VisualDesign™

Tutorial

Practical Examples

Version 2.2

Static Analysis
Steel Design
Composite Beams
Advanced Modelling
Modal & Spectral Analyses and Ductile Steel Design
Timber Design
Linear Time History Analysis
Non-Linear Time History Analysis
General Dynamic Analysis
Reinforced Concrete Design
Prestressed Concrete Design
Foundation Design
2D & 3D Moving Load Analysis

July 2005

Table of Contents

INTRODUCTION ______ **1**

EXAMPLE 1 2D Frame - Static Analysis and Steel Design ______ **7**

EXAMPLE 2 Modelling and Static Analysis of a Small Building ______ **29**

EXAMPLE 3 Steel Design ______ **53**

EXAMPLE 4 Composite Beam ______ **65**

EXAMPLE 5 Advanced Modelling ______ **93**

EXAMPLE 6 Timber Design ______ **107**

EXAMPLE 7 Modal, Spectral and Ductile Steel Design ______ **137**

EXAMPLE 8 Linear Time History Analysis ______ **161**

EXAMPLE 9 Non-Linear Time History Analysis ______ **171**

EXAMPLE 10 General Dynamic Analysis ______ **183**

EXAMPLE 11 Reinforced Concrete Design ______ **195**

EXAMPLES 12 & 13 Prestressed Concrete Design ______ **225**

Pre-tensioning ------ *227*

Post-Tensioning ------ *259*

EXAMPLES 14, 15 & 16 Foundation Design ______ **285**

Footings ------ *291*

Soil/Structure Interaction ------ *305*

Piles ------ *313*

EXAMPLES 17 & 18 Moving Load Analysis ______ **325**

3D Moving Load Analysis ------ *331*

2D Moving Load Analysis ------ *349*

I N T R O D U C T I O N

Basic Principles

Modelling a Structure

This step-by-step process includes a brief description of the different stages in creating a project with VisualDesign™.

The VDBase.mdb Database and Archiving Function

This database includes materials, shapes, reinforcing bars, cables, studs, steel decks and soils data. Users are allowed to add new data so it can become a customized database. Also, it can be shared among users: copy this database in VisualDesign™ *Sections* directory on your workstation. The database is not overwritten when the software is upgraded.

The function ***Archiving Common Objects***, which is located in the **Preferences** tab of **Project Configuration** dialog box (**File** menu) is automatically activated so all objects (standard and customized) used in your project are saved within your .vd1 or .vdz file at closing.

Project Configuration

Before you begin with the modelling, configure your project. Select **Project Configuration** under **File** menu.

Modelling

To quickly model a structure (nodes, supports, members and floors), use the **Automatic Mesh Generation** function. You will find this tool in **Structure/Tools** menu.

You can also create elements separately through spreadsheets located in **Structure** menu or directly on screen with the **Add** function (**Edit** menu).

Loadings

You must define all load case titles and types that you will be using in your project in the **Loads Definition** spreadsheet (**Loads/Load Cases** menu). Then, activate the Load Case mode and choose a load case title in Activation toolbar drop-down list box. Apply loads on your structure by double-clicking on an element on your screen. Also, you can select many elements of the same type and use the **Properties** function. Either ways, a load dialog box will appear on your screen. Enter values according to global or local axis system.

Load Combinations

You must define at least one load combination to be allowed to run an analysis with VisualDesign™.

A quick way to create load combinations is through the **Load Combination Generation Wizard**. This tool, available under **Loads/Load Combinations** menu, will generate all load combinations according to a specific building Code (American and Canadian). Load factors will also be generated and can be modified afterwards. When the generation if completed, you can disable load combinations that you do not wish to analyze at that moment by changing their statuses.

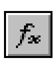
If you prefer to define load combinations yourself, select the **Load Combination Definition** spreadsheet under **Loads/Load Combinations/Definition** menu. This dialog box also includes a **Load Factors** tab. You must specify load factor for each load that is part of a load combination.

Analysis – Design - Verification

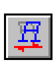
There is no limit on the number of elements included in a model. Neither concerning the number of load cases or load combinations. VisualDesign™ cannot analyse a structure that has no support nodes. If a user forgot to assign a type of shape or material to members, a warning message will be posted on the screen.

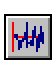
Structures can be analyzed, checked or designed. To do so, you must define an appropriate specification (steel, concrete or foundation) that is of a *design* or *verification* type. It also includes other design criteria. Specifications spreadsheets are available under **Structure/Specifications** menu.

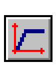
Available analyses are:

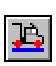
 **Static Analysis**

 **Modal Analysis**

 **Spectral Analysis**

 **Time History Analysis**

 **Non-linear Time History Analysis**

 **Moving Load Analysis**

 **Analysis and Design**

 **Design of Concrete Plates**

Results

VisualDesign™ automatically activates the "Load Combination" mode once that a static analysis is completed. You must choose a load combination title among the drop-down list box of Activation toolbar.

To look at an envelope results, activate the "Envelope" mode and select an envelope title among the drop-down list box of Activation toolbar.

Results may be viewed in many ways:

- Select the **Results** tab or **FE** (Finite Elements) tab in the **View Options** dialog box. Check the boxes that correspond to results (numerical or graphical) that you want to display on the screen;

- With the mouse, double-click on any element to call up the results dialog box for this element or select a few elements (or all of them) and press the **Properties** icon  to call up the Results spreadsheet;

- Select one of the **Results** menu headings to have access to load combinations results (nodes, members, plates, etc.) or envelopes results.

Modelling Strategy

When modelling a structure, it is important to work with strategy in order to minimize input errors and time. Here are recommended steps to model a structure in the most productive way:

- Identify reference spatial coordinates.
- Always work according to functionalities that you will be using. You must master the many **Split** functions (Multiple split, Split according to node, Split at exact position, Split with a pin connection and Split with a rigid connection) and the **Copy/Paste** function (translation, rotation and mirror) before modelling a structure.
- Before splitting a member, specify design criteria such as the Code that you will be using, the type of shape and others. End conditions are also very important before you split a member (hinged or fixed ends).
- Adjust beta angle at this step. The **Rotate** function can also be used.

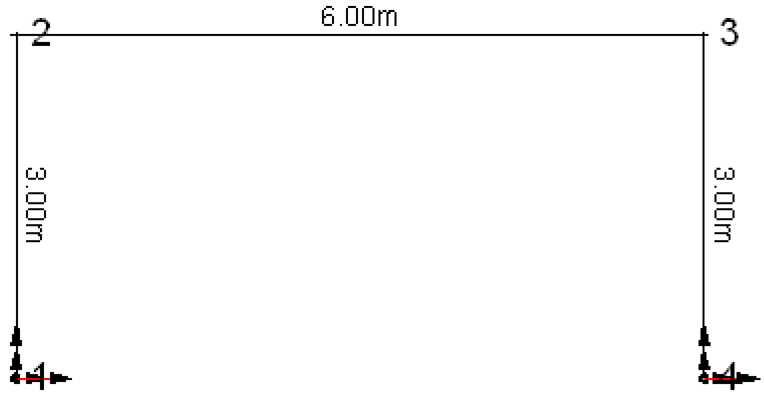
  If you cannot place the member in the right direction in space with the beta angle (it can happen for single symmetry shape), use the option "Swap Node i↔Node j" in the “Incidence” section of **Member Characteristics** dialog box.
- Once that these steps are done, you can split members the number of time that you wish to. Members will keep original characteristics (beta angle, end conditions and design criteria).
- Define load titles and types (live, Add. dead., snow, wind, etc.), load combinations, load factors and envelopes. Then, apply loads on the structure (the Loading activation mode).
- Use the **View Options** to verify the structural model and applied loads.
- If you are planning to run a seismic analysis, try to model a rough structure to avoid local vibration modes usually created by mezzanine, walkway or footbridge, but do not forget to transfer these dead loads on appropriate nodes.

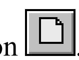
E X A M P L E 1

2D Frame – Static Analysis & Steel Design

Static Analysis

We are going to model the following 2D frame, apply loads, and run a static analysis. After, a steel design will be launched.



- Start VisualDesign™. Click on the **New Project** icon . The Structure mode is automatically activated.

Project Configuration

- Select the **Preferences** tab of **Project Configuration** (**File** menu) and uncheck all the boxes included under heading "Dialog Box Display".

- Go to the **Analyse** tab and activate a linear type of analysis.

Modelling

Nodes

- Create nodes through the **Nodes** spreadsheet (**Structure** menu). Specify nodes 1 and 4 as support nodes: Double click in the "Type" column and select option *Support*.

Nodes Spreadsheet

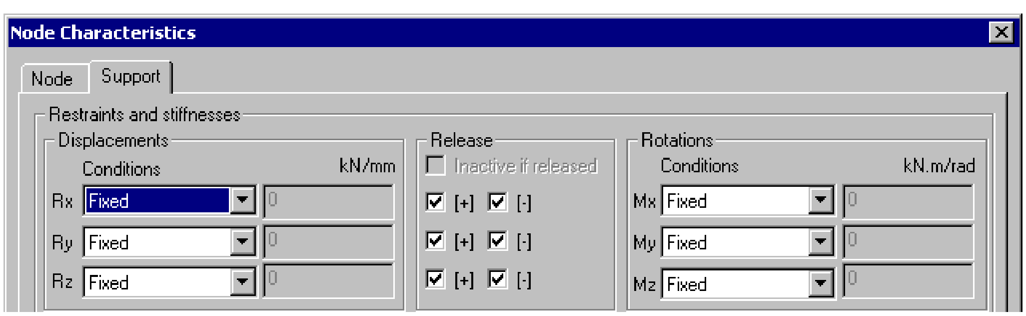
4	Number	Type	Coord. X m	Coord. Y m	Coord. Z m	ID Master No.	Linked DO
1	1	Support	0.00	0.00	0.00	0	n/a
2	2	Normal	0.00	3.00	0.00	0	n/a
3	3	Normal	6.00	3.00	0.00	0	n/a
4	4	Support	6.00	0.00	0.00	0	n/a

Members

- Select the **Member** icon on the Elements toolbar and press the **Add** icon on the Cursor toolbar. To create a member, click on a node (origin node, i) and on a second one (end node, j). Do the same to create other members. Once those elements have been created, select the **Extended selection** mode to exit the **Add** mode.

Supports

- To modify degrees of freedom for supports, activate the **Support** element on Elements toolbar and select supports. Press the **Properties** icon . In the **Support** tab, select degrees of freedom. Displacements and rotations will be fixed. Close the dialog box.



- Press the [Pg Up] key to get an isometric view of the structure.

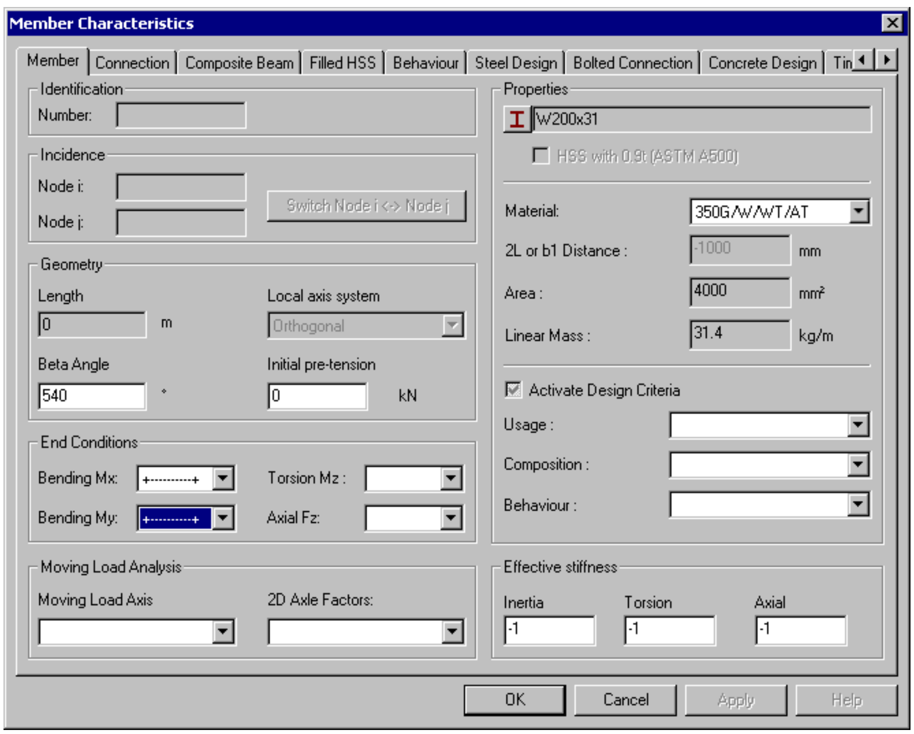
Save your File

- At this stage, save your application. To do so, press the **Save** icon . Give a name to your project and choose a directory.

Shapes, Materials, and Member Properties

- Activate the **Member** icon on Elements toolbar. Select all three members and press the **Properties** icon . The **Member Characteristics** dialog box will appear on screen.

  Notice: When many members are selected, empty fields and blank fields appear in the dialog box. Blank fields means that selected options will be assigned to all selected members. Some other blank fields include negative values but they are not actual values. To consult default values and initialized values, select only one member. This notice also applies to other type of element.



- Click on the icon and open the Steel shape selection *tree.* Expand the W200 branch and activate the radio button next to shape W200x31.

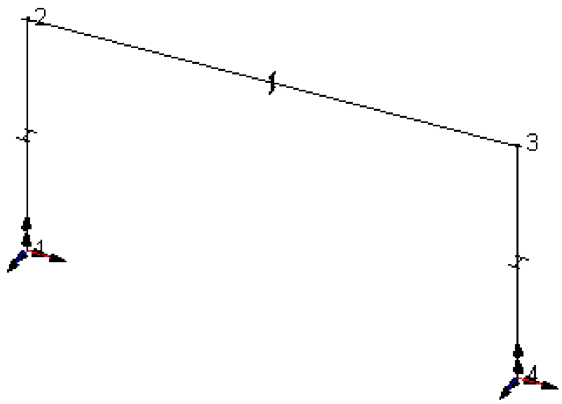
- Select a material among the Steel selection tree. Open the *G40* root branch and double click on 350G /W /WT/AT to select this material.

- Specify member end conditions: Select *fixed-fixed* end conditions (+------+) for bending on strong axis (Mx) and weak axis (My).

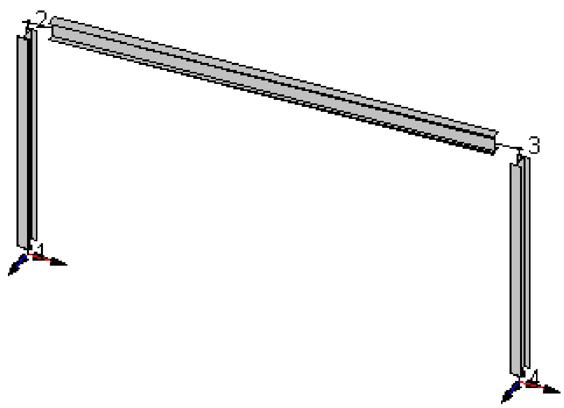
- Click OK.

View Options

- Display the shape outlines on screen: Open the **View Options** dialog box, select the **Attributes** tab, and activate the *Shape Outline* option. Press OK.



- Display members in 3D, by activating option *3D Display* in the member section of **Attributes** tab.

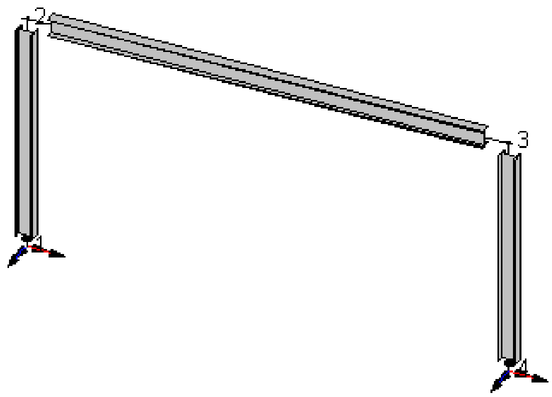


- To increase the member displayed length for 3D display, go to the Preferences tab of project Configuration dialog box and reduce the default length in the field "Fraction of L not drawn". The purpose of this option is to see the member end conditions when the *3D Display* option is activated.

Beta Angle

- We can see that columns are not well oriented. When new members are created, the default beta angle is equal to 0 degree. To modify the beta angle for columns, select them while you press down the [Ctrl] key, and press the **Properties** icon .

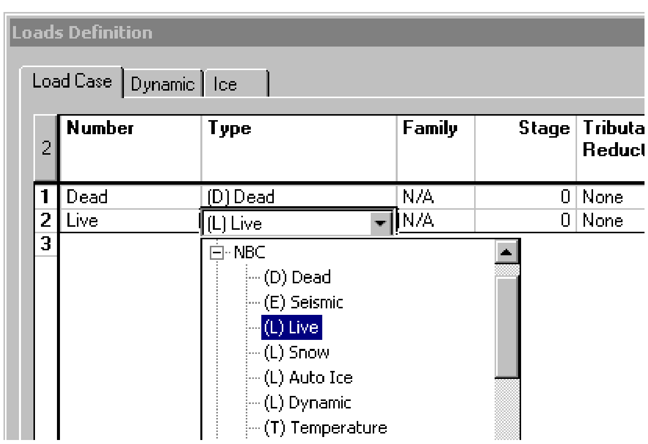
- Enter an angle of 90 in the field *Beta Angle* of **Member Characteristics** dialog box. Press OK.



Load Cases

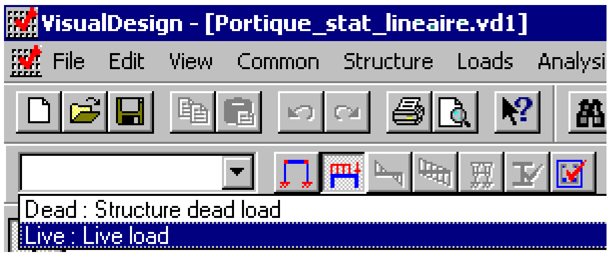
- Open the **Loads Definition** spreadsheet (**Loads** menu /**Load Cases/ Definition)**. To define load cases, you must enter a title and select a type of load for each load case.

- Insert a line in the **Loads Definition** spreadsheet. (To insert a line, select the line number 2 and press down the [Insert] key.) Double-click in the "Number" cell and enter the title *Live* for the live load case. Double-click in the "Type" cell and choose a "Live" type of load. Press OK.

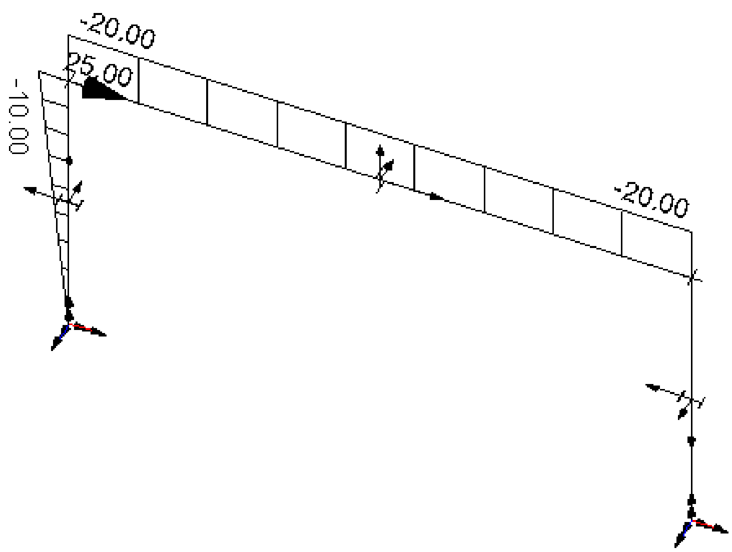


Applying Loads

- Activate the Load Case mode and select the "Live" load case. Answer *Yes* to the message: "Do you wish to save your project?" The load case name will be written at the bottom of your screen.



