this axis.

Z ContSyst = Position of the line of cut along continuous system, relatively to continuous system origin Zo.

Scale: Scale for cross-section, with respect to continuous system drawing.

Group: Continuous System: (titre)

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters describing the cross-section.	Single click
X screen	X-coordinate of displayed cross-section. Generally, for horizontal continuous system, this distance should be the same as Z ContSyst, which is the position of line of cut.	Single click
Y screen	Y-coordinate of displayed cross-section. (Generally, the cross-section is located below the elevation view.) If the continuous system is vertical (columns), this coordinate should be equal to Z ContSyst.	Single click
Z ContSyst	Position of the cross-section line of cut along continuous system, from the origin (zi) of continuous system.	Single click
Scale	Scaling of displayed cross-section relatively to continuous system scaling (1.0).	Single click
Mask Interaction	The 3D interaction curve is masked, by default. Double-click to unmask it.	Double-click or Space bar

See also
Rebar Placement window
Transverse reinforcement Spreadsheet
Main reinforcement Spreadsheet
Design of a continuous system
Rebars Bending Shapes
Editing Cross-sections
Automatic Generation of Cross-sections

Automatic Generation of Cross-sections

Use this function (**Rebar Placement** menu of *Rebar Placement* window) to automatically generate cross-sections.

For continuous beams, cross-sections are always generated at mid-span. If supports are present (or columns), cross-sections will also be drawn at the left and right of supports.

The Cross-sections spreadsheet will be posted on your screen. Cross-sections to be generated will be listed in this spreadsheet. Elevations and scales can be modified, and the 3D interaction curves can be masked or displayed through the "Mask interaction" column.

To create cross-sections yourself, select **Cross-sections** in **Rebar Placement** menu (*Rebar Placement* window).

See also
Cross-sections
Editing Cross-sections

Rebar Placement Window (General)

"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

Rebar Placement Window

This window includes all functions useful to check, modify and design the required reinforcement for concrete members, including prestressed concrete, that are part of any continuous systems. The name of the project file is written in the upper part of the screen.

Activate the *Rebar Placement* mode and double-click on any continuous system to open this window. Then, you will be allowed to look at the design or to create your own design by adding and placing main reinforcement and stirrups.

When the Rebar Placement mode is activated, access the *Rebar Placement* window by doing one of the following procedure:

- Click on any continuous system and select the **Properties** function;

- Double-click on any continuous system.

The *Rebar Placement* window is composed of the following menus:

FILE

Close
Save Rebar Placement
Save Rebar Placement As
DXF Out
Project Configuration
Print
Print Preview
Printer Configuration

EDIT

Undo
Redo
Properties
Add a Longitudinal Rebar
Translation
Delete
Activate Cursor Mode
Move
Stretch

VIEW

View Options
Zoom Window
Global Zoom
Zoom +
Zoom -
Dynamic Pan
Toolbars
Status Bar

COMMON

Concrete Materials
Rebar Steel Grades
Standard Reinforcing Bars
FRP Reinforcing Bars
Meshes
Bending Shapes
Cable Steel Grades
Strands
Post-tensioning Mechanisms

REBAR PLACEMENT

Continuous System
Concrete Specifications
Main Reinforcement spreadsheet
Transverse Reinforcement spreadsheet
Cross sections
Cable Groups and Layout
Bar List
Automatic Generation of Cross-sections

RESULTS

Loss of Prestress in Cables
Stresses at each construction stage
Graphs
Intermediate Results
General Results
3D Interaction Curves

HELP

Using Help
Reminder - Editing Keys
About VisualDesign™

See also
Rebar Placement window Legend

List of Toolbars

Toolbars	Icons

Standard

Exit – Save Rebar Placement – Copy – Paste-
Undo – Redo – Print – Print Preview–
Contextual Help

Cursor

Activate Cursor Mode– Move– Stretch

Toolbars	Icons
Edit	Properties – Delete – Add a longitudinal rebar
View	Zoom Window– Zoom Global– Zoom + – Zoom - – Dynamic Pan– View Options – Increase Font Size – Reduce Font Size – Graphs

See also
Rebar Placement Window

Legend

A legend is posted at the right of the continuous system elevation view. It includes the project title and number and the continuous system number. However, this information must be indicated in the *General* tab of **Project Configuration**. Otherwise, the legend will contain the continuous system number only.

The font that is used in this legend is the one used for rebar placement dimensions (see the *Dimensions* tab of **View Options**) and the colour can be modified in the *Colours* tab, at root *General / Titles*.

Contextual Menu

While in the *Rebar Placement* window, right click to access a contextual menu. To edit elements in a quicker way, choose a function among the menu: **Properties**, **Delete**, **Print** and **Print Preview**.

An element must be selected (longitudinal or transverse rebar, or a cable) before using the **Properties** or **Delete** functions. The **Properties** function will call up the spreadsheet for the selected element.

The **Print** and **Print Preview** functions only applied to the drawing, not to the spreadsheets that can be called up using the **Properties** function.

See also
Properties
Delete
Print
Print Preview

Mouse wheel

Use the mouse wheel to zoom in and zoom out by scrolling the wheel up and down in the Rebar Placement window.

Editing - *Rebar Placement* Window

Properties Function

The Properties icon of Edit toolbar (Rebar Placement Window)

When the "Rebar Placement" activation mode in on, double-click on a continuous system and the **Properties** function will open the *Rebar Placement* window and will display an elevation view of the selected continuous system.

Once in this window, select an element in a cross-section or in the elevation view and use the **Properties** function to call up an element spreadsheet. To do so:

- Click on a longitudinal rebar and press the **Properties** icon to access the **Main reinforcement** spreadsheet.

- Click on a stirrup (all the sequence will be highlighted) and press the **Properties** icon to access the **Transverse reinforcement** spreadsheet.

- Click on a cable and press the **Properties** icon to access the **Cable Groups** spreadsheet.

- Click on a cross-section and press the **Properties** icon to access the **Cross-sections** spreadsheet.