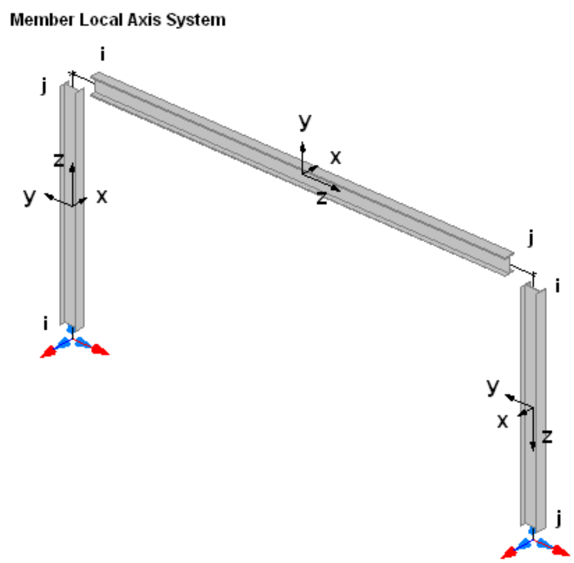
A distributed live load will be applied on the beam. A triangular load, representing wind force, will be applied to the left column. A punctual load will also be applied at the top:



Displaying Member Local Axis

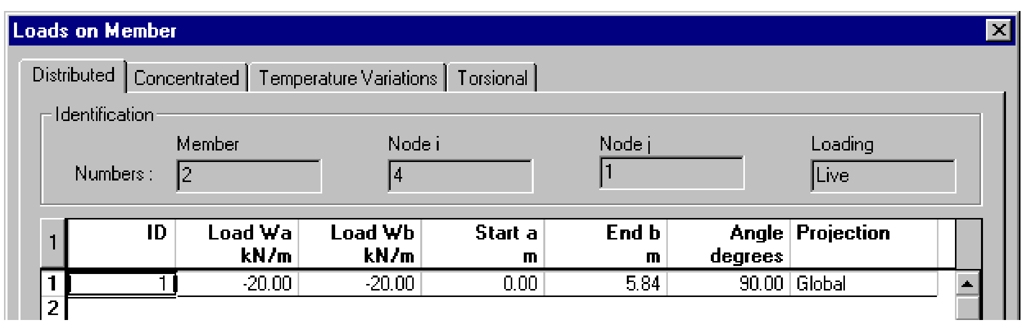
Before applying loads on columns, display member local axis system to orient loads in the right direction. The member local axis system follows the right hand rule. The local z-axis is always pointing towards node j. The local x-axis corresponds to the member strong axis, and y-axis, to the weak axis.

- Open the **View Options** and select the **Attributes** tab. Activate the option *Local Axis System* in the members section.



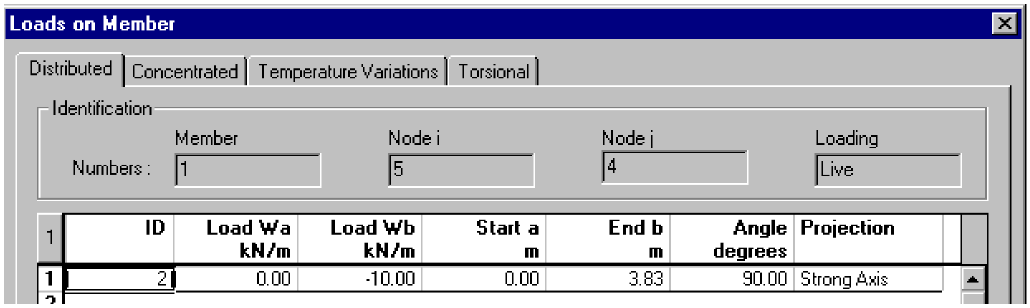
Distributed Load on Beam

- Double-click on the beam.
- In the **Load on Member** dialog box, click on number 1 and insert a line. Double-click in the *Load Wa* cell and enter –20. Double-click in the *Load Wb* cell and enter –20.
- Press OK. This loading diagram will be displayed on screen.



Triangular Load on Column

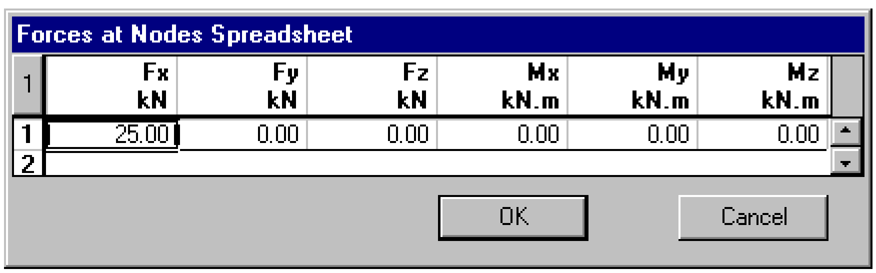
- Double-click on the first column. Load Wa represents a uniform load applied to node i and Load Wb, to node j.
- In our example, node *I* is located at support. This load must be applied towards the negative direction of local y-axis. Therefore, the load must be negative. This load is projected at 90 deg. on the column strong axis.



Concentrated Load

We are going to apply a point load at the top of the first column, towards the global x-axis.

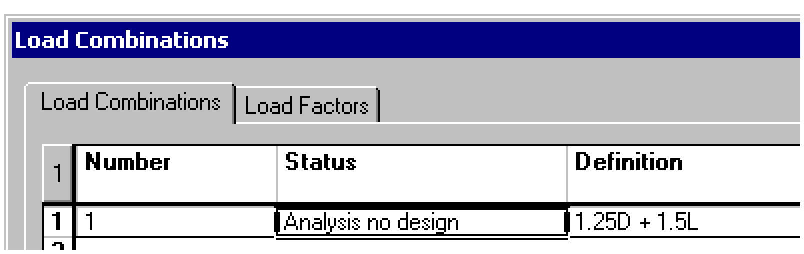
- To apply a concentrated load on a node, activate the **Node** icon and double-click on node #2. Enter 25 kN in the positive global x-axis. Press OK.



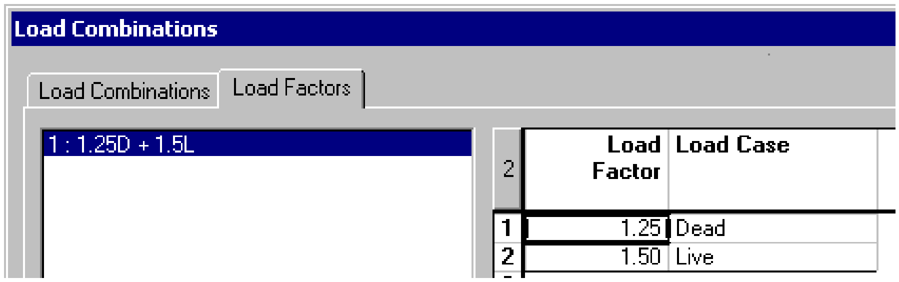
Now, at least one load combination must be defined.

Load Combination

- Go in the **Loads** menu and select **Load Combinations / Definition**. Insert a line in the **Load Combination** spreadsheet. Enter a number or name and select a status that is "Analysis no design". To learn more about load combination statuses, refer to On-Line Help, Chapter 4.



- Select the **Load factors** tab. Insert two lines in the spreadsheet. In the first line, double click in the "Load Case" column and select the *Dead* load case and enter 1.25 as load factor. In the second line, select the "Live" load case and enter 1.5. Click OK.



You are now ready to launch the static analysis.

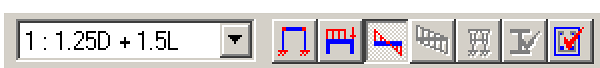
Static Analysis

- Press the **Static Analysis** icon of **Tools** toolbar. The **Static Analysis** dialog box will appear on the screen. Press the "Analyse" button. Close the dialog box when analysis is completed.

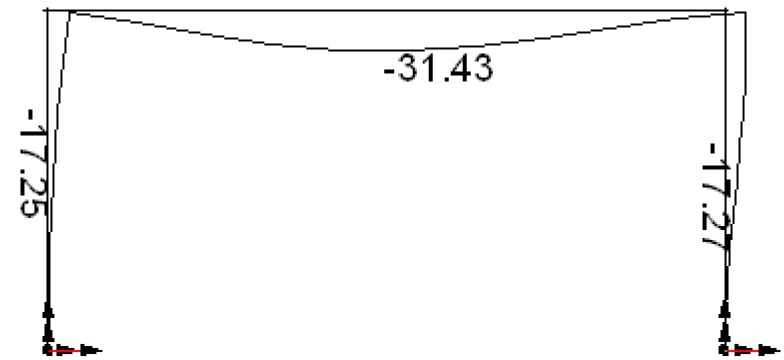
Load Combination Results

The "Load Combination" mode is automatically activated once that the static analysis is done.

- Click on the load combination title in Activation toolbar drop-down list box.

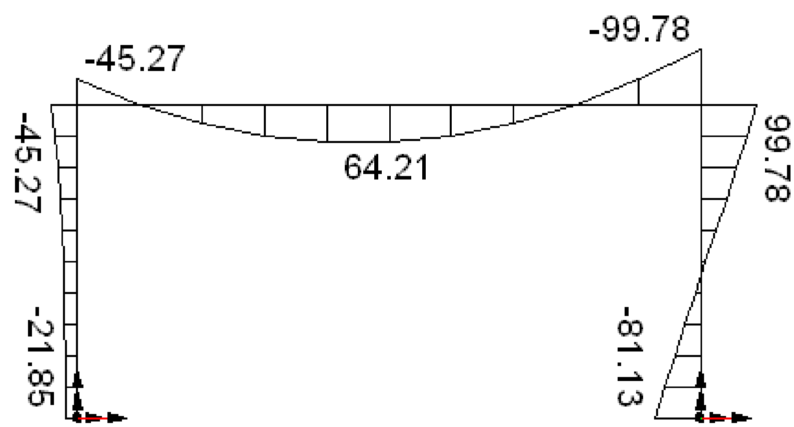


- Activate the **View Options** and select the **Results** tab. Activate the *Deflection* box. To display numerical values, activate the numerical box or press icon on **Diagram** toolbar.



- Load Combination : 1

- Now, in the same tab, uncheck the *Deflection* box and activate the Mx diagram (strong axis) with its numerical values.



- Load Combination : 1

- To adjust the amplitude of diagram, use **Diagrams** toolbar functions.

- Double-click on the beam to quickly access the **Internal forces and Deflections** spreadsheet.

Internal Forces and Deflections Spreadsheet (1)

11	Number	Shape	Position m	Bending Mx kN.m	Shear Vy kN	Axial Nz kN	Strong axis v mm	Axial w mm
1	2	W200x31	0.00	-45.27	-82.07	-60.31	-0.31	17.25
2	2	W200x31	0.60	-1.50	-63.84	-60.31	-9.62	17.20
3	2	W200x31	1.20	31.34	-45.61	-60.31	-18.96	17.16
4	2	W200x31	1.80	53.23	-27.38	-60.31	-26.44	17.11
5	2	W200x31	2.40	64.19	-9.14	-60.31	-30.81	17.07
6	2	W200x31	3.00	64.21	9.09	-60.31	-31.43	17.02
7	2	W200x31	3.60	53.29	27.32	-60.31	-28.32	16.98
8	2	W200x31	4.20	31.43	45.55	-60.31	-22.09	16.93
9	2	W200x31	4.80	-1.37	63.78	-60.31	-14.00	16.89
10	2	W200x31	5.40	-45.11	82.01	-60.31	-5.93	16.84
11	2	W200x31	6.00	-99.78	100.24	-60.31	-0.38	16.80

N. B. A few columns have been masked because values were null.

- Select all members. Go to **Results** menu and select **Load Combinations / Internal Stresses (min/max) - Members**.

Internal Stresses (min/max) in Members Spreadsheet

5	Number	Shape	σz|Nz MPa	σz|Mx MPa	σz Max MPa	σz Min MPa	τyz|Vy MPa
1	1	W200x31	-20.81	151.37	130.86	-171.89	18.76
2							
3	2	W200x31	-15.08	333.67	318.60	-348.75	82.45
4							
5	3	W200x31	-25.35	-333.67	308.61	-358.73	49.61

Note Press the **F1** key to access to the On-line Help topic that is specific for this spreadsheet. Look at columns definition.

We can see that stresses are too high. If you own the **Steel Design** module, follow the next part of this example.

Steel Design

Steel Specification

- Activate the Structure mode. Go to **Structure** menu and select **Specifications** / **Steel**. Look at the design specification parameters according to S16-01- 94 standard.

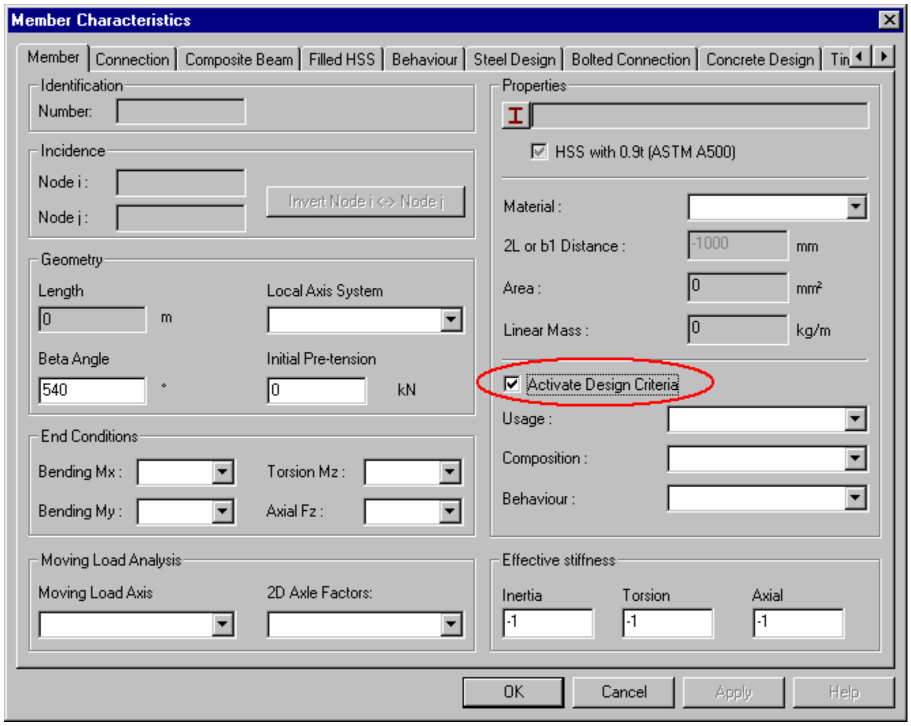
Steel Specifications Spreadsheet

8	Number	Code	Type of analysis	Optimization	Shape	Categor
1	S16-Design	CAN/CSA-S16-01	Design	Area	W	
2	S16-Vérif.	CAN/CSA-S16-01	Verification	Area	W	
3	S6-Design	CAN/CSA-S6-88	Design	Area	W	
4	S6-Vérif.	CAN/CSA-S6-88	Verification	Area	W	
5	LRFD-Design	AISC/LRFD-95	Design	Area	W	
6	LRFD-Vérif.	AISC/LRFD-95	Verification	Area	W	
7	ASD-Design	AISC/ASD-89	Design	Area	W	
8	ASD-Vérif.	AISC/ASD-89	Verification	Area	W	

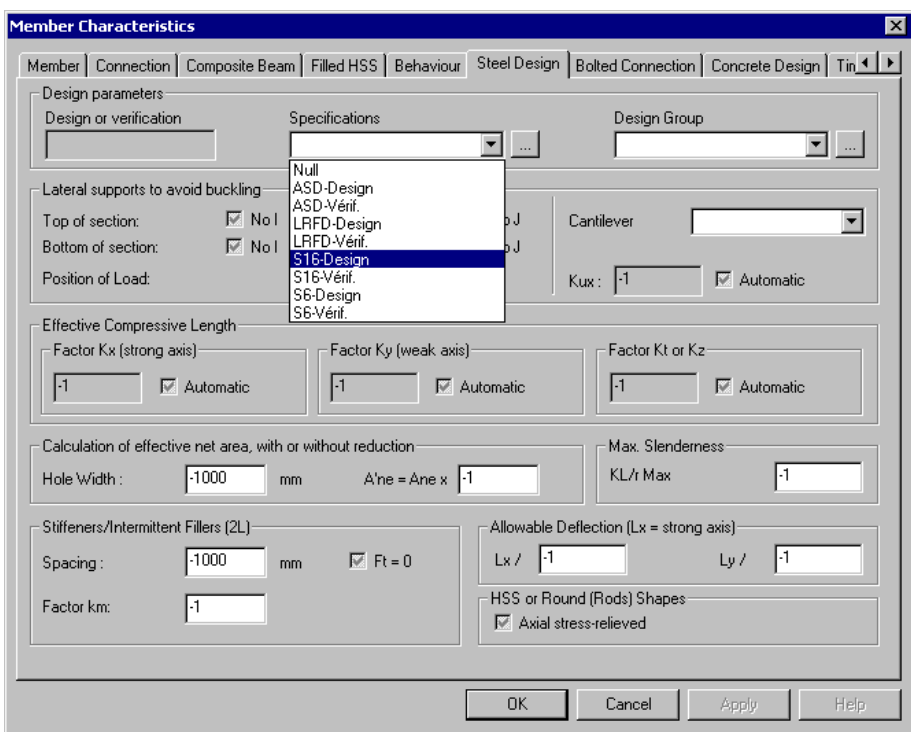
N.B. You are allowed to insert lines and create as many specifications as types of shapes included in your model.

Member Design Criteria

- Activate the **Member** icon on Elements toolbar. Select all members and press the **Properties** icon . Activate *Design Criteria* in the **Member Characteristics** dialog box.



- Then, go to the **Steel Design** tab and select the *S16-Design* specification among the drop-down list box.



- Press OK.

Design Criteria for the Beam

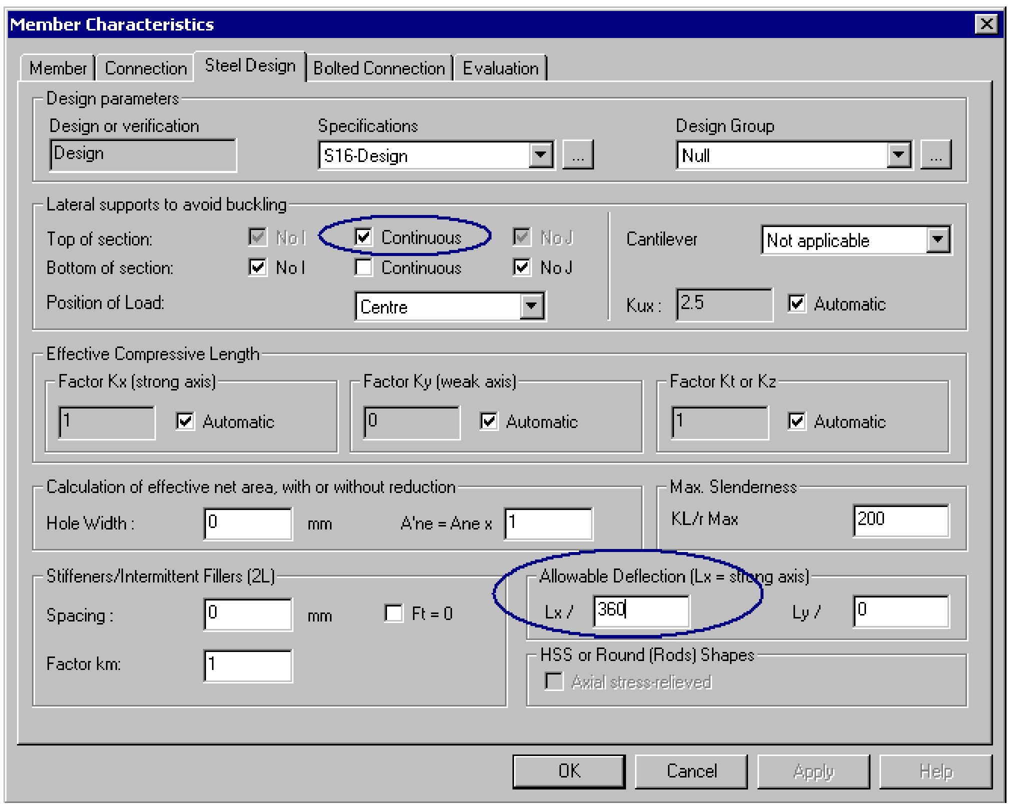
Lateral Supports

- Double click on the beam and go to the **Steel Design** tab. Supply a continuous lateral support at the top of the beam.

N.B. Lateral supports are always supplied on the member weak axis. Therefore, the effective compression length on weak axis, Ky, will be equal to zero when a continuous lateral support is supplied at the top of a beam <u>OR</u> at the bottom.

Allowable Deflection

We are fixing a deflection criterion of L/360 on strong axis.



Load Combinations

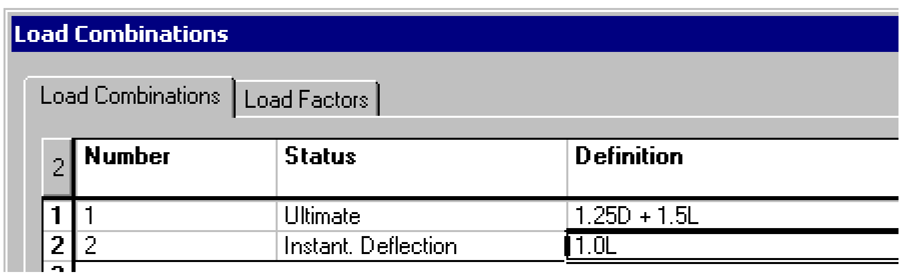
Ultimate load combination (Design)

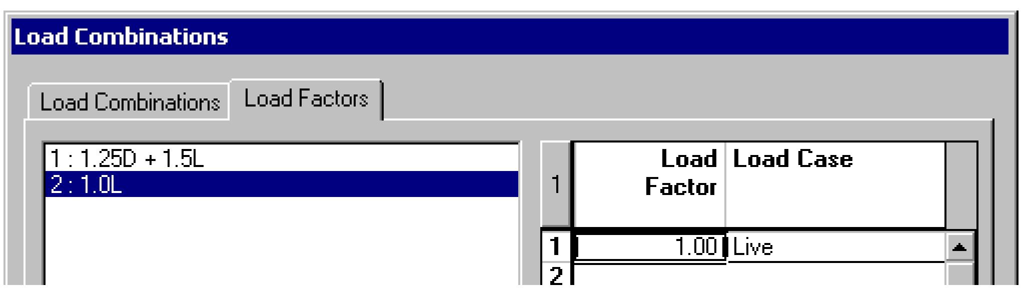
- Open the **Load Combination** spreadsheet. Modify the first load combination status to *Ultimate* because a design will be run.

"Deflection" Status

VisualDesign will choose a steel shape according to the allowable deflection criterion only if at least one load combination has a deflection status.

- Insert a line in the Load Combination spreadsheet. Double click in the "Status" column and select the status "Instantaneous Deflection". Only live load is considered.

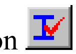




Analysis and Design

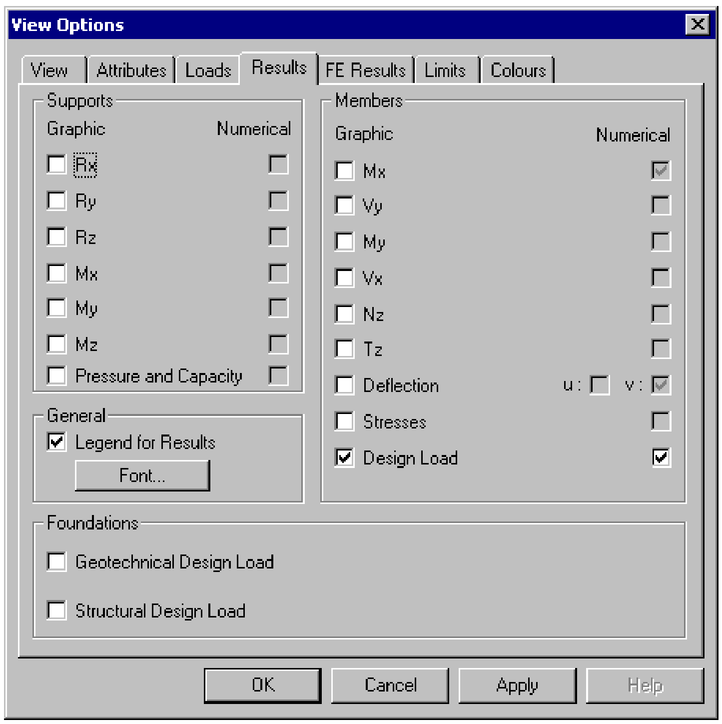
- Press the **Analysis and Design** icon  on Tools toolbar. The **Design** dialog box will be displayed on the screen. It is written that three members are going to be optimized according to the shape area. Press the "Analyse" button. Close the dialog box when analysis will be completed.

Steel Design Results

You will notice that the "Design Results" icon  is automatically activated when the design is completed.

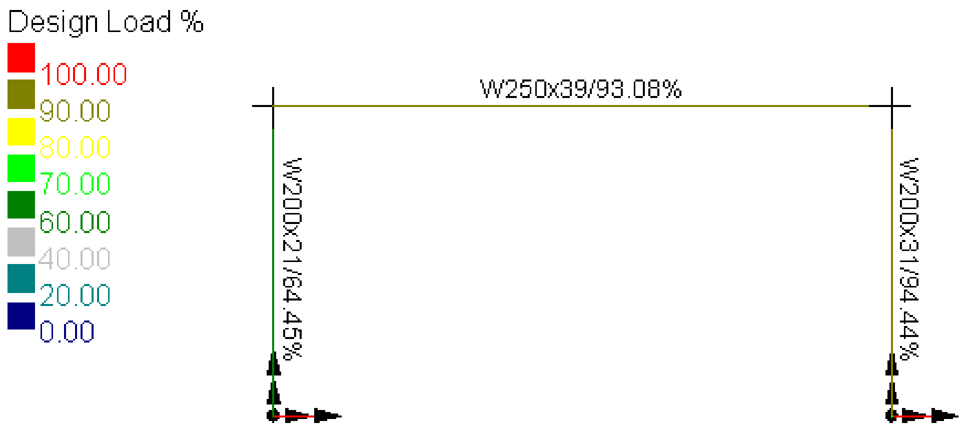
Graphical Results

- Open the **View Options** dialog box by pressing icon . Select the **Results** tab and display members' design loads by checking the *Design Load* box and "Numerical" box in the Members section.



- Modify the legend font and style by clicking on the "Font" button posted in the **Limits** tab.

- Look at the chosen shapes by activating option "Shape" in the **View** tab.



Numerical Results

Open the **Steel Design Results** spreadsheet: Select all members and press the **Properties** icon or go to **Results** menu and select **Structure Design / Steel**.

Don't forget that design results are given for the most critical load combinations (bending and compression & shear) that apply to each member. In our case, there is only one load combination. Also, the Code provision that controlled the design is indicated in the spreadsheet.

Steel Design Results Spreadsheet

3	Number	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear	Design Load Shear %	Code Provision Shear
1	1	W200x21	1	64.45	CSA S16-01 13.8.1c	1	8.86	CSA S16-01 13.4.1.1
2	2	W250x39	1	93.08	CSA S16-01 13.6	1	28.69	CSA S16-01 13.4.1.1
3	3	W200x31	1	94.44	CSA S16-01 13.8.2c	1	20.09	CSA S16-01 13.4.1.1
4								

Close

Steel Design Results Spreadsheet

3	Class Mx Bending	Class My Bending	Class Web	Class Compression	Type of Mfx	Mrx Lu=0 kN.m	Mrx Lu>0 kN.m	Lux m	ω2x	Type of Mfy
1	3	3	1	3	M-	61.45	58.10	3.00	1.61	M-
2	1	1	1	1	M-	161.60	102.36	6.00	1.48	M+
3	1	1	1	1	M+	105.52	105.52	3.00	2.22	M+
4										

Close

Steel Design Results Spreadsheet

3	Mry Luy=0 kN.m	Mry Luy>0 kN.m	Luy m	ω2y	Type of Nz	Tension Tr kN	Cr Resistance kN	Cr Stability kN	KLx m	KLy m
1	11.89	11.89	0.00	1.00	Compression	853.65	853.65	369.34	3.00	3.00
2	39.06	39.06	0.00	1.00	Compression	1549.80	1549.80	1193.54	6.00	0.00
3	29.55	29.55	0.00	1.00	Compression	1260.00	1260.00	582.83	3.00	3.00
4										

Close

Steel Design Results Spreadsheet

3	KLt,KLz m	KL/rx	KL/ry	KL/rz	KL/r max	T/C ratio %	Net Area mm²	Vrx kN	Vry kN	Trz kN.m	Results Resistance
1	3.00	35.10	98.58	0.00	98.58	0.00	2710.00	236.34	211.02	1.17	Sufficient
2	6.00	54.29	0.00	0.00	54.29	0.00	4920.00	458.11	359.50	3.13	Sufficient
3	3.00	33.86	93.70	0.00	93.70	0.00	4000.00	380.64	279.42	2.42	Sufficient
4											

Close

Steel Design Results Spreadsheet

3	Results Deflection	Combination Mx	Lx m	Deflection (Mx) mm	Lx /	Combination My	Ly m	Deflection (My) mm	Ly /
1	n/a		0.00	0.00	5000		0.00	0.00	5000
2	Ok	2	4.54	-9.74	466		0.00	0.00	5000
3	n/a		0.00	0.00	5000		0.00	0.00	5000
4									

Close

Member Forces and Deflections

Select a line in the **Design Results** spreadsheet and click this icon to look at the member forces and deflections.

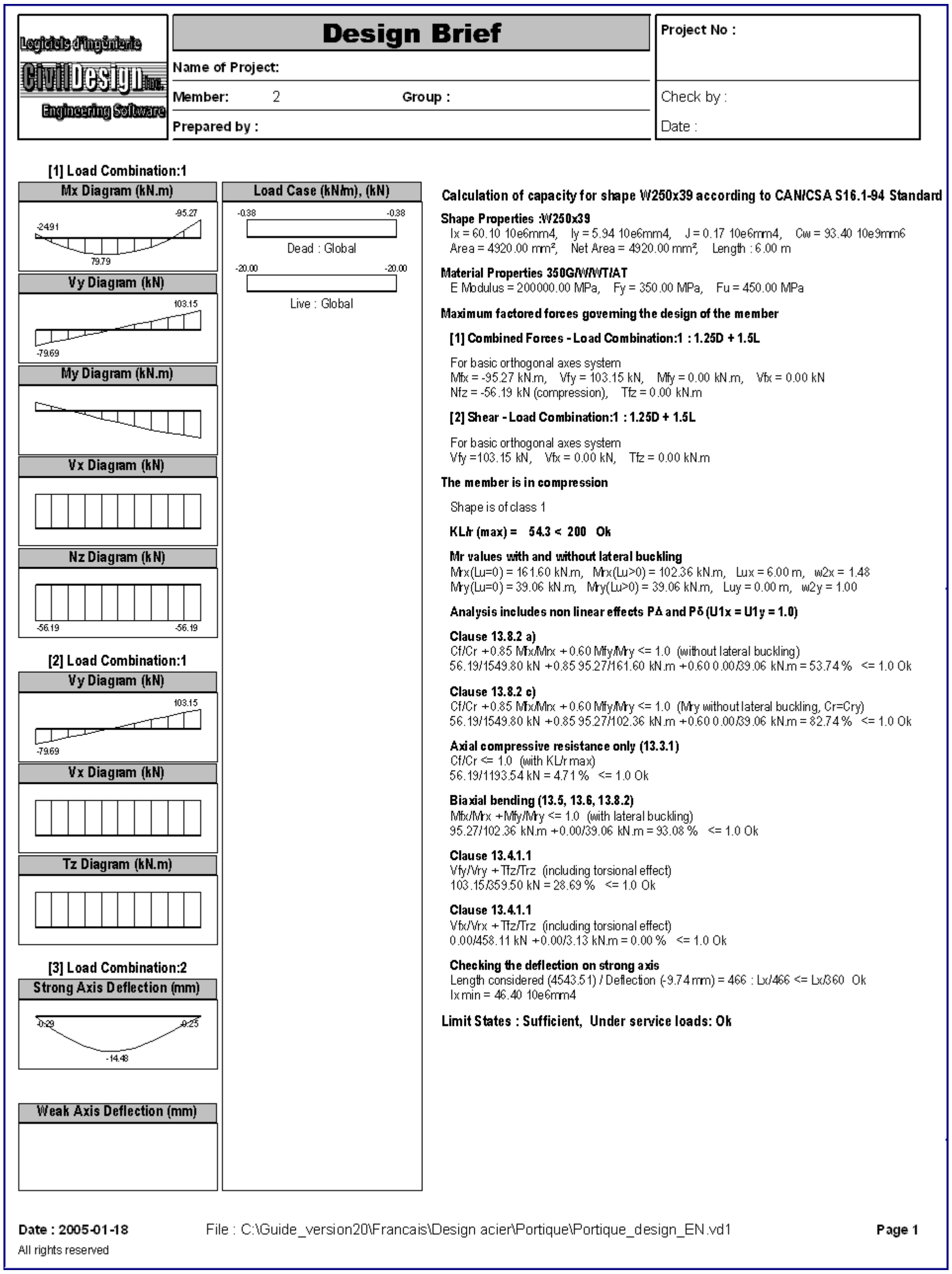
Internal Forces and Deflections Spreadsheet (1)

11	Number	Shape	Position m	Bending Mx kN.m	Shear Vy kN	Axial Nz kN	Strong axis v mm	Axial w mm
1	2	W250x39	0.00	-24.91	-79.69	-56.19	-0.44	14.94
2	2	W250x39	0.60	17.82	-61.41	-56.19	-7.54	14.91
3	2	W250x39	1.20	49.55	-43.13	-56.19	-14.05	14.88
4	2	W250x39	1.80	70.22	-24.84	-56.19	-19.01	14.84
5	2	W250x39	2.40	79.79	-6.56	-56.19	-21.81	14.81
6	2	W250x39	3.00	78.26	11.73	-56.19	-22.16	14.77
7	2	W250x39	3.60	65.63	30.01	-56.19	-20.10	14.74
8	2	W250x39	4.20	41.91	48.29	-56.19	-16.02	14.70
9	2	W250x39	4.80	7.14	66.58	-56.19	-10.63	14.67
10	2	W250x39	5.40	-38.61	84.86	-56.19	-4.96	14.64
11	2	W250x39	6.00	-95.27	103.15	-56.19	-0.39	14.60

Note: Some columns are masked because values were null.

Design Brief

Selected a line in the Steel Design Results spreadsheet and print a member design brief by clicking on this icon . Get a print preview with this icon .



Logiciels d'ingénierie CivilDesign Inc. Engineering Software

Design Brief

Name of Project:

Member: 2 Group :

Prepared by :

Project No :

Check by :

Date :

[1] Load Combination:1

Mx Diagram (kN.m)

-24.91 -95.27 79.79

Vy Diagram (kN)

103.15 -79.69

My Diagram (kN.m)

Vx Diagram (kN)

Nz Diagram (kN)

-56.19 -56.19

[2] Load Combination:1

Vy Diagram (kN)

103.15 -79.69

Vx Diagram (kN)

Tz Diagram (kN.m)

[3] Load Combination:2

Strong Axis Deflection (mm)

-0.29 -0.23 -14.48

Weak Axis Deflection (mm)

Load Case (kN/m), (kN)

-0.38 -0.38

Dead : Global

-20.00 -20.00

Live : Global

Calculation of capacity for shape W250x39 according to CAN/CSA S16.1-94 Standard

Shape Properties :W250x39
Ix = 60.10 10e6mm4, Iy = 5.94 10e6mm4, J = 0.17 10e6mm4, Cw = 93.40 10e9mm6
Area = 4920.00 mm², Net Area = 4920.00 mm², Length : 6.00 m

Material Properties 350G/W/WT/AT
E Modulus = 200000.00 MPa, Fy = 350.00 MPa, Fu = 450.00 MPa

Maximum factored forces governing the design of the member

[1] Combined Forces - Load Combination:1 : 1.25D + 1.5L

For basic orthogonal axes system
Mfx = -95.27 kN.m, Vfy = 103.15 kN, Mfy = 0.00 kN.m, Vfx = 0.00 kN
Nfz = -56.19 kN (compression), Tfz = 0.00 kN.m

[2] Shear - Load Combination:1 : 1.25D + 1.5L

For basic orthogonal axes system
Vfy =103.15 kN, Vfx = 0.00 kN, Tfz = 0.00 kN.m

The member is in compression

Shape is of class 1

KL/r (max) = 54.3 < 200 Ok

Mr values with and without lateral buckling
Mrx(Lu=0) = 161.60 kN.m, Mrx(Lu>0) = 102.36 kN.m, Lux = 6.00 m, w2x = 1.48
Mry(Lu=0) = 39.06 kN.m, Mry(Lu>0) = 39.06 kN.m, Luy = 0.00 m, w2y = 1.00

Analysis includes non linear effects PΔ and Pδ (U1x = U1y = 1.0)

Clause 13.8.2 a)
Cf/Cr +0.85 Mfx/Mrx +0.60 Mfy/Mry <= 1.0 (without lateral buckling)
56.19/1549.80 kN +0.85 95.27/161.60 kN.m +0.60 0.00/39.06 kN.m = 53.74 % <= 1.0 Ok

Clause 13.8.2 c)
Cf/Cr +0.85 Mfx/Mrx +0.60 Mfy/Mry <= 1.0 (Mry without lateral buckling, Cr=Cry)
56.19/1549.80 kN +0.85 95.27/102.36 kN.m +0.60 0.00/39.06 kN.m = 82.74 % <= 1.0 Ok

Axial compressive resistance only (13.3.1)
Cf/Cr <= 1.0 (with KL/r max)
56.19/1193.54 kN = 4.71 % <= 1.0 Ok

Biaxial bending (13.5, 13.6, 13.8.2)
Mfx/Mrx +Mfy/Mry <= 1.0 (with lateral buckling)
95.27/102.36 kN.m +0.00/39.06 kN.m = 93.08 % <= 1.0 Ok

Clause 13.4.1.1
Vfy/Vry +Tfz/Trz (including torsional effect)
103.15/359.50 kN = 28.69 % <= 1.0 Ok

Clause 13.4.1.1
Vfx/Vrx +Tfz/Trz (including torsional effect)
0.00/458.11 kN +0.00/3.13 kN.m = 0.00 % <= 1.0 Ok

Checking the deflection on strong axis
Length considered (4543.51) / Deflection (-9.74 mm) = 466 : Lx/466 <= Lx/360 Ok
Ix min = 46.40 10e6mm4

Limit States : Sufficient, Under service loads: Ok

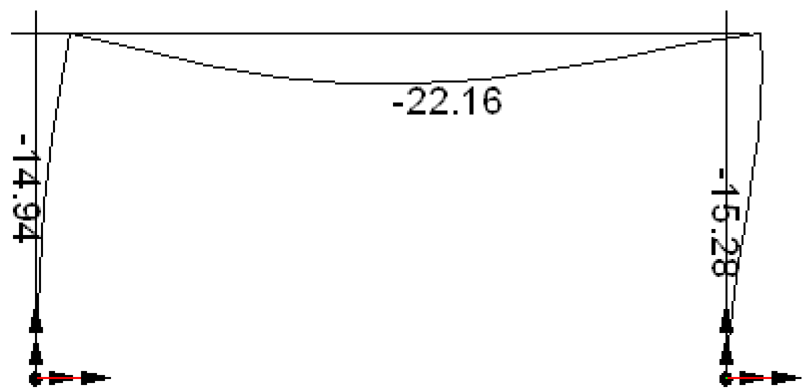
Date : 2005-01-18 File : C:\Guide_version20\Francais\Design acier\Portique\Portique_design_EN.vd1 Page 1

All rights reserved

Load Combination Results

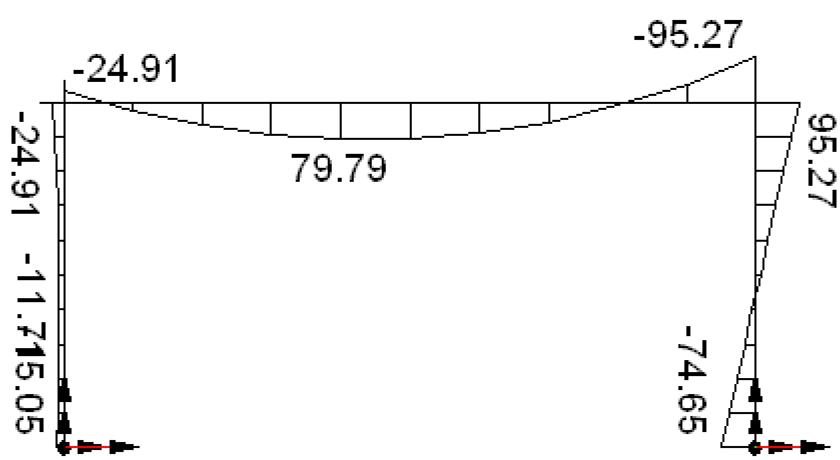
- To look at forces and deflection diagrams, activate the Load Combination mode and choose a combination title in the Activation toolbar list box.
- Open the View Options. Go to the **Results** tab and activate the diagram that you want to look at.