Use the short-cut key [**Ctrl**]+**T**.

See also
Rebar Placement window
Main Reinforcement spreadsheet
Transverse Reinforcement spreadsheet
Cross-sections

Selecting Several Elements of the Same Type

Multiple selections in the Rebar Placement window

You are allowed to select several elements of same type (longitudinal rebars, transverse rebars, cables or cross-sections) when you are working in the *Rebar Placement* window by pressing down the **Ctrl** key before clicking on elements that you want to select. All elements can be selected this way except for stirrups (transverse rebars) that are drawn in cross-sections. Main reinforcement, cross-sections and cables can be select either in cross-sections or in the continuous system.

When elements are selected, call up the spreadsheet by selecting the **Properties** function.

Selection of patterns of transverse rebars: select patterns in cross-sections and press the **Properties** function. The **Transverse Reinforcement** spreadsheet will include rebars that are part of the patterns. If you select two patterns (or more) which include the same transverse reinforcement, they will be listed only once in the spreadsheet.

If an object is already selected in the continuous system, it is not allowed to select it once again in a cross-section and vice versa.

Multiple selections in a spreadsheet

You can also select several elements through a spreadsheet by selecting contiguous lines and then by exiting the spreadsheet (by pressing OK or CANCEL). The following spreadsheets accept multiple selections:

- Longitudinal rebars;

- Transverse rebars;

- Cross-sections;

- Cables;

- Loss of prestress in cables.

When exiting a spreadsheet after having selected rebars or cables, only the rebars and cables that are located in the continuous system will be highlighted. They will not be selected in cross-sections. If you selected cross-sections in the cross-sections spreadsheet, they will be highlighted on your screen but their line of cut (on the continuous system) will not be selected.

If you modified data in a spreadsheet, the multiple selections will not be transposed on the screen. You must exit the modified spreadsheet and call it up again.

Spreadsheets allow multiple selections of elements in the *Rebar Placement* window if this selection is homogeneous, meaning that elements must be of the same type. If this is the case, the **Properties** and **Delete** functions will be activated.

Translation Function

You are allowed to translate elements in the x, y and/or z direction with the function **Translation**, available in the **Edit** menu of *Rebar Placement Window*. The translation can be apply to some elements only or to all elements include in a continuous system such as longitudinal rebars, stirrups or prestressing cables.

Delta of translation is always given according to the continuous system local axes of a beam or column.

The **Translation** function calls up this dialog box:

Options	Definition
Objects to be translated	The *Stirrups*, *Longitudinal rebars* and/or *Cables* boxes will be shaded and checked if you selected elements on your screen before selecting the Translation function. Checked boxes inform you on elements that will be translated.
	If you did not select any elements and you call up the Translation dialog box, check the box corresponding to elements that you want to translate.
Translation (delta)	Enter deltas of translation according to the continuous system local axes.

The **Translation** function can be used in two different ways:

• If no element is selected on the screen, delta of translation will be applied to all elements (longitudinal rebars, stirrups and cables). All boxes will be checked and shaded in the **Translation** dialog box. This function is useful if you modified node zi or zj coordinates or if you modified the locations of rigid extensions because the placement of rebars and cables is always relevant to the axis that goes through rigid extensions. So, use this function to replace the position of all cables and reinforcement.

CivilDesign inc. Engineering Software

- Double-click on any element that you wish to translate and select the **Translation** function in **Edit** menu. The delta of translation will be applied to this element only. One box will be checked and shaded in the **Translation** dialog box.

Delete

The Delete icon of Edit toolbar (Rebar Placement window)

The **Delete** function deletes the selected elements on your screen: reinforcing bars and cables.

Short-cut key: **Del**.

Activate Cursor Mode

The Cursor mode icon of Cursor toolbar (Rebar Placement window)

Use this function to activate the editing of reinforcing bars or cables with functions **Stretch** or **Move**.

To deactivate the editing mode, press the icon again.

See also
Stretch
Move

Move Function

The Move icon of Cursor toolbar (Rebar Placement window)

Activate the **Cursor** mode and select the **Move** function to move a rebar or a cable. You are allowed to move a rebar (longitudinal and transverse) on the elevation view or in a cross-section.

Press the **Cursor** mode again to deactivate the editing mode.

Important. Use the red target that appears on your screen to accurately position the rebar or cable. The first circle corresponds to a 30 mm concrete cover and the second one, to 40 mm concrete cover. The distance is always measured from the rebar or cable outside diameter that you are currently moving.

Move Transverse Reinforcing Bars

IN A CROSS-SECTION

Press the **Move** icon and click on a stirrup. Keep the right mouse button down while you move the rebar either in the x or y direction. Release the mouse button to fix the position.

IN THE ELEVATION VIEW

Press the **Move** icon and click on a stirrup sequence. Keep the right mouse button down while you move the stirrup sequence in the z direction. Release the mouse button to fix the position.

Move Main Reinforcing Bars

IN A CROSS-SECTION

Press the **Move** icon and click on a rebar layer. Keep the right mouse button down while you move the rebar layer either in the x or y direction. Release the mouse button to fix the position.

N.B. Press down the [Shift] key before clicking on a rebar layer if you want to fix the x coordinate while you move the layer in the y direction.

IN THE ELEVATION VIEW

Press the **Move** icon and click on a longitudinal rebar. You will select the layer. Keep the right mouse button down while you move the rebar layer in the z direction. Release the mouse button to fix the position.

See also
Cross-sections
Rebar Placement window
Activate Cursor mode

Stretch Function

The Stretch icon of Cursor toolbar (Rebar Placement window)

Use this function to stretch a rebar on the elevation view or in cross-section. It can also modify the spacing in a rebar layer or stirrup sequence. Activate the Cursor mode before selecting the **Stretch** function and then, select a rebar or stirrup sequence.

Disable the Cursor mode to exit the editing.

Important. Use the red target that appears on your screen to accurately position the rebar or cable. The first circle corresponds to a 30 mm concrete cover and the second one, to 40 mm concrete cover. The distance is always measured from the rebar or cable outside diameter that you are currently moving.