Deflection:



- Load Combination : 1

Bending on strong axis:



- Load Combination : 1

EXAMPLE 2

Modelling, Linear & Non-linear Static Analysis of a 2-Storied Building

Modelling

We are going to model a small two-storied building. The lateral resisting system will be bracings. Material is steel. We will run a static analysis of this building and look at results.

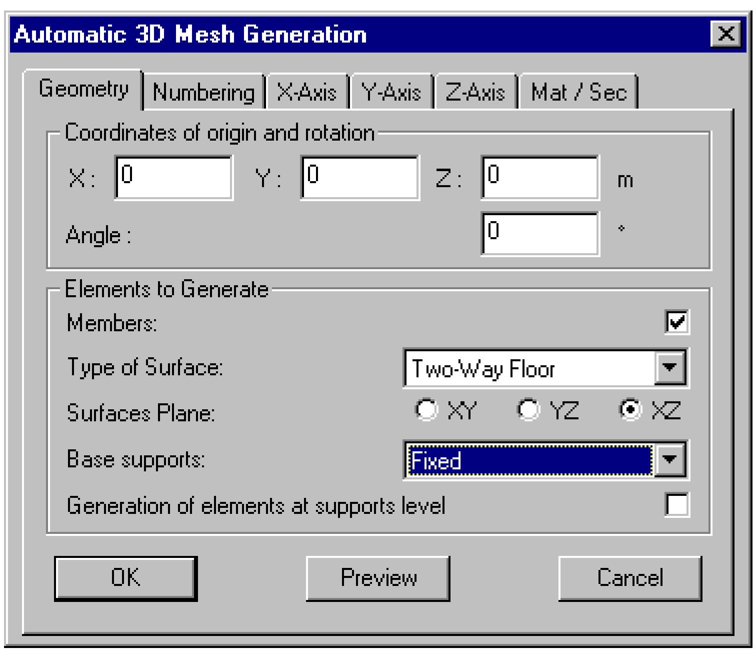
- Press the icon **New Project** of Standard toolbar.

Project Configuration

- Select **Project Configuration** of **File** menu. Enter general information about the project in the **General** tab. Look at default values in the **Preferences** tab. Then, go to the **Analysis** tab and activate a linear static analysis. Press OK.

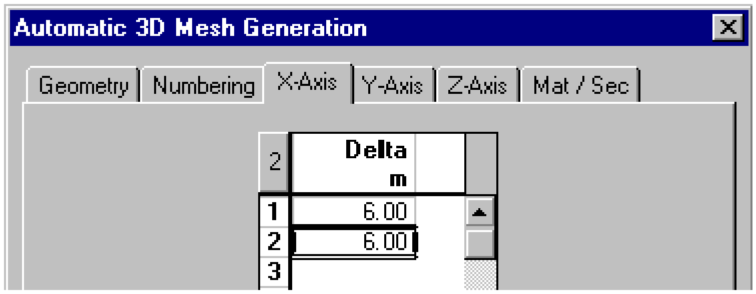
Automatic Mesh Generation

- Use the tool **Automatic Mesh Generation** by pressing the icon . Enter the following parameters in the *Geometry* tab:



As indicated in the dialog box, the generated structure will have two-way floors in the XZ plane and fixed supports. Members will be generated at superior levels (Above supports).

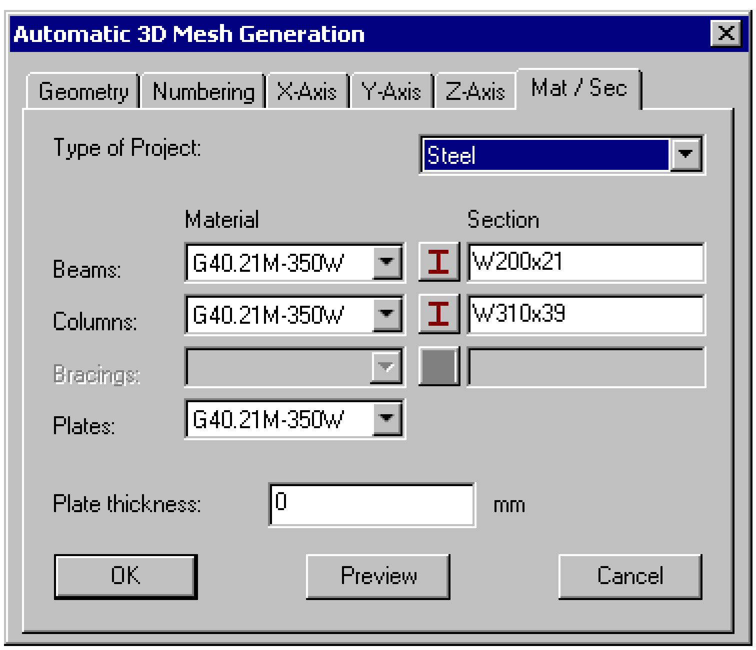
- Go to the **X Axis** tab. Select line 1 and insert two lines using your keyboard [Insert] key. Select column "Delta" and press your right mouse button. Select the **Replace** function and enter "6". The building will have two bays of 6 meters of width in the x direction.



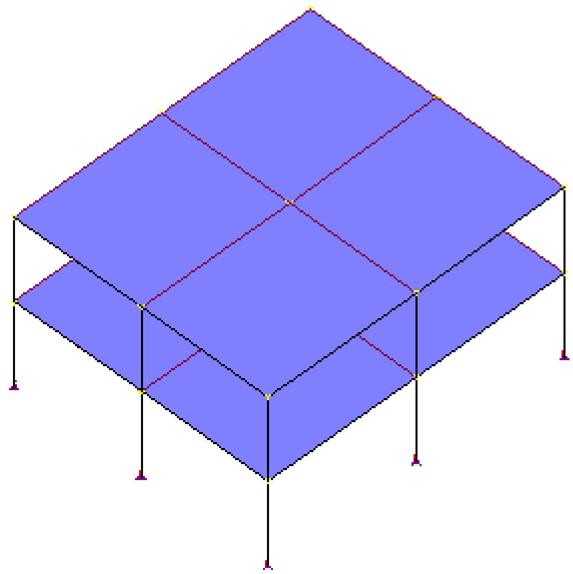
- Go to the **Y Axis** tab. Insert two lines. Select column "Delta" and press your right mouse button. Select function "Replace" and enter "4". The building will have two storeys of 4 meters high centre-to-centre of beams.

- Go to the **Z Axis** tab. Insert two lines. Select column "Delta" and press your right mouse button. Select function "Replace" and enter "7". The building will have two bays of 7 meters of width in the z direction.

- Go to the **Mat / Sec** tab. Select preliminary shapes for columns and beams. Choose steel grades.

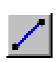


- Press OK. To get an isometric view of the structure, press the [Pg Up] key on the numerical keyboard or choose function **Camera** from **View** menu. Activate the 45 degrees radio button and press OK.



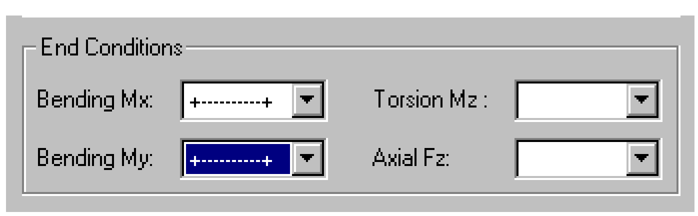
- Save your project: Go to **File** menu and select **Save as**. Choose a directory and give a name to your .vd1 file.

Members' End Conditions

- Activate **Member**  icon of Elements toolbar.

Columns

- Go to **Edit** menu and select function **Select** /**Columns**. Press the **Properties**  icon to call up the **Characteristics of the Member** dialog box.
    - Specify column end conditions as "Fixed-Fixed" for Mx and My bending.



    - Press OK.

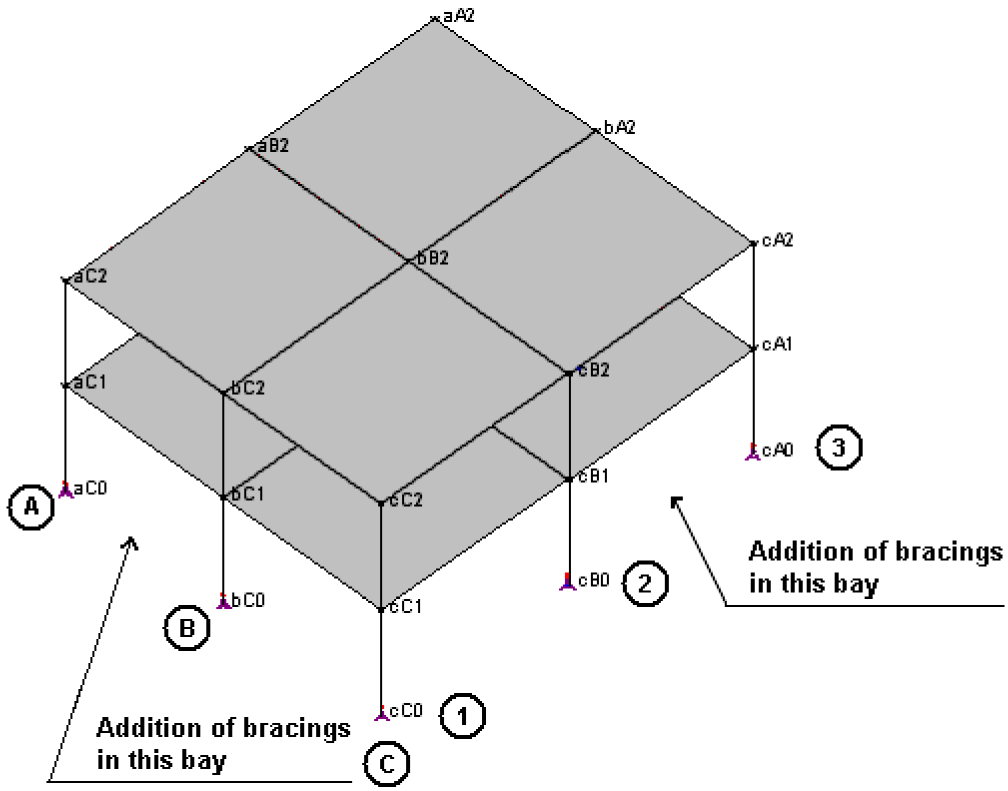
Beams

- In the **Edit** menu, select function **Select / Beams**. Click on the **Properties** icon and in the **Member Characteristics** dialog box.
    - ♦ Specify beam end conditions as *Hinged-Hinged* (o-------o) for Mx and My bending.
    - ♦ Press OK.

View Members' End Conditions

- Open the **View Options**. Go to the **Attributes** tab and activate the option *End Conditions*.

Add Bracings



- Display the node numbers through the **View** tab of **View Options** dialog box . Check the "Number" box in the "Node" section.

- Go to **File** menu and select **Project Configuration**. In the *Preferences* tab, uncheck boxes in *Dialog Box Display* section (for faster editing, dialog box will not appear on the screen each time that you add a new element). Bracing properties will be defined all together when they will be all created.

- Activate **Member** icon of Elements toolbar and activate the **Add** function by clicking on icon .

- Click on node cA2 and then on node cB1. Do the same for nodes cB2 and cA1, cA1 and cB0, cB1 and cA0. To exit the **Add** mode, click on icon **Restricted window** or **Extended window** . If the display of node numbers is not clear, refresh the screen display by selecting **Refresh** in **Window** menu.

- Define bracing properties: While keeping the [Ctrl] key pressed down, select the four members and click on the **Properties** icon. Choose an L102x102x13 shape, a 350W steel grade, and select *Hinged-Hinged* end conditions in the **Member Characteristics** dialog box.

Important Always define bracing end conditions BEFORE creating the pin connection because you will generate an hinge at the centre of bracings and this will cause instability in the structural model. When analyzing this model you will obtain a warning message (Null pivot in the stiffness matrix = mechanism).

- Create a pin connection at bracing junction: Select the two crossing members and click on the **Pin Connection** icon of **Split** toolbar. Do the same for the other bracing.

Bracings are now created in the z direction.

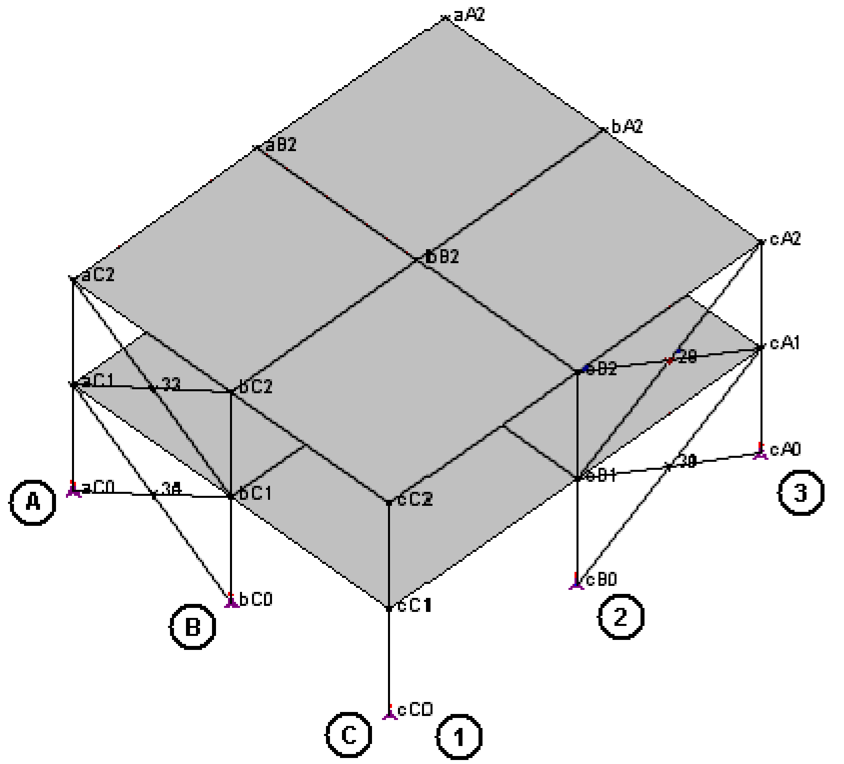
- Repeat the steps to create bracings on axis 1, between axes A and B.

Model Rigid Frames

We are going to create rigid frames for axes that have no bracings. There are four of them.

- Select beams located on axes A, B, 2, and 3 and press the **Properties** icon. Modify Mx and My beam end conditions to *fixed-fixed* in the **Member Characteristics** dialog box. Press OK.

The model is now completed.

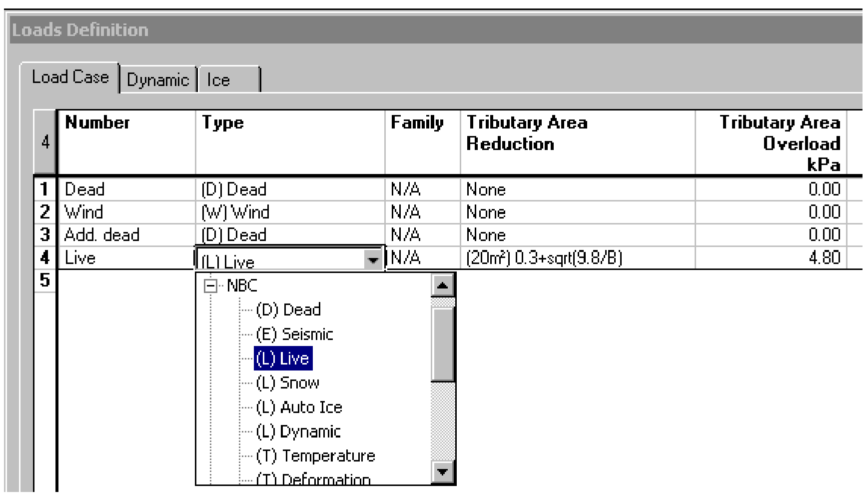

aA2
bA2
aB2
cA2
aC2
bB2
cA1
aC1
33
bC2
cB2
28
cA0
30
A
aC0
34
bC1
cC2
cB1
3
cB0
bC0
2
B
cC1
cC0
C
1

Loads

Definition of Load Cases

Load case titles and types must be defined in the **Loads Definition** spreadsheet. An additional dead load and a live load will be applied on floors and beams. A wind load will be applied to columns.

- Go to **Loads/ Load Cases /Definition** and insert three lines in the **Loads Definition** spreadsheet. (The first line, structure dead load, is automatically created and is not editable). Enter the following data:



Reduction of Tributary Areas: A live load reduction is applied to floors. Double-click in column *Tributary Area* and choose the appropriate formula (See CNB – 2005). Enter the floor overload, 4.8kPa, in the *Overload* column for the calculation of live load reduction for column.

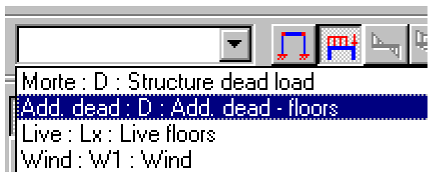
Important The value entered in the "Overload" column is only used by the software to calculate the live load reduction that will be applied to columns. This overload input DOES NOT replace the step that consists of applying loads graphically on the structure as you will see further on.

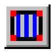
- Press OK to save data.

Applying Loads

Additional Dead Load on Floors

- Activate the Loading mode on Activation toolbar. Click on the arrow in the **Title Selection** box and select the "Add. dead" title in the drop-down list box.

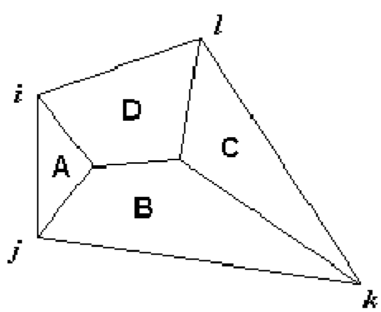


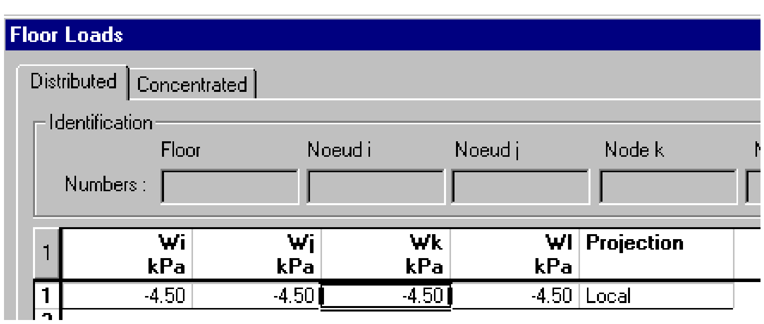
- Activate the floor icon  of Elements toolbar.

- With the cursor, draw a window around the whole structure to select all floors. Press the **Properties**  icon to call up the **Load on Floors** dialog box.

- Select the **Distributed** tab. Click on line 1 and press the [Insert] key. Wi, Wj, Wk and Wl represent load that is applied at each floor corner. Our building has concrete floors. The dead weight is a distributed load of 4.5 kPa. Double-click in the *Wi* cell and enter -4.5. Click in the *Wj* cell and you will notice that the same load is automatically entered in other cells.

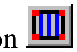
(When we automatically generated the structure, we specified two-way floor system. So floor loads will be distributed as shown below.)



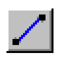
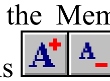


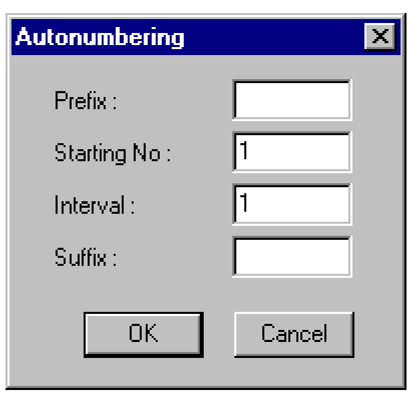
- Press OK to save inputs. Loads are displayed on your screen if the display of floor outline is activated in the **View** tab (**View Options** dialog box).

Live Loads on Floors

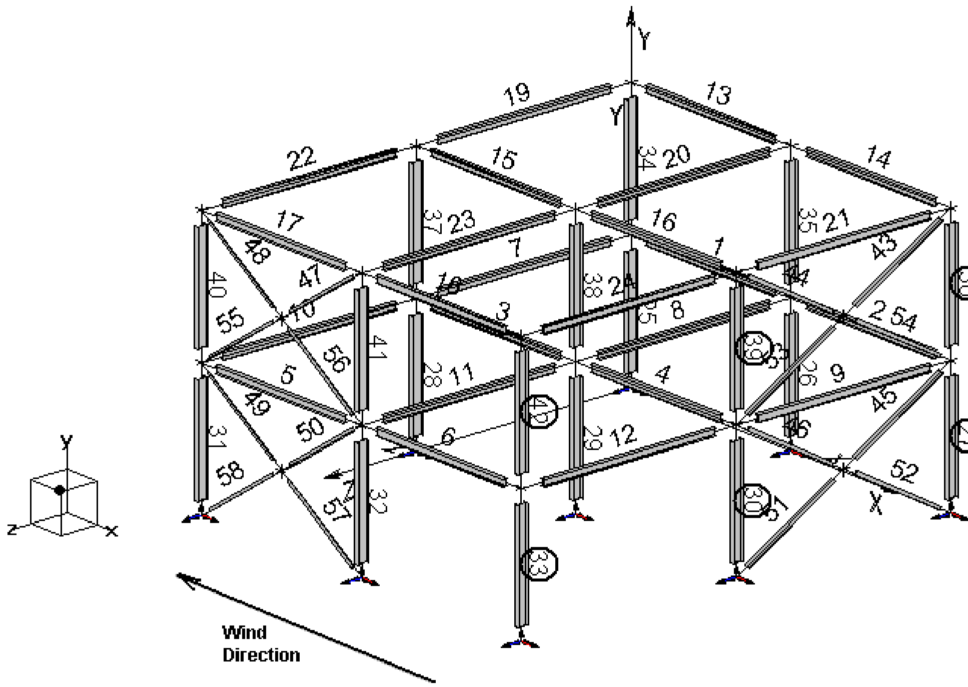
- Choose the "Live" load case on Activation toolbar. (The floor icon  is still activated on Elements toolbar).
- With the cursor, draw a window around the whole structure to select all floors. Press the **Properties**  icon to call up the **Floor Loads** dialog box.
- In the **Distributed** tab, select line 1 and press the [Insert] key. The floor live load is 4.8 kPa. Double-click in the *Wi* cell and enter -4.8. Click in the *Wj* cell.
- Press OK.

Wind Loads on Members

- Select the Choose "Wind" load case on Activation toolbar.
- Activate the **Member**  icon of Elements toolbar.
- Display member numbers: Open the **View Options** dialog box  and select the **View** tab. Check the *Number* box in the Members section. Increase or reduce font size using the View toolbar icons .
- Re-number members. To do so, activate the Structure mode and draw a window around the whole structure to select all members. Go to **Structure** / **Members**.
- In the **Members** spreadsheet, select the *Number* column title and right click to open the contextual menu. Select function **Auto numbering.** Keep default setting. Click OK.

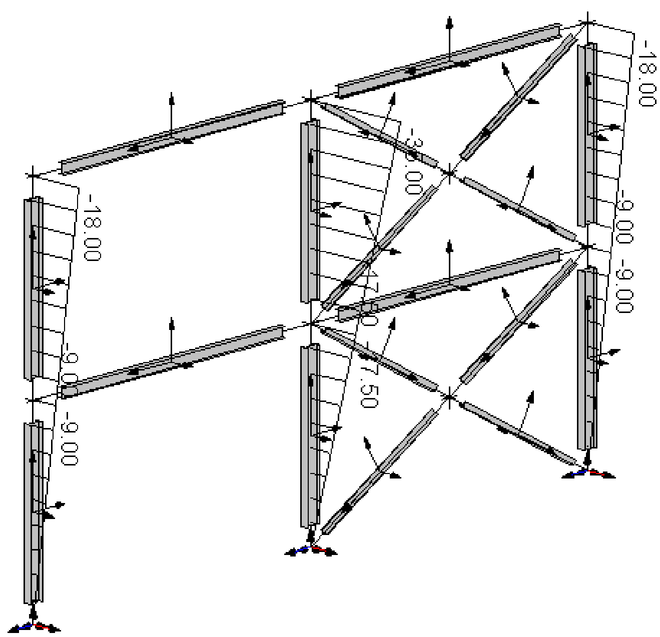


We are going to apply trapezoidal wind loads on columns #36, 39, 42, 27, 30 & 33. Activate the member *3D Display* option through the **Attributes** tab of **View Options**. We can see that the direction of wind loads points towards the negative global x-axis.

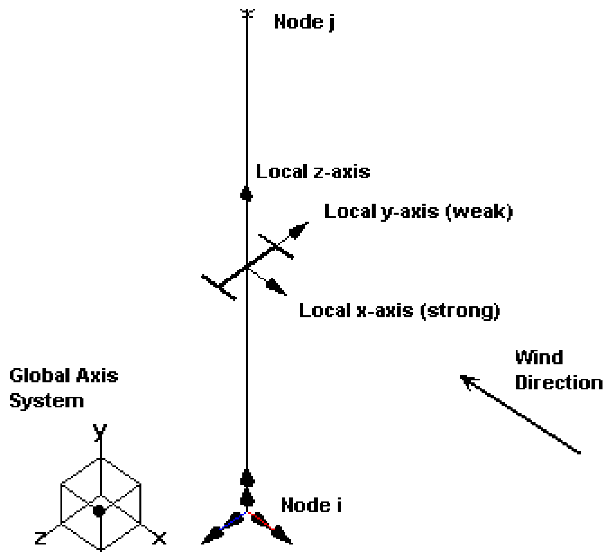


- To apply loads in the right direction, display the members' local axis system through the **Attributes** tab of **View Options**.

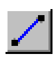
Wind loads will be applied to columns located on axis C, as shown below:

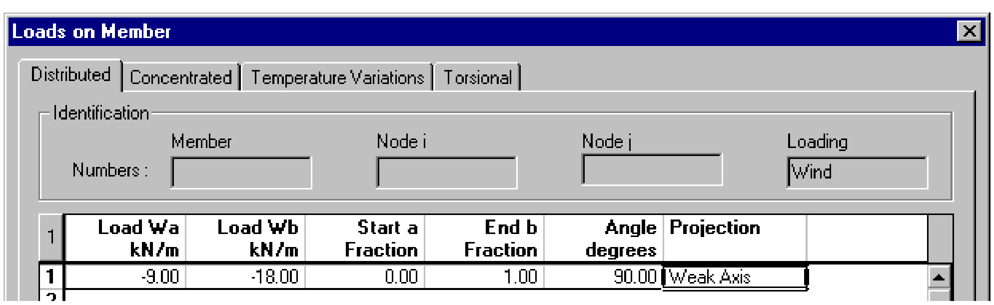


The "z" component of member local axis is always pointing towards node j. Therefore, we can see the member incidence, i.e., the position of nodes i and j. It is useful when applying loads on columns.



The wind loads will be applied in the negative direction of local x-axis and will be projected on the column weak axis (local y-axis) at an angle of 90 degrees.

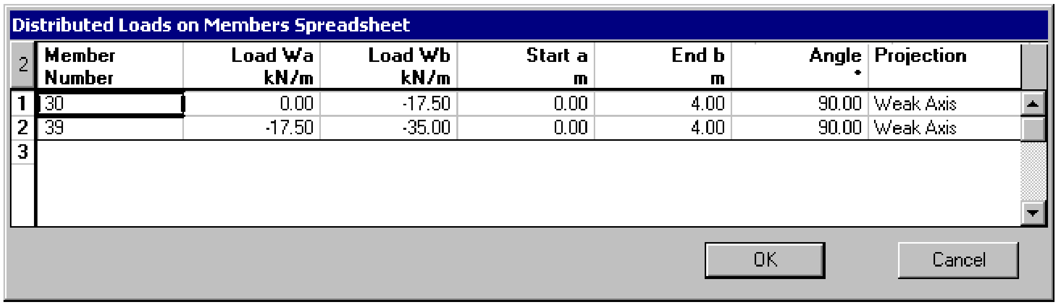
- Activate the Load Case mode and select the "Wind" load case.
- Activate the **Member**  icon of Elements toolbar.
- While pressing down the [Ctrl] key, select members 36 and 42 and click on the **Properties**  icon.
- In the **Distributed** tab of **Loads on Member** dialog box, insert a line and enter the following data. Wind load is projected at an angle of 90 degrees on the member weak axis.



- Click OK.

- While pressing down the [Ctrl] key, select members 27 and 33 and click on the **Properties** icon. Enter a trapezoidal load varying from 0 kN/m at node i to –9 kN/m at node j.

- Select member 39 and 30, and press the short-cut key **[Ctrl] + H** to open the default spreadsheet. Enter the following loads on this continuous column. Click OK.



Distributed Loads on Members Spreadsheet

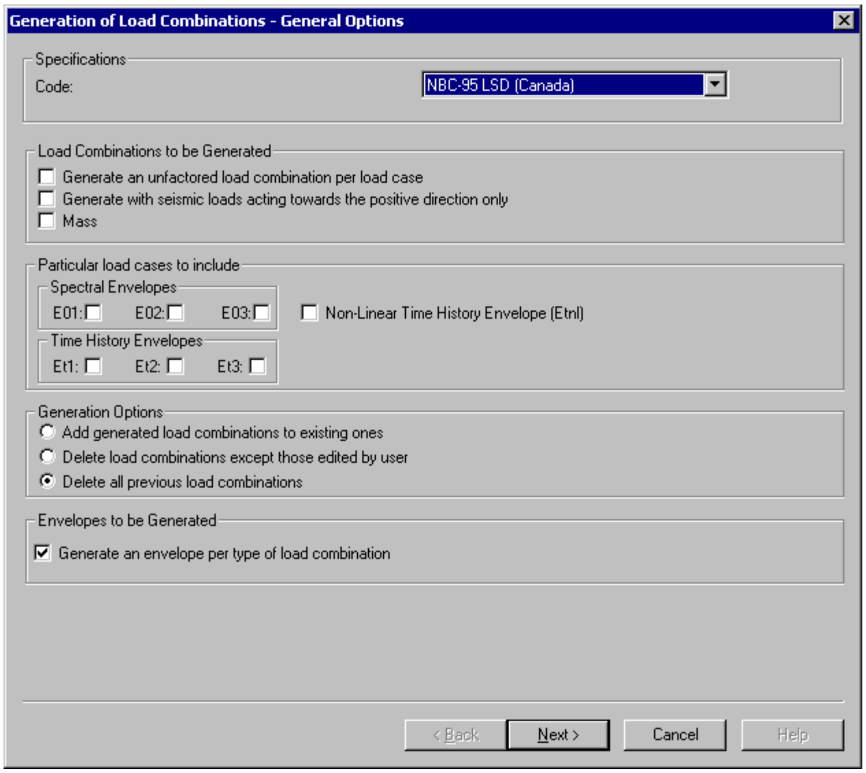
2	Member Number	Load Wa kN/m	Load Wb kN/m	Start a m	End b m	Angle •	Projection
1	30	0.00	-17.50	0.00	4.00	90.00	Weak Axis
2	39	-17.50	-35.00	0.00	4.00	90.00	Weak Axis
3							

OK Cancel

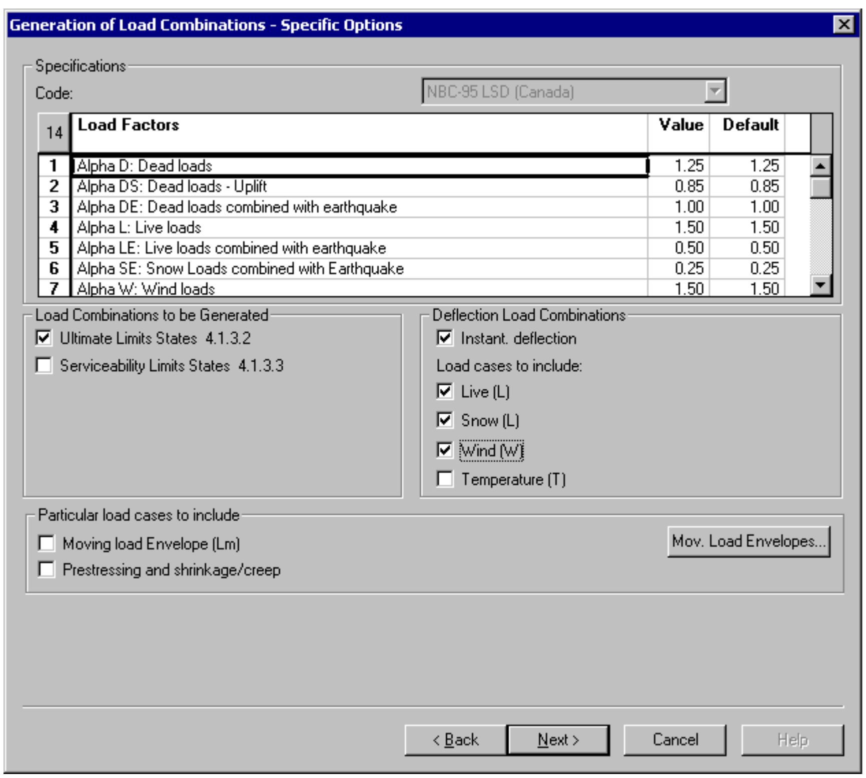
Loads are all defined and applied on the structure. You must at least define one load combination to be allowed to run an analysis. This will be done with the help of the **Load Combination Generation Wizard**.

Generating Load Combinations

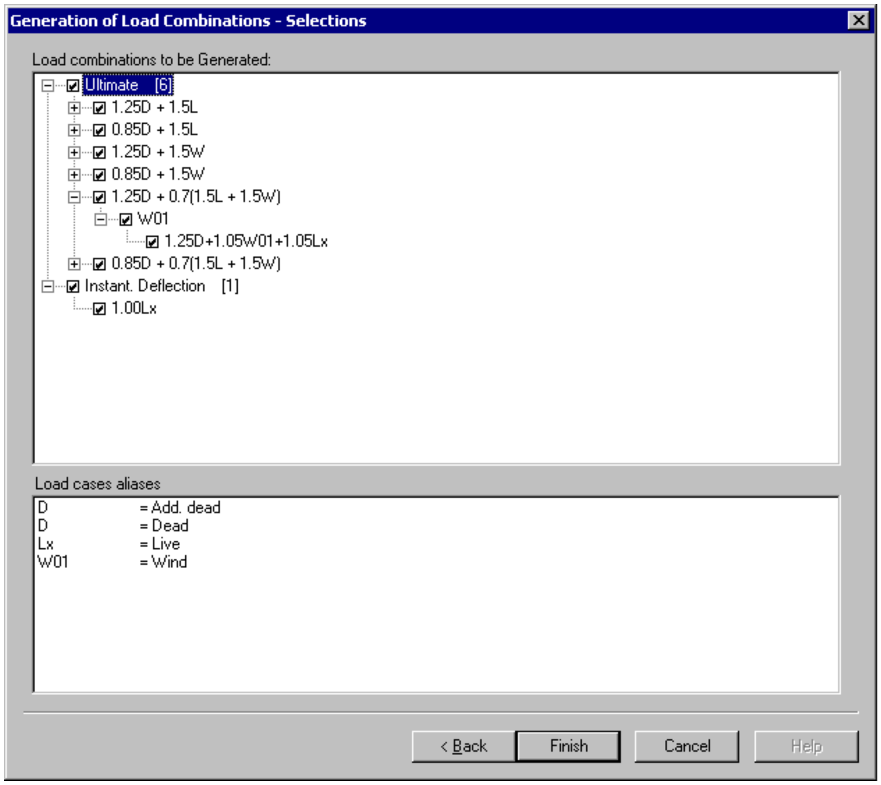
- Go to **Loads** / **Load Combinations/Generation Wizard.** The **General Options** page will be displayed on your screen. Select the NBC code and activate the generation of envelopes.



- Click on the "Next" button to access the **Specific Options** page. Activate ultimate load combinations, and the deflection, using the applied live loads.