Stretch a Transverse Reinforcing Bar (Stirrup)

IN A CROSS-SECTION

Press the **Stretch** icon and click on a stirrup leg. Keep the right mouse button down while you stretch the stirrup leg to another position. Release the mouse button to fix its position.

IN THE ELEVATION VIEW

Press the **Stretch** icon and click on a stirrup sequence to modify spacing between stirrups. Keep the right mouse button down while you stretch or shorten the stirrup sequence. Release the mouse button to fix its position.

Stretch a Main Reinforcing Bar

IN A CROSS-SECTION

Press the **Stretch** icon and click on a rebar layer to modify spacing between longitudinal rebars. Keep the right mouse button down while you stretch or shorten the spacing between rebars by moving your cursor. Release the mouse button to fix its position.

IN THE ELEVATION VIEW

Press the **Stretch** icon and click on a longitudinal rebar that you want to stretch or shorten. Keep the right mouse button down while you stretch or shorten the rebar. Release the mouse button to fix its position.

See also
Cross-sections
Move
Activate Cursor mode

Editing Cross-sections

You must activate the Cursor mode before editing a cross-section.

Editing the Cross-Section Line of Cut

To move the cross-section line of cut, press the **Move** icon, select the line of cut and, while pressing down the left mouse button, move your cursor. Release the mouse button. The cross-section AND the line of cut will be moved.

Editing the cross-section

You can delete, edit and move a cross-section. To move the cross-section itself, activate the Move icon, select the cross-section by clicking on its contour and move it with your cursor, in the y and z direction. Only the drawing will be moved, not the line of cut.

Double-click on a cross-section to call up the Cross-sections spreadsheet. Modify parameters.

N.B. The Undo and Redo functions can be used while editing a cross-section.

See also
Stretch
Move

Add a Main Rebar

The "Add a Main Rebar" icon of Edit toolbar - Rebar Placement window

Use this function, available in **Edit** toolbar, to graphically add a longitudinal rebar on the elevation view of any continuous system of the *Slab* type, with variable geometry.

Procedure:

- Activate the icon on Edit toolbar.

- Use your cursor and click on a first point (beginning of the rebar) and then on a second point (end of rebar) representing the length of the rebar that you want to add. Then, the following dialog box will appear on your screen:

Add Longitudinal Rebar

- From outer side
- From inner side

Longitudinal Rebars: 15M

Concrete Cover: 50 mm

Spacing: 200 mm

OK   Cancel

CivilDesign inc. Engineering Software

- Choose the location of the bar relatively to continuous system (outer or inner side). Select a type of rebar, concrete cover and transverse spacing.

- Press OK.

- Check forces and resistances diagrams displayed on your screen.

See also
Rebar Placement window
The Slab Type of Continuous System with Variable Geometry

Save Rebar Placement As

Use this function available in the **File** menu (*Rebar Placement* window) to copy the reinforcing details of current continuous system to other continuous systems.

The following dialog box will appear on your screen:
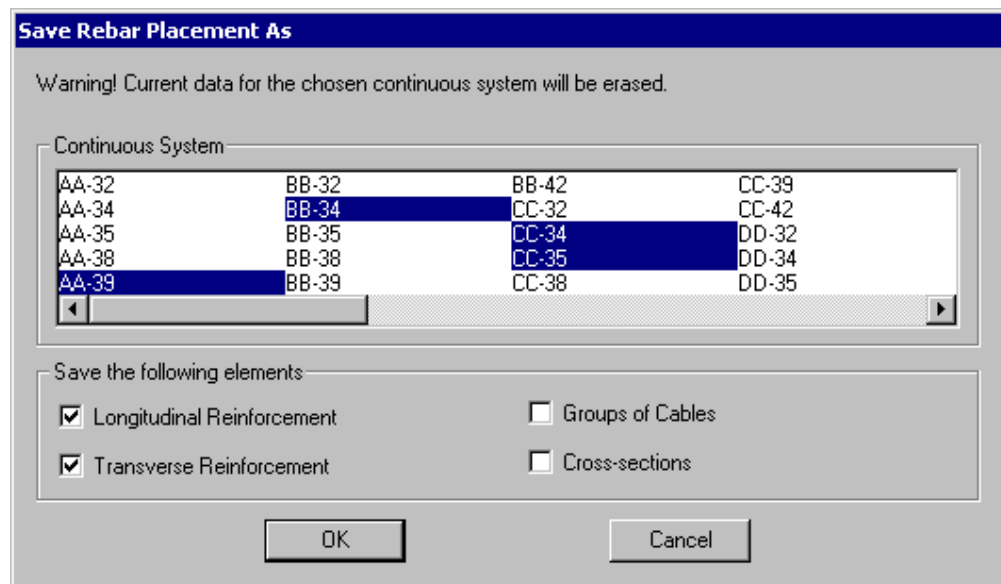
- Select the continuous system numbers that will be modified. Select the elements that will be copied (longitudinal rebars, transverse rebars, groups of cables and/or cross-sections) by activating appropriate check boxes.

- Click OK.

Select Horizontal Continuous Systems

To select all the horizontal continuous systems of your structure, choose the **Select Elements/Horizontal Continuous Systems** under **Edit** menu. This function is useful to get the Bar list or concrete quantities for beams only.

Select Vertical Continuous Systems

To select all the vertical continuous systems of your structure, choose the **Select Elements/Vertical Continuous Systems** under **Edit** menu. This function is useful to get the Bar list or concrete quantities for columns only.

Reminder - Editing Keys

Look at tables below to learn how to edit elements in the *Rebar Placement* window. Some editing is done with function **Stretch** or **Move** but some others must be done by pressing down a control key such as [Ctrl] or [Shift], and by clicking on either **Stretch** or **Move** function.

Editing Longitudinal Reinforcing Bar

Location	Action	Control Keys	Select Function
Cross-section	Move the selected bar layer towards direction x	N/a	Move
Cross-section	Modify the spacing between selected bars towards x-axis	N/a	Stretch
Cross-section	Move selected bars towards direction y	[Shift] (Optional) to move towards y without moving the x coordinate.	Move
Elevation view	Stretch selected bars towards direction z	N/a	Stretch
Elevation view	Move selected bars towards direction z	N/a	Move

Editing Stirrups

Location	Action	Control Keys	Select Function
Cross-section	Move selected stirrup towards direction x or y.	N/a	Move
Cross-section	Stretch selected stirrup leg towards direction x or y.	N/a	Stretch
Elevation view	Move the whole selected stirrup sequence along continuous system (z-direction)	N/a	Move
Elevation view	Modify the spacing of stirrup in the selected sequence (z-direction)	N/a	Stretch

Editing Prestressing Cables

Location	Action	Control Keys	Select Function
Cross-section	Move selected cable towards direction x	N/a	Move
Cross-section	Modify spacing between selected cables (x-direction)	N/a	Stretch
Elevation view	Move selected cable towards direction y	[Ctrl]+[Shift]	Move
Elevation view	Move selected cable towards direction z (along continuous system length)	[Ctrl]	Move
Elevation view	Create a cable deformation towards y-direction	[Shift]	Move
Elevation view	Create a cable deformation towards z-direction.	N/a	Move

Editing Cross-sections

Location	Action	Control Keys	Select Function
Cross-section	Selected the cross-section outline and move it towards y or z-direction. The line of cut on elevation view will not move.	N/a	Move
Line of cut on Elevation view	Move the line of cut along continuous system (z-direction). The cross-section will move along the line of cut.	N/a	Move

See also

Activate Cursor mode
Move
Stretch
Editing Cross-sections
Save Rebar Placement As (Copy Identical Continuous system)
Moving a Whole Cable Layout
Deforming a Cable Segment
Modifying Spacing between Cables

View Options
Rebar Placement Window

View Options Dialog Box

The "View options" icon of Rebar Placement window

While designing continuous system reinforcement in the *Rebar Placement* window, press this icon to call up a **View Options** dialog box that will help you visualizing elements. The **View Options** dialog box that will appear is composed of three tabs: **Rebar Placement**, **Dimensions** and **Colours**.

Use these tabs to select what you want to see on your screen: elements, numbers, display of beams diagrams (moments, shears, deflections, etc.), display 3D interaction curves for columns and also reinforcing bars detailing for your concrete elements. Modify colours through the **Colours** tab of the same dialog box.

Rebar Placement tab

Select this tab and select what you want to see on screen. The tab is composed of a selection tree that has four roots: *General*, *Beams Diagram*, *Columns Diagrams* and *Dimensions*. Expand the roots and check the items that correspond to your choice of display. Modify the diagrams colours through the **Colours** tab.

General

The *General* root includes the following items:

- Project Number;

- Project Name;

- Continuous system number;

- Continuous system local axes.

Beam Diagrams

NOTES:

Vfy max without stirrups: This diagram represents the shear resistance of the concrete section without any stirrups (according to the selected construction code). For example, this value is equal to 0.5*Vc in the A23.3-94 Code.

The Variation of stresses in rebars will be displayed if at least two "Service" load combinations are defined and calculated.

CivilDesign inc. Engineering Software

View Options

Rebar Placement | Dimensions | Colours

```
⊞···☐ General
⊟···☑ Beam Diagrams
        ☑ Mfx, Mrx vs z
        ☑ Vfy, Vry vs z
        ☐ VfyMax without stirrups vs Vfy
        ☐ vfy, vfy max. vs z
        ☐ Mfy, Mry vs z
        ☐ Vfx, Vrx vs z
        ☐ Nfz, Nrz vs z
        ☐ Tfx, Trz vs z
        ☑ Cracking
        ☑ 1.20 * Mcr, Mrx vs z
        ☑ 1.33 * Mfx, Mrx vs z
        ☑ Proportioning of Longitudinal Reinforcement
        ☐ Stresses Variation in Rebars
        ☐ Design Load vs z
        ☐ Icr, Ie vs z
⊞···☑ Column Diagrams
⊞···☑ Dimensions
```

Font...

OK   Cancel   Apply   Help

Column Diagrams

The *Columns Diagrams* root includes the following diagrams:

- Mfx, Mrx;

- Vfy, Vry;

- Mfy, Mry;

- Vfx, Vrx;

- Nfz, Nrz

- Tfx, Trz;

- Mrx vs. Nz;

- Mry vs. Nz

- Mrx, Mry vs. Nrz (3D Interaction Curves)

- Mnx vs. Nnz;

- Mny vs. Nnz

- Mnx, Mny vs. Nnz (3D Interaction Curves)

CivilDesign inc. Engineering Software

- Mpx vs. Npz;

- Mpy vs. Npz

- Mpx, Mpy vs. Npz (3D Interaction Curves)

Dimensions

The *Dimensions* root includes the following items for the displayed drawing:

Main Reinforcement:

- Dimensions;

- Rebar numbers and lengths;

Transverse Reinforcement:

- Dimensions;

- Rebar numbers and stirrup spacing;

Group of cables:

- Dimensions;

- Cable numbers and spacing;

Cross-sections dimensions.

The Dimensions tab

Select this tab and modify the pen weight and distances (extensions) for dimensioning.