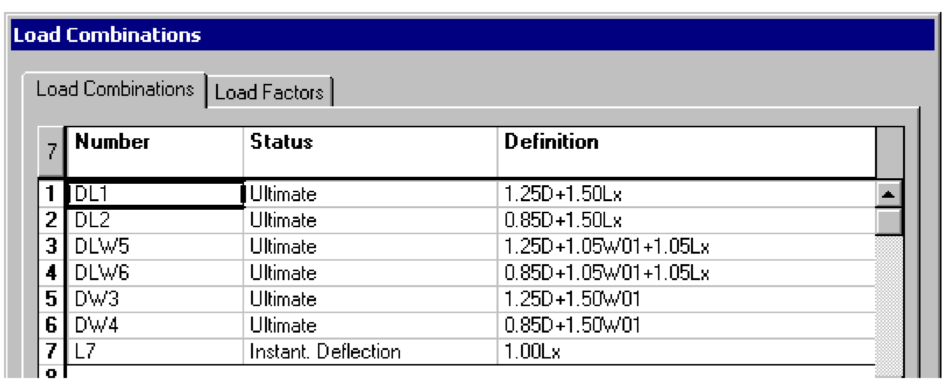


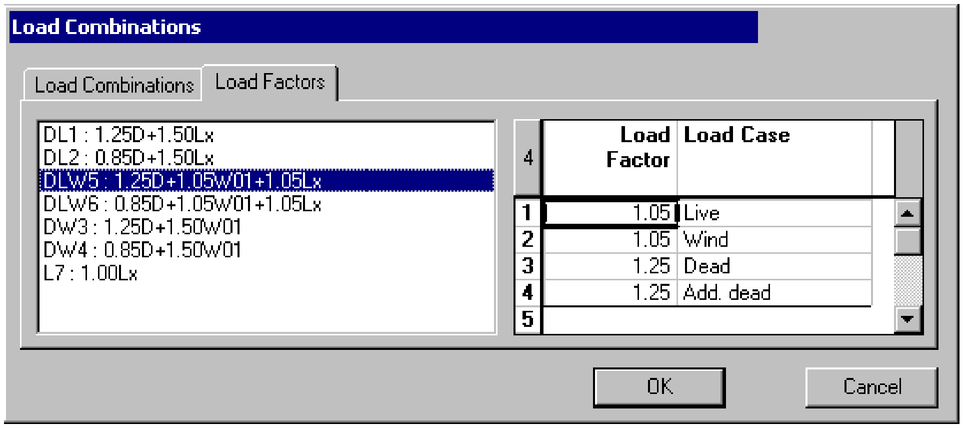
- Click on the "Next" button to access the **Selections** page. You can uncheck load combinations that you do not want to generate.



- Click on the "Finish" button. The **Load Combinations** spreadsheet will appear on your screen.

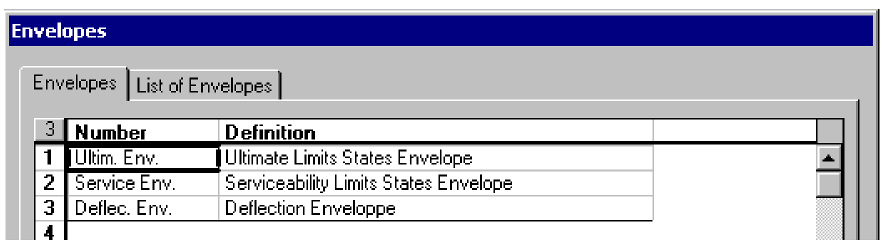
If you wish not to consider some load combinations for a particular analysis, modify the "Status" cell: Double-click and select option "Not required". To modify load factors, select the **Load Factors** tab.

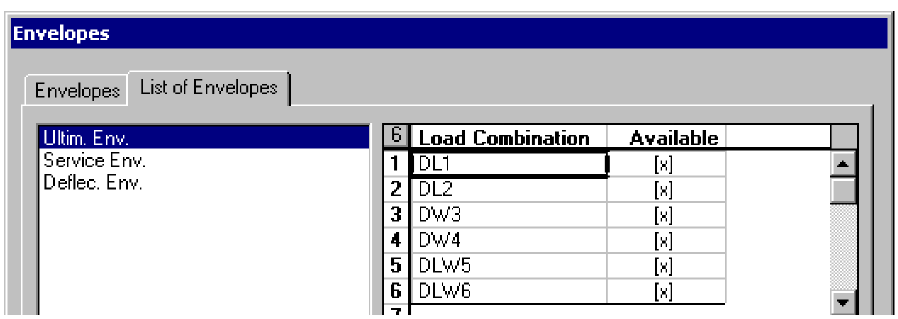


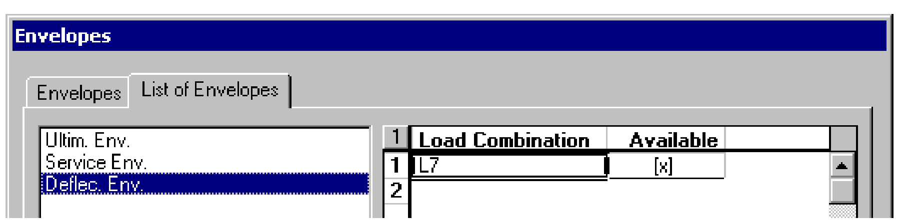


Generated Envelopes

Envelopes have been generated with the Wizard. To have a look at them, go to **Loads** / **Envelopes.**







- Click OK.

You are now ready to run a linear static analysis.

Linear Static Analysis

- Click on the **Analysis** icon of Tools toolbar or select **Static Analysis** in **Analysis** menu.
- Click on the "Analyse" button in the **Analysis** dialog box. When analysis is completed, close the dialog box.

Results

The "Load Combination" mode is automatically activated when analysis is done and the *Title Selection* drop-down list box is open. Select a load combination or press the [Esc] key, and select an envelope.

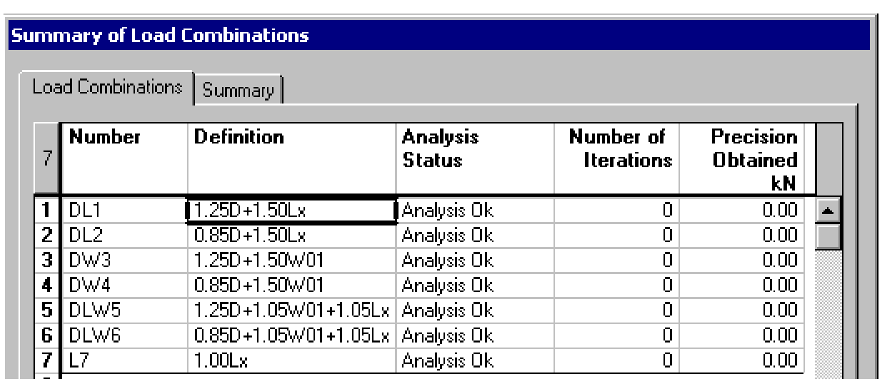
Summary for Analysed Load Combinations

We recommend having a look at this spreadsheet, which includes information about convergence, number of iterations, maximum displacement and rotations.

If load combinations have not converged, load combinations results will be erroneous. Check the maximum rotations in this spreadsheet. Your model can be unstable if too many hinges are present. Display the member end conditions and look carefully.

If the model seems to be adequate but convergence is still not reached, go to the **Analyse** tab of **Project Configuration**, increase the number of iterations or increase the convergence parameter, P axial.

- Go to **Results / Load Combinations / Summary**.



7	Number	Definition	Analysis Status	Number of Iterations	Precision Obtained kN
1	DL1	1.25D+1.50Lx	Analysis Ok	0	0.00
2	DL2	0.85D+1.50Lx	Analysis Ok	0	0.00
3	DW3	1.25D+1.50W01	Analysis Ok	0	0.00
4	DW4	0.85D+1.50W01	Analysis Ok	0	0.00
5	DLW5	1.25D+1.05W01+1.05Lx	Analysis Ok	0	0.00
6	DLW6	0.85D+1.05W01+1.05Lx	Analysis Ok	0	0.00
7	L7	1.00Lx	Analysis Ok	0	0.00

Summary of Load Combinations

Load Combinations | Summary

7	Number	Σ Fx kN	Σ Fy kN	Σ Fz kN	Node Number (Max displ.)	Displ. x mm	Displ. y mm	Displ. z mm	θx °	θy °	θz °
1	DL1	0.00	-4398.39	0.00	bB2	-0.01	-7.13	-0.07	-0.00	-0.00	0.00
2	DL2	0.00	-3765.05	0.00	bB2	-0.01	-6.12	-0.06	-0.00	-0.00	0.00
3	DW3	-426.00	-1979.19	0.00	cB2	-475.70	-1.15	0.42	0.00	-0.09	0.76
4	DW4	-426.00	-1345.85	0.00	cB2	-475.66	-0.71	0.27	0.00	-0.08	0.50
5	DLW5	-298.20	-3672.63	0.00	cB2	-333.17	-2.38	0.85	0.01	-0.07	1.51
6	DLW6	-298.20	-3039.29	0.00	cB2	-333.12	-1.94	0.70	0.00	-0.07	1.24
7	L7	0.00	-1612.80	0.00	bB2	-0.01	-2.65	-0.03	-0.00	-0.00	0.00

Node cB2 is the critical one when wind is considered. The corresponding load combination is DW3.

Display the Structure Deflection

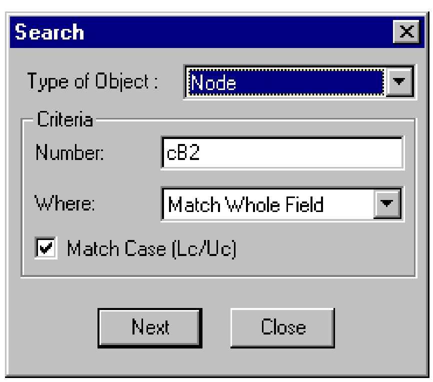
Select load combination DW3 on Activation toolbar:

- Press the **View Options** icon and select the **Results** tab. Check the *Deflection* box in the "Members" section of this tab. Press OK.
- Use the **Diagrams** toolbar functions to adjust the diagram amplitude. To rotate the structure, use the keyboard arrows or the control keys [Home], [Pg Up], [End] or [Pg Dn].

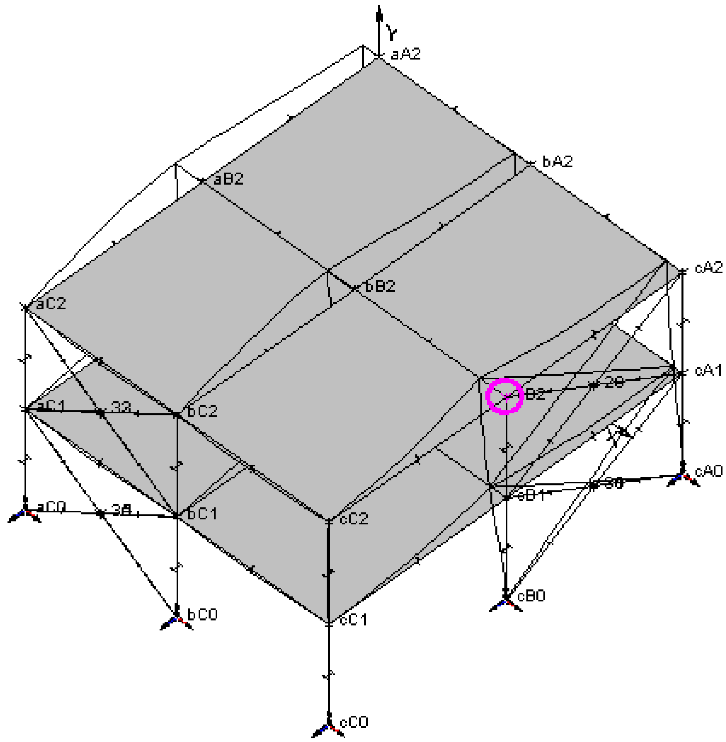
The Find Function

Find the location of node cB2 with the **Find** function of **Edit** menu.

- Click on the **Find** icon of **Edition** toolbar and select a "Node" object in the drop-down list box. Enter the node number and click the "Next" button.



VisualDesign™ will draw a fuchsia circle (default colour) around the element, as shown in the image below.



- To look at numerical values for nodes displacements, go to **Results/Load Combinations** / **Node Displacements**.

Node Displacements Spreadsheet

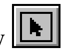
35	Number	Displ. x mm	Displ. y mm	Displ. z mm	θx °	θy °	θz °
19	aA2	-295.34	-0.87	-1.55	0.13	-0.96	0.12
20	bA2	-295.53	-1.53	1.77	0.29	-0.58	0.37
21	cA2	-295.95	-0.54	0.38	0.01	0.31	0.50
22	aB2	-474.43	-1.69	-1.77	-0.01	0.35	-0.02
23	bB2	-474.84	-3.10	1.26	0.01	0.20	0.62
24	cB2	-475.70	-1.15	0.42	0.00	-0.09	0.76
25	aC2	-1.61	-0.98	-2.01	-0.16	5.61	-0.00
26	bC2	-1.70	-1.06	0.77	-0.28	5.69	0.00
27	cC2	-2.12	-0.75	0.42	0.00	0.00	-1.47
28	28	-117.21	-0.21	0.00	-0.00	0.70	1.32
29	29	-117.21	-0.21	0.00	0.01	-1.09	2.27

The Mask Function

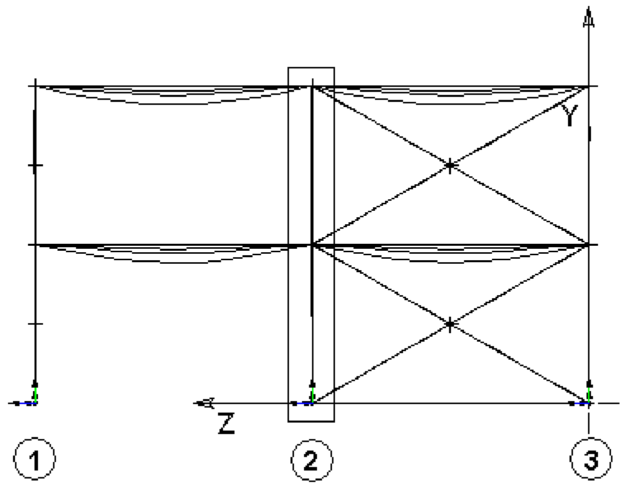
Use this function that masks elements that are not selected to help you visualizing diagrams.

- Select load combination DLW6 on Activation toolbar.

- Move the structure in a ZY view (select the **Camera** function or click icon).

- Activate the **Restricted Window**  selection mode.

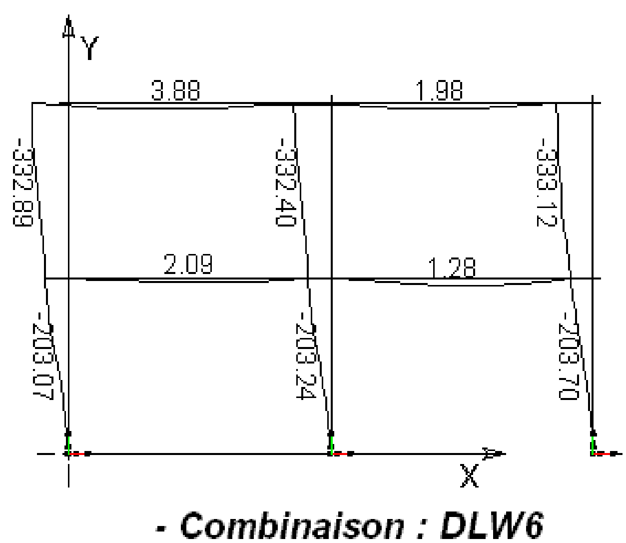
- Select members located on axis 2 (draw a window with the cursor, as shown below).



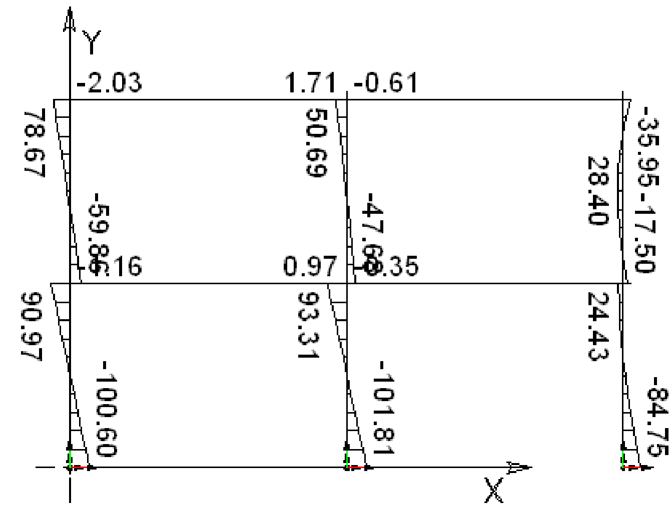
- Activate the **Mask** function by doing one of the following:
  - Click this icon ;
  - Go to **View / Mask;**
  - Use the short-cut key **M**.

- Move the structure in a XY view.

Deflections on weak axis, u, are the following:



Bending moments on weak axis, My, are:



- Combinaison : DLW6

- Unmask the structure by clicking this icon .

Envelope Results

- Activate the Envelope mode on Activation toolbar and select the *Ultimate* Envelope.
- Go to **Results / Envelope** and select one of available spreadsheets.

Support Reactions Spreadsheet & Critical Load combinations

143	Number	Value	Load Combination	Rx kN	Ry kN	Rz kN	Mx kN.m	My kN.m	Mz kN.m
65	bB0	RxMax	DW4	69.69	347.82	-0.59	-1.88	-0.00	-145.44
66	bB0	RxMin	DL1	-0.00	928.47	-0.00	0.04	-0.00	0.01
67	bB0	RyMax	DL1	-0.00	928.47	-0.00	0.04	-0.00	0.01
68	bB0	RyMin	DW4	69.69	347.82	-0.59	-1.88	-0.00	-145.44
69	bB0	RzMax	DL2	-0.00	762.34	-0.00	0.03	-0.00	0.01
70	bB0	RzMin	DW3	69.69	513.94	-0.59	-1.87	-0.00	-145.44
71	bB0	MxMax	DL1	-0.00	928.47	-0.00	0.04	-0.00	0.01
72	bB0	MxMin	DW4	69.69	347.82	-0.59	-1.88	-0.00	-145.44
73	bB0	MyMax	DL2	-0.00	762.34	-0.00	0.03	-0.00	0.01
74	bB0	MyMin	DW3	69.69	513.94	-0.59	-1.87	-0.00	-145.44
75	bB0	MzMax	DL1	-0.00	928.47	-0.00	0.04	-0.00	0.01
76	bB0	MzMin	DW4	69.69	347.82	-0.59	-1.88	-0.00	-145.44
77									
78	bB0	Max		69.69	928.47	-0.00	0.04	-0.00	0.01
79	bB0	Min		-0.00	347.82	-0.59	-1.88	-0.00	-145.44

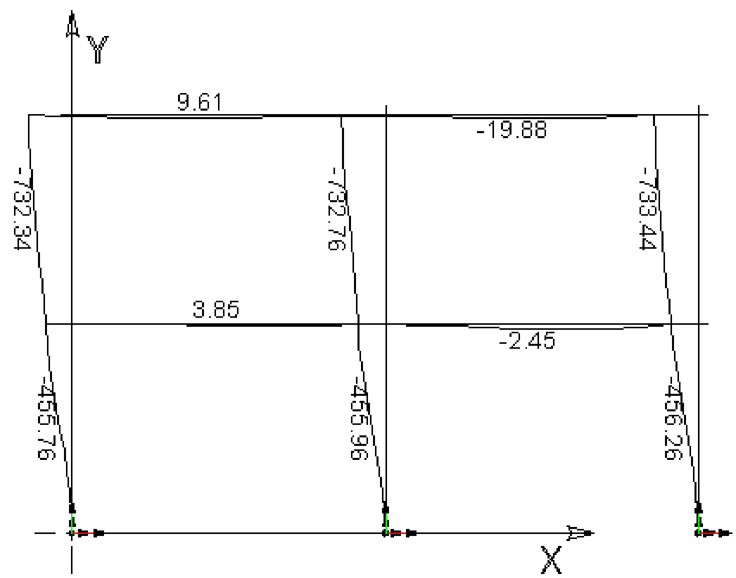
Non-linear Static Analysis

The non-linear static analysis will consider P-Delta effects due to lateral loads such as wind loads.