Press the "Font" button and choose the font style that will be used for dimensions. Specify units, minimum spacing between drawing and edge, drawing scale and text height.
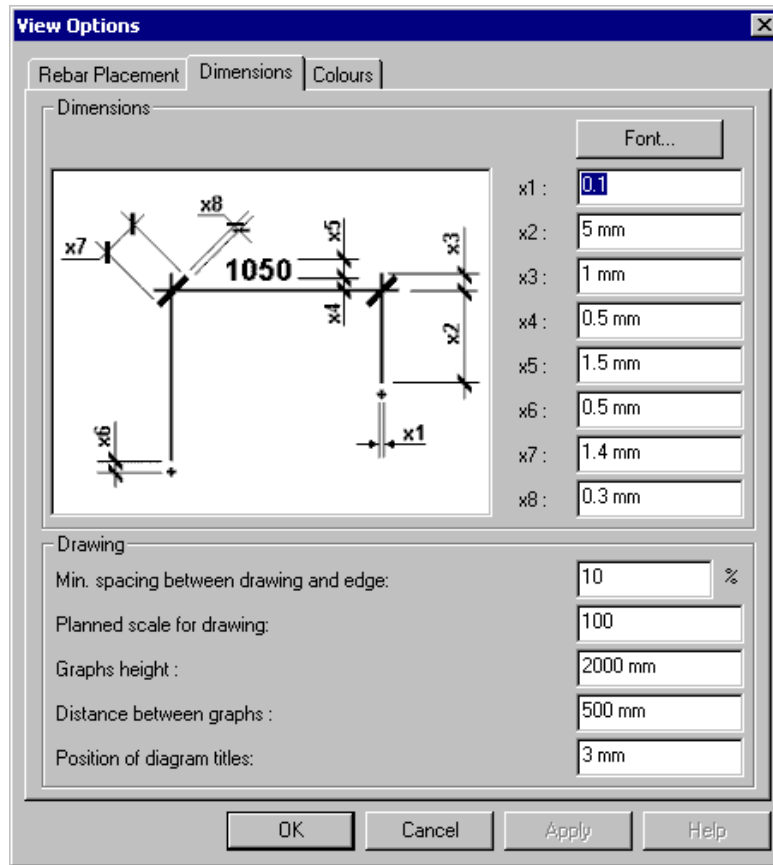
See also
Rebar Placement tab
Colors tab

The Colours tab

Select this tab (**View Options** of *Rebar Placement* window), to modify the colours of displayed elements. You are allowed to modify element colour, line style and symbol. To print in black and white, check the "Black and White" option in the dialog box.

The tab is a selection tree that is composed of the following roots: *General*, *Diagram*, *Dimension* and *Prestressing*. Expand those roots and highlight an item. Click on a coloured square to modify the colour of this item.

CivilDesign inc. Engineering Software
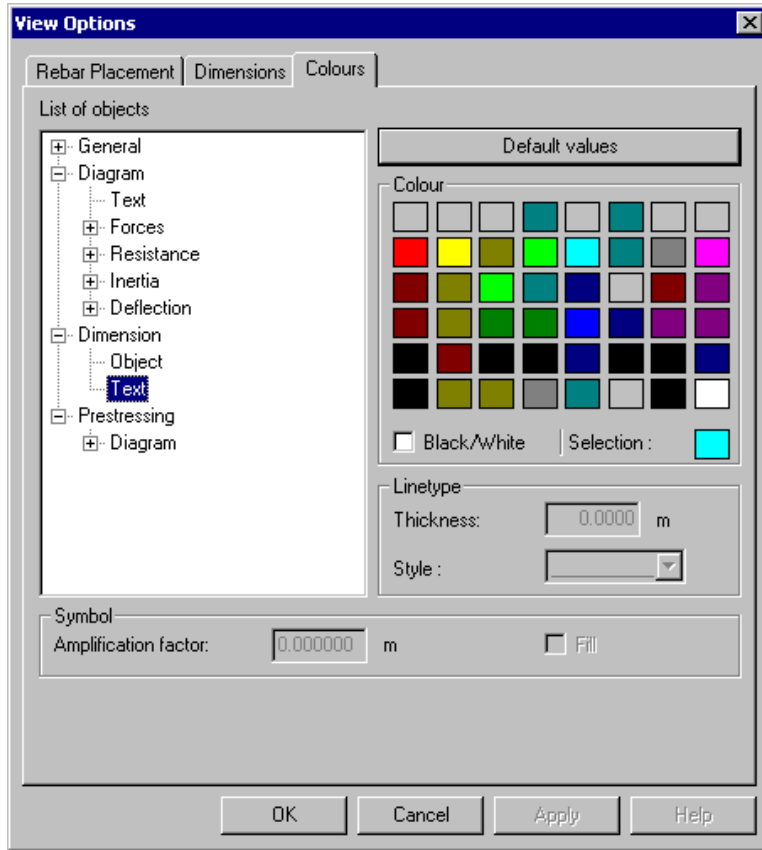
Graphical Results

Drawing - Concrete Design

When activating the *Rebar Placement* window, display the details dimensions through the **Rebar Placement** tab of **View Options**. Description of longitudinal rebars and prestressed cables is as follows:

Description of longitudinal rebars :
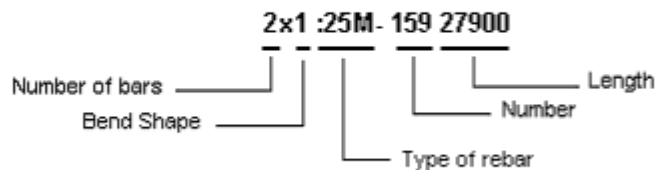
Stirrups:

15 – 10M@ 300: 15 bars no. 10 (Metric), 300 mm centre to centre.

Export drawings in DXF (AutoCAD compatible):

You are now allowed to export rebar placement drawing of concrete elements.

See also
View Options
Rebar Placement Window
Bend Shapes
Transverse Rebars Patterns
Cross-sections
Export in DXF format

Displaying 3D interaction curves (columns)

In the *Rebar Placement* window, display 3D interaction curves to verify the design of concrete columns and shear walls. Follow the procedure below:

- Before displaying interaction curves, you must first create cross-sections. To quickly generate cross-sections, select **Automatic Generation of Cross-sections** in the **Rebar Placement** menu. To define cross-sections "by-hand", use the **Cross-sections** spreadsheet and enter the required information to position the cross-sections.

- To unmask the curve, double-click in the "Mask interaction" column of **Cross-sections** spreadsheet.

- Open the **View Options** and select 3D interaction curves in the *Rebar Placement* tab. Interaction curves will be displayed next to cross-sections.

To magnify the curve on the screen, use functions **Zoom window** and **Zoom +**.

To see all points within the interaction curve, click once on the curve and rotate it with the keyboard arrows. Check the design. If the capacity exceeds 100%, points will be located outside the 3D curve. The column or shear wall design load is written below the cross-section.

The design load is also supplied at the location of the cross-section. Blue crosses inside the curve represent forces acting at this cross-section.

- Go to **Results** menu and select **Interaction curves**. The appearing dialog box allows you to look at Mr-Pr (resistance), Mn-Pn (nominal resistance) and/or Mp-Pp (probable resistance) interaction curves according to a plane passing through the curves and for a selected cross-section. To select a plane, enter an angle in appropriate field and look at the displayed curves (Resistances P and M). Use the *Graph* toolbar that is supplied with this dialog box, to print or edit the graph attributes. Refer to the Graph toolbar.

See also
Rebar Placement tab
Colours tab
General Results - Columns

Mr & Mp in Interaction Curves

For Ductile Columns and Shear Walls:
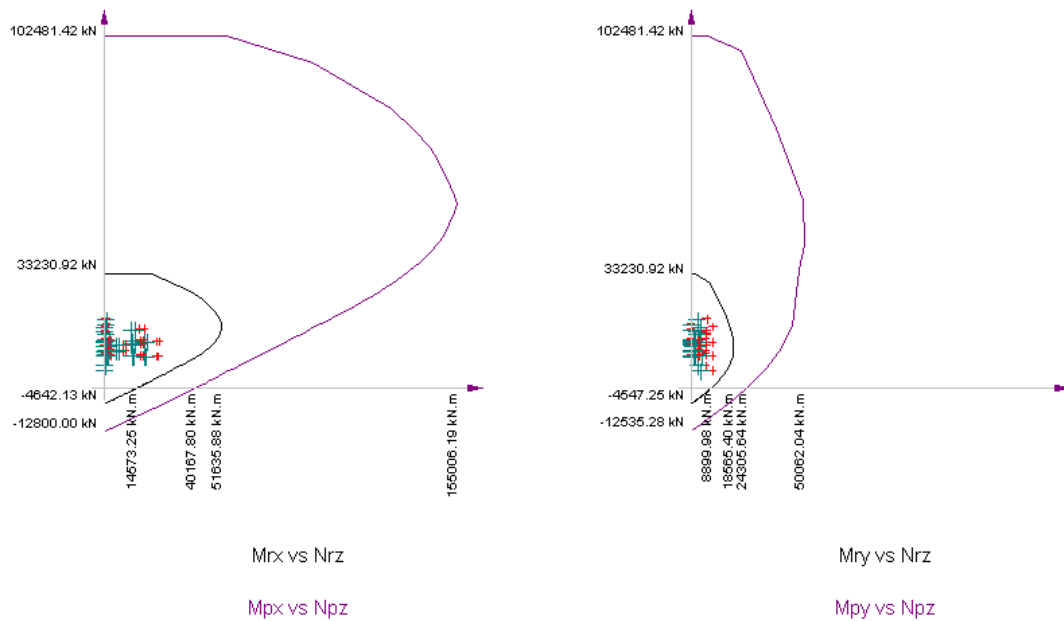
In the images shown above, we can see that the Mr curve is very small to the Mp curve. It can be explain by the following: To obtain probable forces, the yield strength, fy, of steel must be multiply by a factor ϕ of about 1.25 to1.3. Capacity equations use a factor ϕ of 0.85. Furthermore, when the amount of supplied steel reinforcement is smaller than the required amount, CAN/CSA-A23.3-95 code provision 10.10.5 reduces the axial strength of such elements to limit its creeping.

See also

3D Interaction Curves
Shear Wall Ductility

Numerical Results

Concrete Design Results

When the concrete design is completed, go to the **Results** menu in VisualDesign main window, and select **Structure Design / Concrete** to have a look at the calculated design loads of all continuous systems in your project. The spreadsheet is composed of the following tabs: **Beam/Joist**, **Column**, **Shear wall** and **Beam-slab**.

The Beam / Joist tab

Group: Design Results

Column	Description	Editing
ID	Automatically calculated	No
Number	Section number	No
Design Load Positive Moment	Mf max + / Mr +	No
Design Load Negative Moment	Mf max - / Mr -	No
Design Load Shear	Vf max / Vr	No
Cracking Positive Moment	Ratio between the calculated cracking under positive bending moment and the specified crack control parameter for positive bending.	No
Cracking Negative Moment	Ratio between the calculated cracking under negative bending moment and the specified crack control parameter for negative bending.	No

The Column tab

Group: Design Results

Column	Description	Editing
ID	Automatically calculated	No
Number	Section number	No
Design Load Shear Strong axis	Vfy max / Vry	No
Design Load Shear Weak axis	Vfx max / Vrx	No
Design Load Interaction	Ratio between combined factored compression and bending and resistance to these combined forces.	No
As / Ag max	Maximum ratio of supplied area of steel to the gross area of concrete section.	No

The Shear Wall tab

Group: Design Results

Column	Description	Editing
ID	Automatically calculated	No
Number	Section number	No
Design Load Shear	Ratio between the maximum factored shear force in the shear wall and the shear wall resistance to shear.	No
Design Load Interaction	Ratio between combined factored compression and bending and resistance of the shear wall to these combined forces.	No
As / Ag max	Maximum ratio of supplied area of steel to the gross area of concrete section.	No

---

CivilDesign inc. Engineering Software **13-113**

The Beam-slab tab

Group: Design Results

Column	Description	Editing
ID	Automatically calculated	No
Number	Section number	No
Design Load Positive Moment	Mf max + / Mr +	No
Design Load Negative Moment	Mf max - / Mr -	No
Design Load Shear	Vf max / Vr	No
Cracking Positive Moment	Ratio between the calculated cracking under positive bending moment and the specified crack control parameter for positive bending.	No
Cracking Negative Moment	Ratio between the calculated cracking under negative bending moment and the specified crack control parameter for negative bending.	No
Design Load Mf – Nf +	Ratio between combined factored compression and bending and resistance of the slab to these combined forces.	No
Design Load Mf – Nf -	Ratio between combined factored tension and bending and resistance of the slab to these combined forces.	No

Bar List – Continuous System

The program creates a bar list for a chosen continuous system or for the whole project.