- Rename the file.
- Select **Project Configuration** of **File** menu. Then, go to the **Analysis** tab and activate a non-linear static analysis. Press OK.
- Launch the non-linear static analysis.

Comparison - Deflection

We are going to compare the displacements for columns (weak axis, u) located on axis 2, for load combination DLW6.



- Load Combination : DLW6

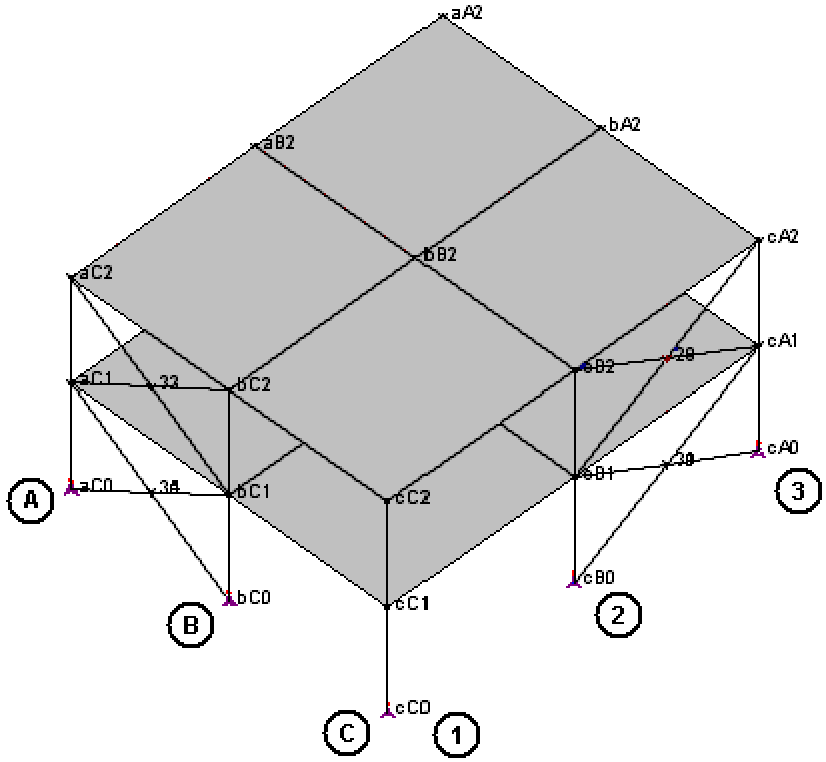
Deflections (u) on weak axis have almost doubled. (See page 49).

The next example will show you how to design this building with the Steel Design module.

EXAMPLE 3

Steel Design of a 2-Storied Building

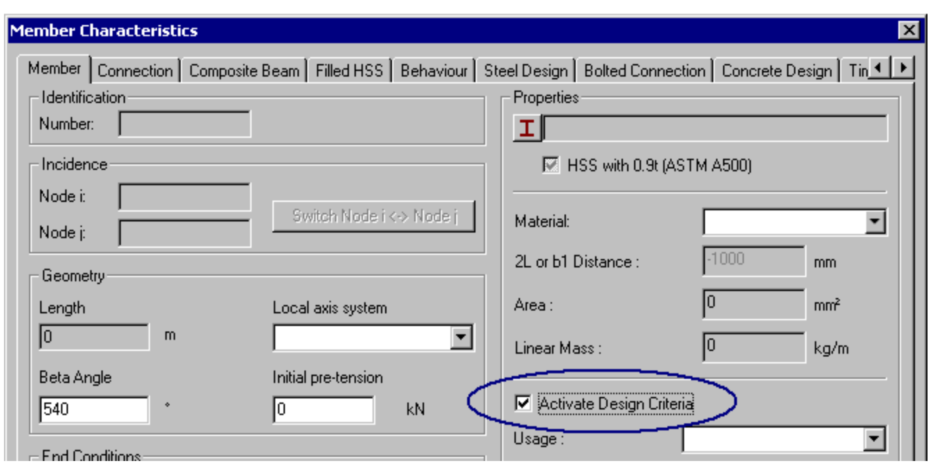
We will use the previous example and show how to design the building according to *CAN/CSA-S16-01* Standard.



Design Criteria

- Activate the Structure mode, the Member element and select all members.

- Click the **Properties** icon  and activate design criteria in the **Member** tab of **Member Characteristics** dialog box. Click OK.



Steel Specifications

We will need 2 steel specifications for this design: One for W shapes (beams and columns) and a second for double steel angles (bracings).

- Go to **Structure** menu and select **Specifications**/**Steel**.

Steel Specifications Spreadsheet

14	Number	Code	Type of analysis	Optimization	Shape	Cat
1	S16-01-Design	CAN/CSA-S16-01	Design	Area	W	
2	S16-01-Vérif.	CAN/CSA-S16-01	Verification	Area	W	
3	S6-88-Design	CAN/CSA-S6-88	Design	Area	W	
4	S6-88-Vérif.	CAN/CSA-S6-88	Verification	Area	W	
5	S6-00-Design	CAN/CSA-S6-00	Design	Area	W	
6	S6-00-Vérif.	CAN/CSA-S6-00	Verification	Area	W	
7	S6-00-Éval.	CAN/CSA-S6-00	Bridge Evaluation	Area	W	
8	S37-01-Design	CAN/CSA-S37-01	Design	Area	L (b=d)	
9	S37-01-Vérif.	CAN/CSA-S37-01	Verification	Area	L (b=d)	
10	LRFD-95-Design	AISC/LRFD-95	Design	Area	W	
11	LRFD-95-Vérif.	AISC/LRFD-95	Verification	Area	W	
12	ASD-89-Design	AISC/ASD-89	Design	Area	W	
13	ASD-89-Vérif.	AISC/ASD-89	Verification	Area	W	
14	S16-01-DesignL	CAN/CSA-S16-01	Design	Area	2EL	

Add a new specification for double steel angles:

- Copy the first line (*S16-01 Design*) and paste it in the bottom of the spreadsheet. In the *Number* column, enter *S16-Design_L*. Double-click in the *Shape* column and select 2EL (2 Equal Legs) steel angle for bracings.
- Close the spreadsheet.

Design Groups

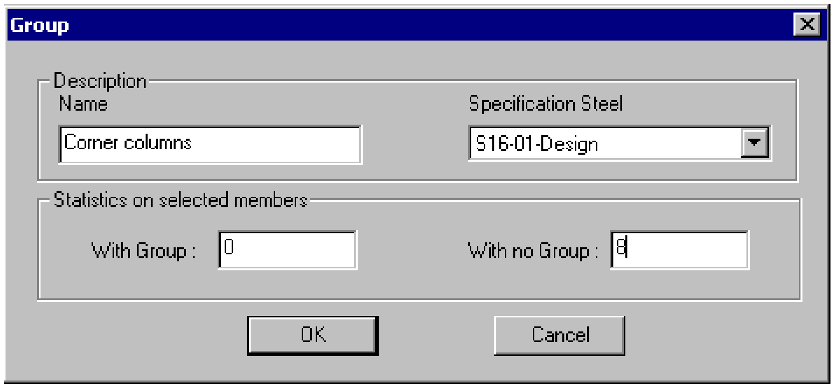
We are going to create the following design groups: Edge beams, Central beams, Corner columns, Inter columns, Central columns, and bracings.

To quickly create design groups, select members that you want to group, and use the **Group** function in menu **Structure / Groups** (short-cut keys are **[Ctrl]+G**).

Example: Corner columns

- Activate the **Restricted Window**  selection mode.
- Activate the Member element and select columns at the corner of the building, keeping the [Ctrl] key down while you select each one.
- Go to **Structure / Groups / Group** or use the short-cut keys **[Ctrl]+G**.

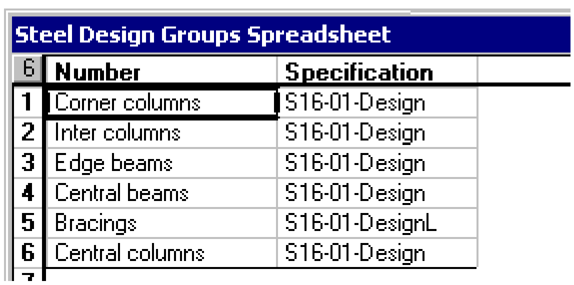
- Type in the name of the group and assign specification *S16-01-Design* to this group, as shown below.



- Press OK.

- Follow the same steps to define other design groups. Don't forget to select specification *S16-01-DesignL* for bracings.

- To edit the names of design groups or select another steel specification, open the **Steel Design Groups** spreadsheet (**Structure** / **Groups** / **Steel Members**).



- To activate the coloured display of a design group, go to **View Options** and activate the *Design Group* option in **Attributes** tab. Select a design group in the drop-down list box.

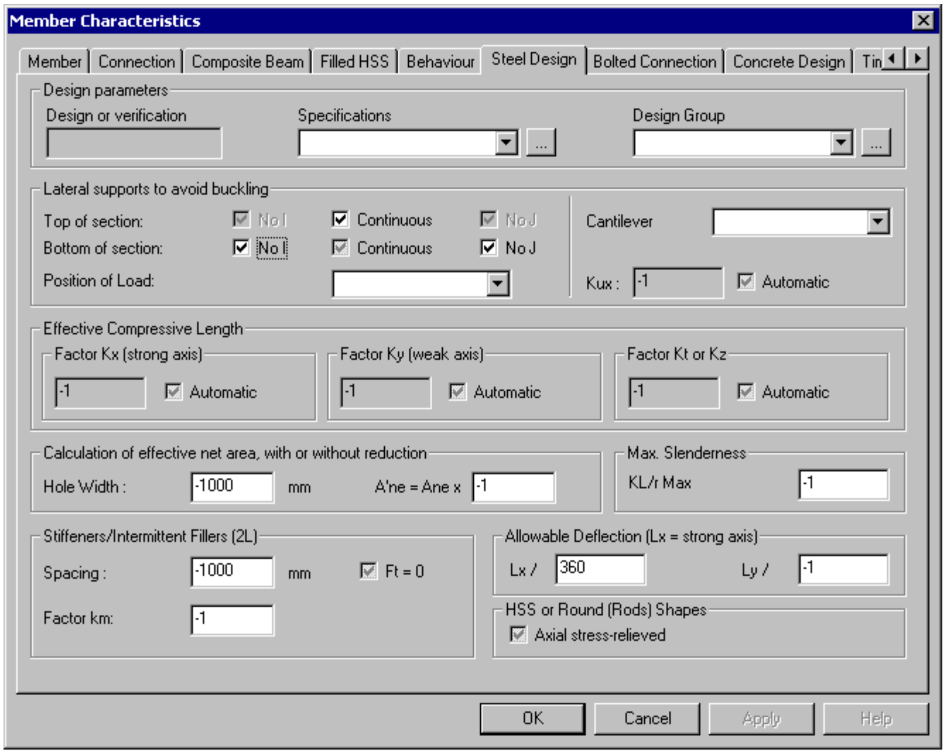
Steel Design Tab

Specifications are already assigned to members through design groups. Other design criteria will be specified in the **Steel Design** tab of **Member Characteristics** dialog box.

Beams

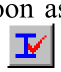
- Go to **Edit** / **Select** / **Beams**. Click on the **Properties** icon  and go to the **Steel Design** tab.

- Specify a continuous lateral support at the top, as shown below, and an allowable deflection of Lx/360 on strong axis.
- Press OK.



Analysis and Design

- Click the **Analysis and Design** icon  of Tools toolbar or go to **Analysis** menu and select **Analysis and Design**.
- In the **Analysis and Design** dialog box, press the "Analyse" button. When design is completed, close the dialog box.

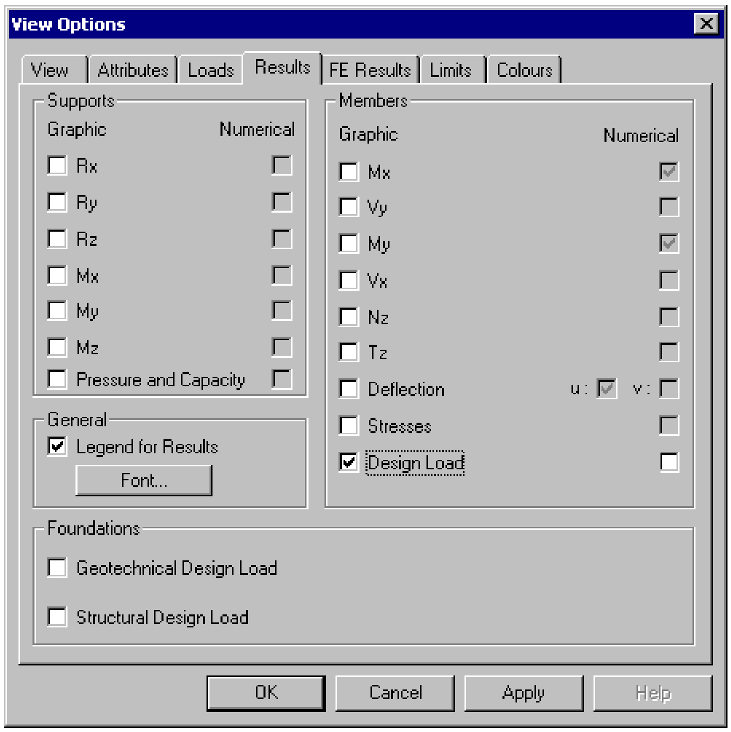
As soon as the design is completed, you will notice that the *Steel Design Results* mode  is automatically activated.

Graphic Results

Display the Members' Coloured Design Loads

- To be able to see the colours for members, disable the display of floor outlines in the **Attributes** tab (**View Options** dialog box).

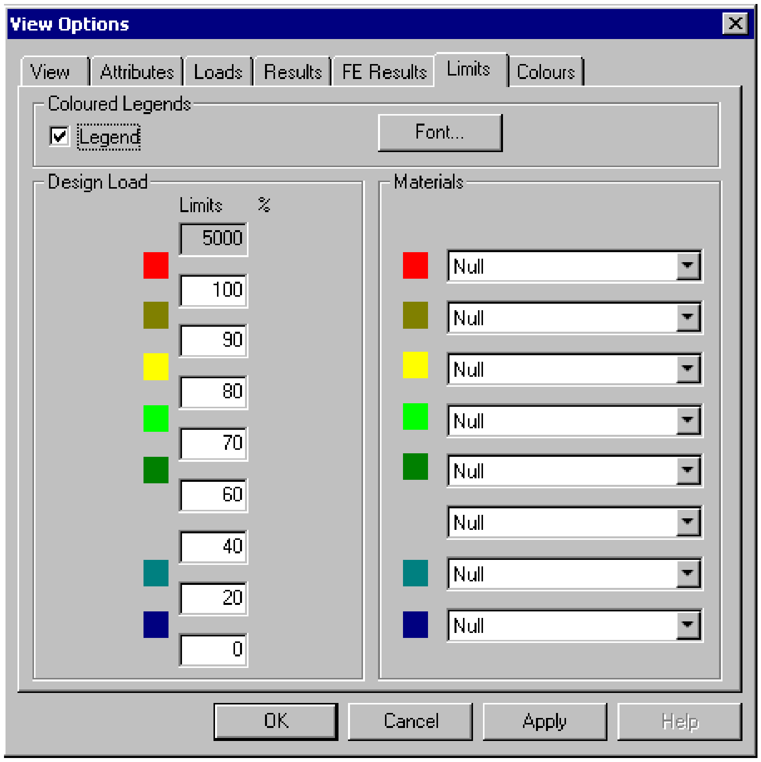
- Go to the **Results** tab and activate the *Design Load* option. Activate the "Numerical" option to see the design load values on screen.



Legend

The colours and values composing the legend for design loads are listed in the **Limits** tab. You can change the colours by clicking on a coloured square. Limit values can be modified also.

- To modify the font and style of displayed legend, click the "Font" button.



Numerical Results and Design Brief

Numerical results are supplied in the **Steel Design Results** spreadsheet. It can be reached in many ways:

- Go to **Results** / **Structure Design** / **Steel**. All members will be part of the spreadsheet.
- Double click on a member to consult results for this member only.
- Select a few members and press the **Properties** icon (or use the short-cut keys **[Ctrl] + H).** The spreadsheet will include only selected members.

To locate a particular member on screen, highlight its line in the spreadsheet and close it. The member will be highlighted on your screen. You can also choose several members in the spreadsheet but lines must be contiguous.

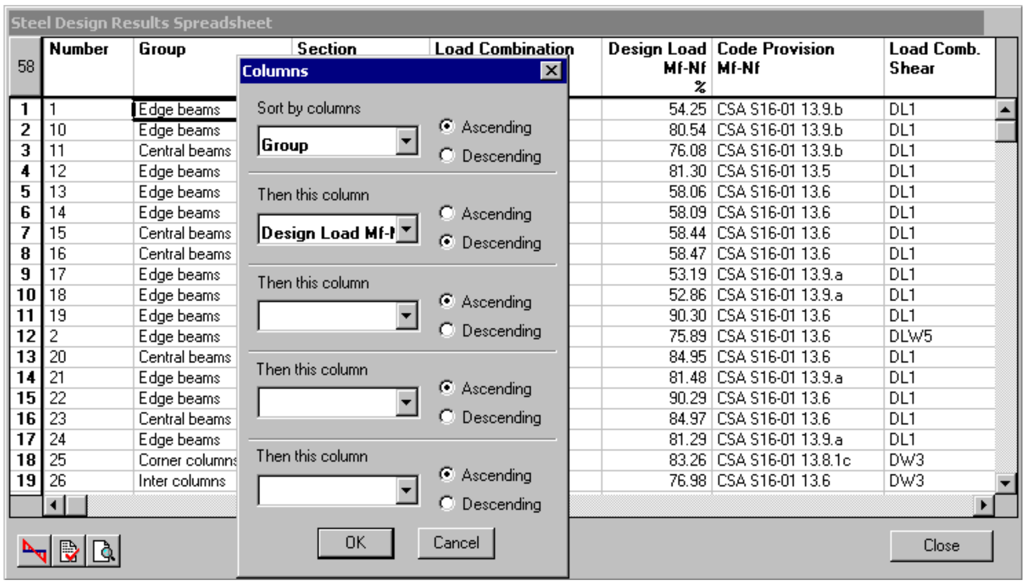
Yellow lines may appear in spreadsheet. They mean that the design is not OK. It may be the resistance, deflection, or KL/r values. Consult the *Results-Resistance* column. If resistance is insufficient, it means that VisualDesign™ have not found a shape that is resistant enough. Modify the steel specification by choosing another shape. For example, if a W shape was specified in the specification, VisualDesign™ has chosen the biggest available shape of this type in the design. So, you must specify a WWF shape and launch another design.