Bar list for a particular continuous system

While you are working in the *Rebar Placement* window, select **Bar List** heading under **Rebar Placement** menu.

Bar list for the whole project

While you are working in the VisualDesign™ main menu, you are allowed to print the bar list for one or several selected continuous systems. Select the heading **Bar List / Continuous System** under **Results** menu. However, if you have not selected any continuous system, the bar list will include all rebars for the whole project.

Group: Continuous System (title)

Column	Description	Editing
ID	Automatically calculated	No
Continuous System	Continuous system number which elements belong.	No
Rebar	Type of rebar: longitudinal, open or closed ties, cross-tie, etc.	No
Quantities	Number of bars of this type in the continuous system.	No
Diameter	Rebar diameter	No
Length	Rebar length	No
Bend shape	Bend shape of this type of rebar.	No
Protection	Epoxy coated or not	No
Material	Rebar steel grade	No
A	Length A	No
B	Length B	No
C	Length C	No
D	Length D	No
E	Length E	No
F	Length F	No
G	Length G	No
H	Length H	No
J	Length J	No
K	Length K	No
O	Length O	No
R	Length R	No
Mass	Mass for this type of rebar.	No
Mass/m of width	Mass per meter of width for a constant spacing of rebar.	No

Column	Description	Editing
Total Mass	Total mass of all rebars	No

General Results spreadsheet

This spreadsheet is available in the *Rebar Placement* window when the design is completed. It includes many results that can be displayed in a graphic form through the **View Options** of *Rebar Placement* window.

Please verify that ratio **c/d** does not exceed **c/d max** (clause10.5.2 of Code A23.3-95), which is given as a result in this spreadsheet.

If you used the ***Bridge Evaluation*** module, you will also find results related to this evaluation.

Please note that some results will be available in the spreadsheet only if they are displayed in the *Rebar Placement* window. This is the case for cracking factors (β2 or z) and fatigue in main rebars (*Variation of stresses in rebars* diagram). Do not forget that, if you want to obtain these results, load combinations having a *Service* or *Fatigue* status must be defined.

Lines that are marked with yellow mean that there are some parameters exceeding the limits permitted per code.

The **General Results** spreadsheet in split into five spreadsheets to make consultation easier: **Positive Bending Moment**, **Negative Bending Moment**, **Shear Force**, **Axial Force** and **Position of Cables**. The latter is for prestressed concrete projects only.

See also
Bridge Evaluation
Load Combination Status
View Options - Rebar Placement window

Positive Bending Moment tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Mfx max	Envelope of maximum bending moment for strong axis.	No
R'	**Factored resistance** of tensioned reinforcement, considering stresses that can be developed, according to clause 11.4.9.1 of Code A23.3:	No
F'	**Forces** in tensioned reinforcement, considering stresses that can be developed, according to equation 11.4.9.1 of Code A23.3:	No
Mrx	Factored moment resistance.	No
Mnx	Nominal moment (Refer to Ch. 21 of A23.3-95 standard)	No
Mpx	Probable moment (Refer to Ch. 21 of A23.3-95 standard)	No
Design load	Bending moment at this location within continuous system.	No

Bridge Evaluation module only

F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to flexural resistance.	No
Mcr	Bending moment when a tensile stress of fcr is induced in the concrete.	No
fcr	Cracking strength of concrete.	No

Column	Description	Editing
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system. N.B. VisualDesign considers all tensioned rebars, even those that have not attained the yield stress fy. (The value of d that appears here does not correspond to the value of d used in the code equations, except at the beginning of continuous system)	No
dv	Distance between tension and compression resultant forces due to bending, at this location within continuous system.	No
bw	Width of concrete section web at this location within continuous system. For prestressed concrete elements: this width is reduced by total sheath diameter if there is no grout or 1/2 sheath diameter if it is grouted.	No
As	Area of reinforcing steel in tension. (VisualDesign considers all rebars in tension, even those that have not reached fy). However, when considering the strain – deformation compatibility in calculations, the useful area of each bar in tension is factored with this ratio: fs/fy.	No
ϱ	Percentage of steel reinforcement at this location within continuous system.	No

For a Prestressed Concrete Project

Column	Description	Editing
ωp	Ratio of reinforcing bars and prestressing reinforcement according to clause 9.6.8.5.1 of Code S6-88. See the note below.	No
Φp*fps*Aps	Factored tensile strength of prestressing cables.	No
εx	Strain in longitudinal reinforcement used for calculating the shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that considers the shear resistance of cracked sections.	No
c/d	c/d ratio of clause 10.1.4 - Code A23.3, relative to the balanced condition *Concrete Deformation and yield strength of steel*. Compare with c/d max below.	No

Column	Description	Editing
c/d max	Maximum c/d ratio.	No

For load combinations with a "Service" status.
$\beta 2$: S6-00, Cl. 8.12.3
z : A23.3-95, Cl. 10.6.1

Column	Description	Editing
$\beta 2$ or z	Function modified by the proportioning and type of longitudinal rebars.	No
$\beta 2$ max or z max	Value limiting the proportion of longitudinal rebars.	No
Fcr	S6-00 only: Factor that controls cracking in buried structures as per clause 7.8.9.1. This column is present if a "Buried" type of structure was select in the concrete specification.	No
Mw	S6-00 only: Moment at a section where a tensile stress of 0.4 fcr is induced in the concrete.	No
Ms	S6-00 only: Flexural moment at a section under consideration at the SLS load.	No
fs	Tensile stress in reinforcing bars, considering stress-strain compatibility.	No
I eff	Effective moment of inertia.	No
I cr	Moment of inertia of the cracked section, transformed to concrete.	No

Negative Bending Moment tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Mfx min	Envelope of minimum bending moment for strong axis.	No
R'	**Factored resistance** of tensioned reinforcement, considering stresses that can be developed, according to clause 11.4.9.1 of Code A23.3:	No

Column	Description	Editing
F'	**Forces** in tensioned reinforcement, considering stresses that can be developed, according to equation 11.4.9.1 of Code A23.3:	No
Mrx	Factored moment resistance.	No
Design load	Bending moment at this location within continuous system.	No

Bridge Evaluation module only

Column	Description	Editing
F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to flexural resistance.	No
Mcr	Moment at a section where a tensile stress of fcr is induced in the concrete.	No
fcr	Cracking strength of concrete.	No
Mn	Nominal moment nominal (Refer to Ch. 21 of A23.3-95 standard)	No
Mp	Probable moment (Refer to Ch. 21 of A23.3-95 standard)	No
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system. N.B. VisualDesign considers all tensioned rebars, even those that have not attained the yield stress fy. (The value of d that appears here does not correspond to the value of d used in the code equations, except at the beginning of continuous system)	No
dv	Distance between tension and compression resultant forces due to bending, at this location within continuous system.	No
bw	Width of concrete section web at this location within continuous system. For prestressed concrete elements: this width is reduced by total sheath diameter if there is no grout or 1/2 sheath diameter if it is grouted.	No

Column	Description	Editing
As	Area of reinforcing steel in tension. (VisualDesign considers all rebars in tension, even those that have not reached fy). However, when considering the strain – deformation compatibility in calculations, the useful area of each bar in tension is factored with this ratio: fs/fy.	No
ϱ	Percentage of steel reinforcement at this location within continuous system.	No

For a Prestressed Concrete Project

Column	Description	Editing
ωp	Ratio of reinforcing bars and prestressing reinforcement according to clause 9.6.8.5.1 of Code S6-88. See the note below.	No
Φp*fps*Aps	Factored tensile strength of prestressing cables.	No
εx	Strain in longitudinal reinforcement for the calculation of shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that takes into account the shear resistance of cracked sections.	No
c/d	c/d ratio of clause 10.1.4 - Code A23.3, relative to the balanced condition *Concrete Deformation and yield strength of steel*. Compare with c/d max below.	No
c/d max	Maximum c/d ratio.	No

For load combination with a "Service" status.
β2 : S6-00, Cl. 8.12.3
z : A23.3-95, Cl. 10.6.1

Column	Description	Editing
β2 or z	Function modified by the proportioning and type of longitudinal rebars.	No
β2 max or z max	Value limiting the proportion of longitudinal rebars.	No
Fcr	S6-00 only: Factor that controls cracking in buried structures as per clause 7.8.9.1. This column is present if a "Buried" type of structure was select in the concrete specification.	No
Mw	S6-00 only: Moment at a section where a tensile stress of 0.4 fcr is induced in the concrete.	No

Column	Description	Editing
Ms	S6-00 only: Flexural moment at a section under consideration at the SLS load.	No
fs	S6-00 only: tensile stress in reinforcing bars.	No
I eff	Effective moment of inertia.	No
I cr	Moment of inertia of the cracked section, transformed to concrete.	No

Shear Force tab - Beam and Shear Wall (1)

In tables below, you will find the definition of columns included in this spreadsheet if the selected type of continuous system is *Beam, Column* or *Shear Wall.*

Note 1: For a shear wall, the shear force is given per meter of wall length. Please refer to this topic: Calculation of required transverse reinforcement for shear walls

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Vfy max	Envelope of maximum shear (strong axis).	No
Vfy min	Envelope of minimum shear (strong axis).	No
Design load for Shear	Design load for shear at this location within continuous system.	No
εx	Strain in longitudinal reinforcement for the calculation of shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that takes into account the shear resistance of cracked sections.	No
Vc	Concrete shear resistance.	No

Column	Description	Editing
Vs	Shear resistance of steel reinforcement	No
Vr	Shear resistance Vr of the section at this location within continuous system.	No
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system.	No
dv	Distance between extreme fibre in compression and centre of gravity of bending reinforcement. VisualDesign uses the smallest calculated value, if it is relevant.	No

Bridge Evaluation only

F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to shear resistance.	No

Shear Force tab - Column

Group: Concrete Results

Column	Description	Editing
On Strong Axis		
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Vfy max	Envelope of maximum shear force on strong axis.	No
Vfy min	Envelope of minimum shear force on strong axis.	No
Design load for Shear	Design load for shear on strong axis, at this location within continuous system.	No
Vcy	Concrete shear resistance on strong axis.	No
Vsy	Shear resistance of stirrups on strong axis.	No
Vry	Shear resistance Vr of the section on strong axis, at this location within continuous system.	No

Column	Description	Editing
dy	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, on strong axis, at this location within continuous system.	No
dvy	Distance between the centroid of rebars in compression and those in tension, on strong axis. VisualDesign uses the smallest calculated value, if it is relevant.	No
On Weak Axis		
Vfx max	Envelope of maximum shear force on weak axis.	No
Vfx min	Envelope of minimum shear force on weak axis.	No
Design load for Shear	Design load for shear on weak axis, at this location within continuous system.	No
Vcx	Concrete shear resistance on weak axis.	No
Vsx	Shear resistance of stirrups on weak axis.	No
Vrx	Shear resistance Vr of the section on weak axis, at this location within continuous system.	No
dx	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, on weak axis, at this location within continuous system.	No
dvx	Distance between the centroid of rebars in compression and those in tension, on weak axis. VisualDesign uses the smallest calculated value, if it is relevant.	No

Axial Force tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Nfz max	Envelope of maximum axial force	No
Mfx max	Envelope of maximum bending moment on strong axis.	No
Nfz min	Envelope of minimum axial force	No
Mfx min	Envelope of minimum bending moment on strong axis.	No
Maximum Mf-Nf Design Load	Maximum design load due to combined bending & forces for a continuous system with variable geometry of the *Slab* type.	No
Minimum Mf-Nf Design Load	Minimum design load due to combined bending & forces for a continuous system with variable geometry of the *Slab* type.	No