Multiple Sorts

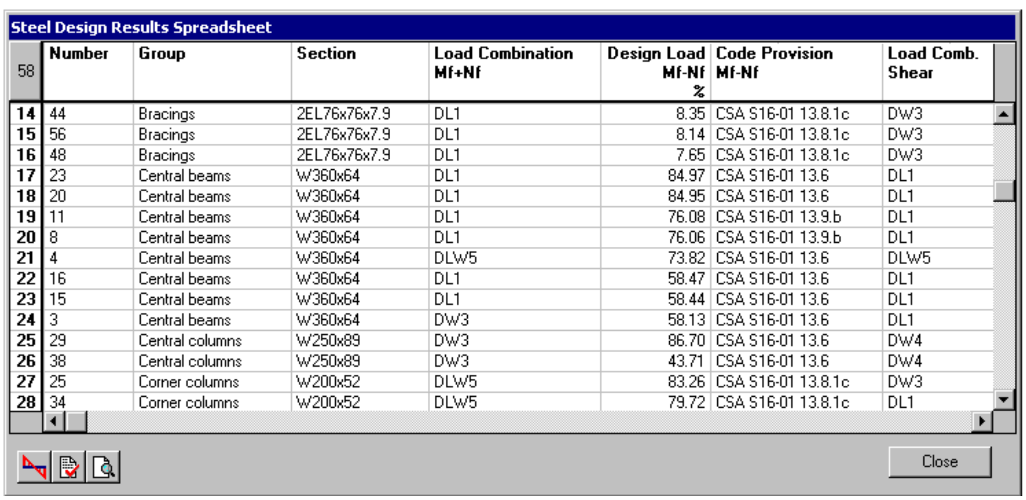
When consulting numerical results, we recommend using the **Sort** function of contextual menu. VisualDesign allows sorting with a maximum of 5 columns.

Procedure:

- Open the spreadsheet.
- Click in any cell, right click, and select the **Sort** function in contextual menu.
- In the **Columns** dialog box, select the order of columns that will be sorted. Specify an ascending or descending sort for each column. Click OK.



58	Number	Group	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear
1	1	Edge beams	54.25	CSA S16-01 13.9.b	DL1
2	10	Edge beams	80.54	CSA S16-01 13.9.b	DL1
3	11	Central beams	76.08	CSA S16-01 13.9.b	DL1
4	12	Edge beams	81.30	CSA S16-01 13.5	DL1
5	13	Edge beams	58.06	CSA S16-01 13.6	DL1
6	14	Edge beams	58.09	CSA S16-01 13.6	DL1
7	15	Central beams	58.44	CSA S16-01 13.6	DL1
8	16	Central beams	58.47	CSA S16-01 13.6	DL1
9	17	Edge beams	53.19	CSA S16-01 13.9.a	DL1
10	18	Edge beams	52.86	CSA S16-01 13.9.a	DL1
11	19	Edge beams	90.30	CSA S16-01 13.6	DL1
12	2	Edge beams	75.89	CSA S16-01 13.6	DLW5
13	20	Central beams	84.95	CSA S16-01 13.6	DL1
14	21	Edge beams	81.48	CSA S16-01 13.9.a	DL1
15	22	Edge beams	90.29	CSA S16-01 13.6	DL1
16	23	Central beams	84.97	CSA S16-01 13.6	DL1
17	24	Edge beams	81.29	CSA S16-01 13.9.a	DL1
18	25	Corner columns	83.26	CSA S16-01 13.8.1c	DW3
19	26	Inter columns	76.98	CSA S16-01 13.6	DW3

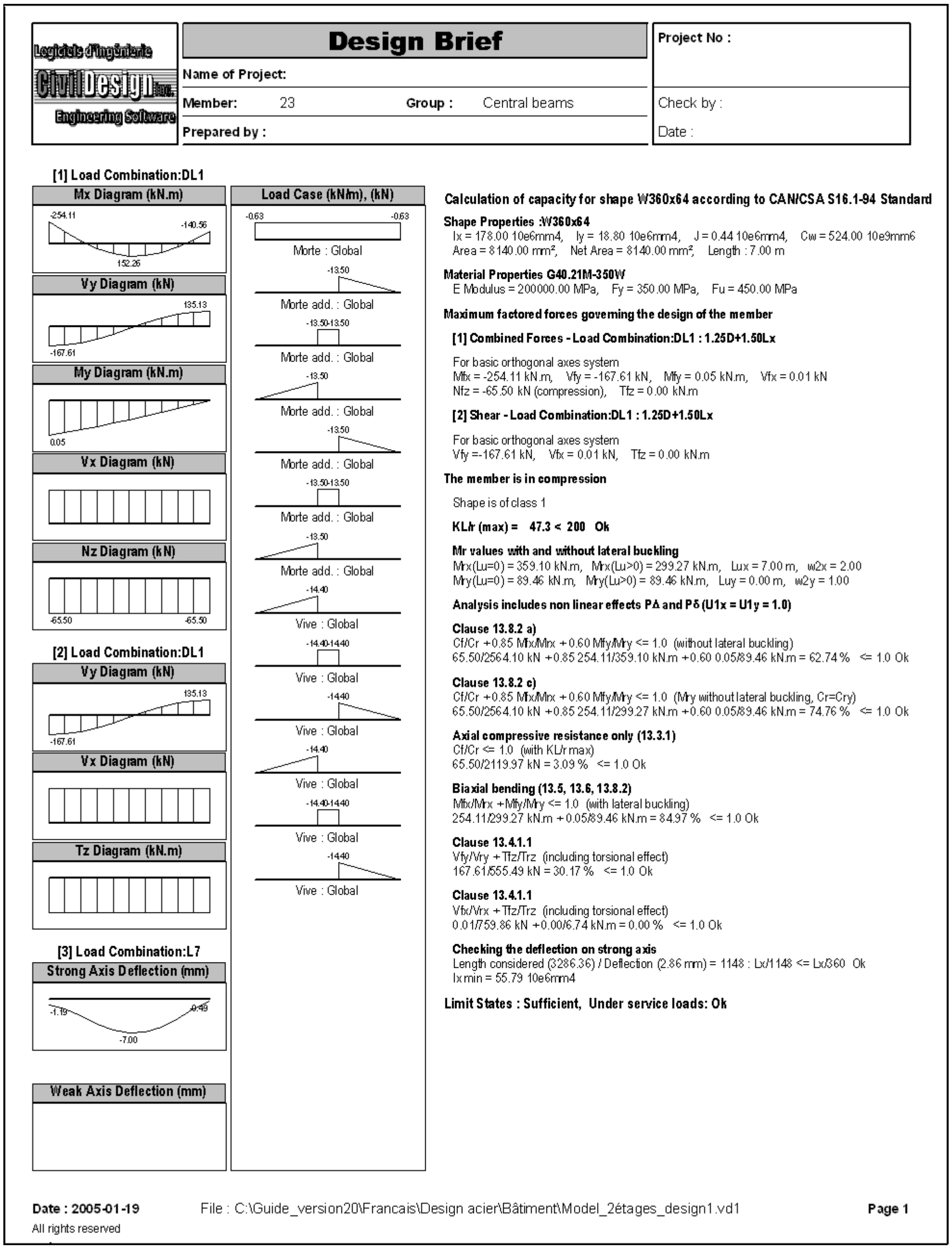


58	Number	Group	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear
14	44	Bracings	2EL76x76x7.9	DL1	8.35	CSA S16-01 13.8.1c	DW3
15	56	Bracings	2EL76x76x7.9	DL1	8.14	CSA S16-01 13.8.1c	DW3
16	48	Bracings	2EL76x76x7.9	DL1	7.65	CSA S16-01 13.8.1c	DW3
17	23	Central beams	W360x64	DL1	84.97	CSA S16-01 13.6	DL1
18	20	Central beams	W360x64	DL1	84.95	CSA S16-01 13.6	DL1
19	11	Central beams	W360x64	DL1	76.08	CSA S16-01 13.9.b	DL1
20	8	Central beams	W360x64	DL1	76.06	CSA S16-01 13.9.b	DL1
21	4	Central beams	W360x64	DLW5	73.82	CSA S16-01 13.6	DLW5
22	16	Central beams	W360x64	DL1	58.47	CSA S16-01 13.6	DL1
23	15	Central beams	W360x64	DL1	58.44	CSA S16-01 13.6	DL1
24	3	Central beams	W360x64	DW3	58.13	CSA S16-01 13.6	DL1
25	29	Central columns	W250x89	DW3	86.70	CSA S16-01 13.6	DW4
26	38	Central columns	W250x89	DW3	43.71	CSA S16-01 13.6	DW4
27	25	Corner columns	W200x52	DLW5	83.26	CSA S16-01 13.8.1c	DW3
28	34	Corner columns	W200x52	DLW5	79.72	CSA S16-01 13.8.1c	DL1

The design is OK.

Design Brief

Highlight a line in the spreadsheet to select a member and use the **Print Preview** icon to consult the design brief. To directly print it, select a line and click the icon. Look at member 23's **Design Brief**.



Logiciels d'ingénierie CivilDesign inc. Engineering Software

Design Brief

Name of Project:			**Project No :**
Member: 23	**Group :** Central beams		Check by :
Prepared by :			Date :

Calculation of capacity for shape W360x64 according to CAN/CSA S16.1-94 Standard

Shape Properties :W360x64
Ix = 178.00 10e6mm4, Iy = 18.80 10e6mm4, J = 0.44 10e6mm4, Cw = 524.00 10e9mm6
Area = 8140.00 mm², Net Area = 8140.00 mm², Length : 7.00 m

Material Properties G40.21M-350W
E Modulus = 200000.00 MPa, Fy = 350.00 MPa, Fu = 450.00 MPa

Maximum factored forces governing the design of the member

[1] Combined Forces - Load Combination:DL1 : 1.25D+1.50Lx

For basic orthogonal axes system
Mfx = -254.11 kN.m, Vfy = -167.61 kN, Mfy = 0.05 kN.m, Vfx = 0.01 kN
Nfz = -65.50 kN (compression), Tfz = 0.00 kN.m

[2] Shear - Load Combination:DL1 : 1.25D+1.50Lx

For basic orthogonal axes system
Vfy =-167.61 kN, Vfx = 0.01 kN, Tfz = 0.00 kN.m

The member is in compression

Shape is of class 1

KL/r (max) = 47.3 < 200 Ok

Mr values with and without lateral buckling
Mrx(Lu=0) = 359.10 kN.m, Mrx(Lu>0) = 299.27 kN.m, Lux = 7.00 m, w2x = 2.00
Mry(Lu=0) = 89.46 kN.m, Mry(Lu>0) = 89.46 kN.m, Luy = 0.00 m, w2y = 1.00

Analysis includes non linear effects PΔ and Pδ (U1x = U1y = 1.0)

Clause 13.8.2 a)
Cf/Cr +0.85 Mfx/Mrx +0.60 Mfy/Mry <= 1.0 (without lateral buckling)
65.50/2564.10 kN +0.85 254.11/359.10 kN.m +0.60 0.05/89.46 kN.m = 62.74 % <= 1.0 Ok

Clause 13.8.2 c)
Cf/Cr +0.85 Mfx/Mrx +0.60 Mfy/Mry <= 1.0 (Mry without lateral buckling, Cr=Cry)
65.50/2564.10 kN +0.85 254.11/299.27 kN.m +0.60 0.05/89.46 kN.m = 74.76 % <= 1.0 Ok

Axial compressive resistance only (13.3.1)
Cf/Cr <= 1.0 (with KL/r max)
65.50/2119.97 kN = 3.09 % <= 1.0 Ok

Biaxial bending (13.5, 13.6, 13.8.2)
Mfx/Mrx +Mfy/Mry <= 1.0 (with lateral buckling)
254.11/299.27 kN.m + 0.05/89.46 kN.m = 84.97 % <= 1.0 Ok

Clause 13.4.1.1
Vfy/Vry + Tfz/Trz (including torsional effect)
167.61/555.49 kN = 30.17 % <= 1.0 Ok

Clause 13.4.1.1
Vfx/Vrx + Tfz/Trz (including torsional effect)
0.01/759.86 kN +0.00/6.74 kN.m = 0.00 % <= 1.0 Ok

Checking the deflection on strong axis
Length considered (3286.36) / Deflection (2.86 mm) = 1148 : Lx/1148 <= Lx/360 Ok
Ix min = 55.79 10e6mm4

Limit States : Sufficient, Under service loads: Ok

Date : 2005-01-19 File : C:\Guide_version20\Francais\Design acier\Bâtiment\Model_2étages_design1.vd1 **Page 1**

Member Internal Forces and Deflections

To look at a member forces and deflections, highlight the line in the spreadsheet and click on this icon . Do not forget that results correspond to the most critical load combination, in our case: DL1.

Internal Forces and Deflections Spreadsheet (DL1)

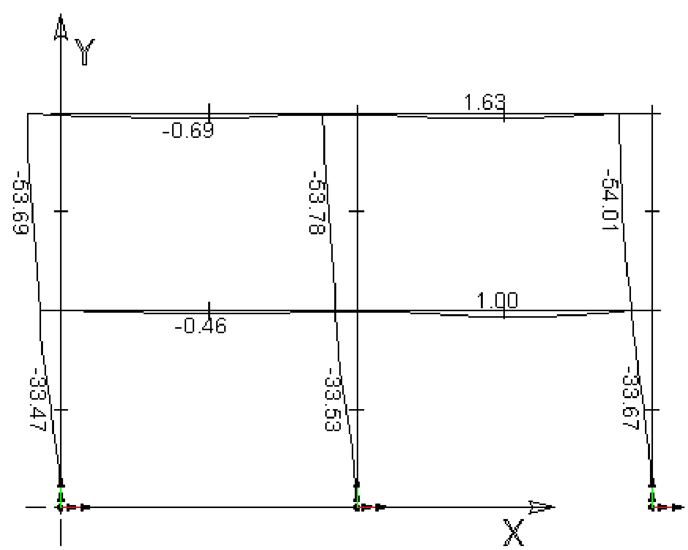
11	Number	Shape	Position m	Bending Mx kN.m	Shear Vy kN	Bending My kN.m	Axial Nz kN	Strong axis v mm	Axial w mm
1	23	W360x64	0.00	-254.11	-167.61	0.05	-65.50	-3.26	-0.04
2	23	W360x64	0.70	-138.25	-160.99	0.05	-65.50	-5.39	-0.07
3	23	W360x64	1.40	-31.15	-142.24	0.04	-65.50	-9.46	-0.10
4	23	W360x64	2.10	58.61	-111.34	0.04	-65.50	-13.90	-0.13
5	23	W360x64	2.80	122.42	-68.31	0.03	-65.50	-17.40	-0.15
6	23	W360x64	3.50	152.26	-16.24	0.03	-65.50	-19.02	-0.18
7	23	W360x64	4.20	145.22	35.83	0.02	-65.50	-18.32	-0.21
8	23	W360x64	4.90	104.18	78.87	0.02	-65.50	-15.41	-0.24
9	23	W360x64	5.60	37.15	109.76	0.01	-65.50	-10.92	-0.27
10	23	W360x64	6.30	-47.28	128.52	0.01	-65.50	-5.82	-0.30
11	23	W360x64	7.00	-140.56	135.13	-0.00	-65.50	-1.36	-0.32

Load Combination Results

Load Combination DWL6

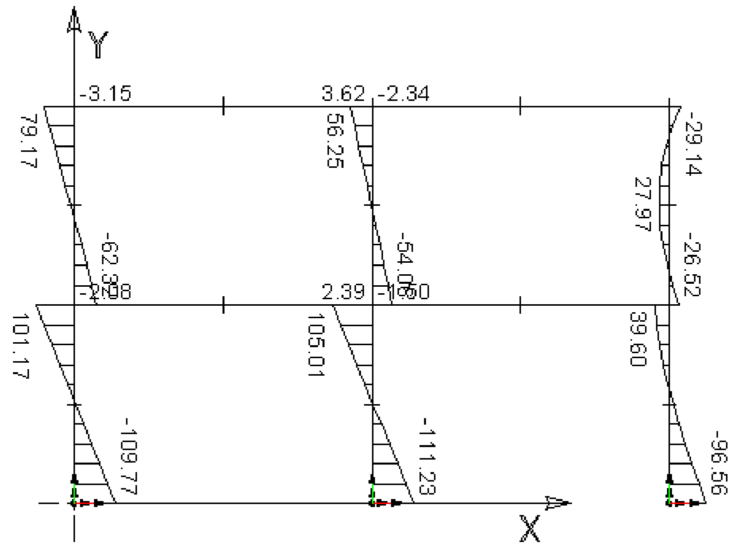
Display deflections on weak axis, u, and bending moments on weak axis, My, for elements located on axis 2 and compare with the results that we obtained from non-linear static analysis (page 51).

Deflections on weak axis for Axis 2:



- Load Combination : DLW6

Bending moments on weak axis, for axis 2:



- **Load Combination : DLW6**

E X A M P L E 4

Steel Design - Composite Beams

Basic Principles

Project Configuration

If composite beams are not shored during construction, you must define construction stages in the **Composite Beam** tab of **Project Configuration** dialog box if you own the Steel Design module. If not, composite beams will be considered as shored.

Creating a Slab

Create a slab in the **Slabs** spreadsheet (**Structure** menu).

Defining a Member As Composite Beam

Open the **Member Characteristics** dialog box. Activate design criteria and choose option *Composite Beam* in the "Composition" field. Then, select the **Composite Beam** tab and specify the composite slab parameters.

If you want to get stresses on other points on the beam, press the button "Extra Calculation – Stresses" and indicate the location of these points. The results at these points will be available in **Results / Load Combinations / Stresses – Composite Beams** after the design is completed.

Load Cases

Define one load case per construction stage. Add other load cases if necessary.

Load Combinations

Define a load combination per construction stage in the **Load Combination Definition** spreadsheet and indicate the stage number in the appropriate cell. These load combinations must have an appropriate status such as *Service*.

Use the **Load Combination Generation Wizard** to create all required load combinations according to selected code.

If you want to obtain graphic results for load combinations other than construction stages, select a *Service* or ULS no 1 (CAN/CSA-S6-00) for these.

Graphic Results for Composite Beams (Beams Not Shored)

When the steel design is completed, go to **Results** / **Load Combinations / Stresses – Composite Beams**.

Stresses in Composite Beams (Shored)

The spreadsheet **Variation of Stresses in Member**, which is available in menu **Results / Envelope**, is only valid for composite beams with no construction stages. Stresses are calculated for the steel shape only, none for the slab.

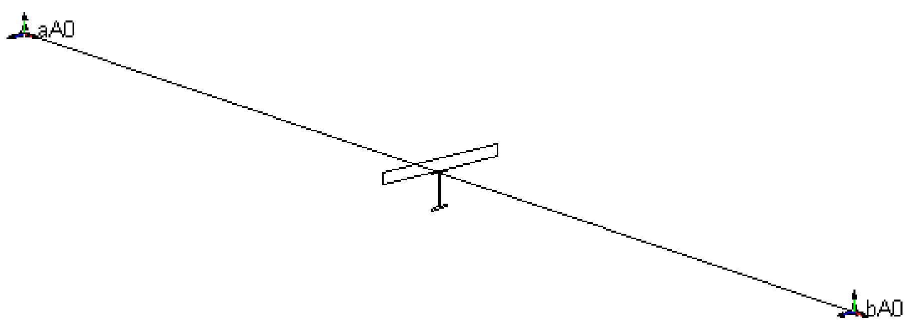
Project Configuration

Project

A composite beam of 20 meters long will be designed according to code *CAN/CSA-S6-00*. This beam will not be shored during construction so we will define proper construction stages.

Load cases are: The concrete slab, bitumen, and new jerseys. Moving loads will also be applied.

This simple model will look like this:

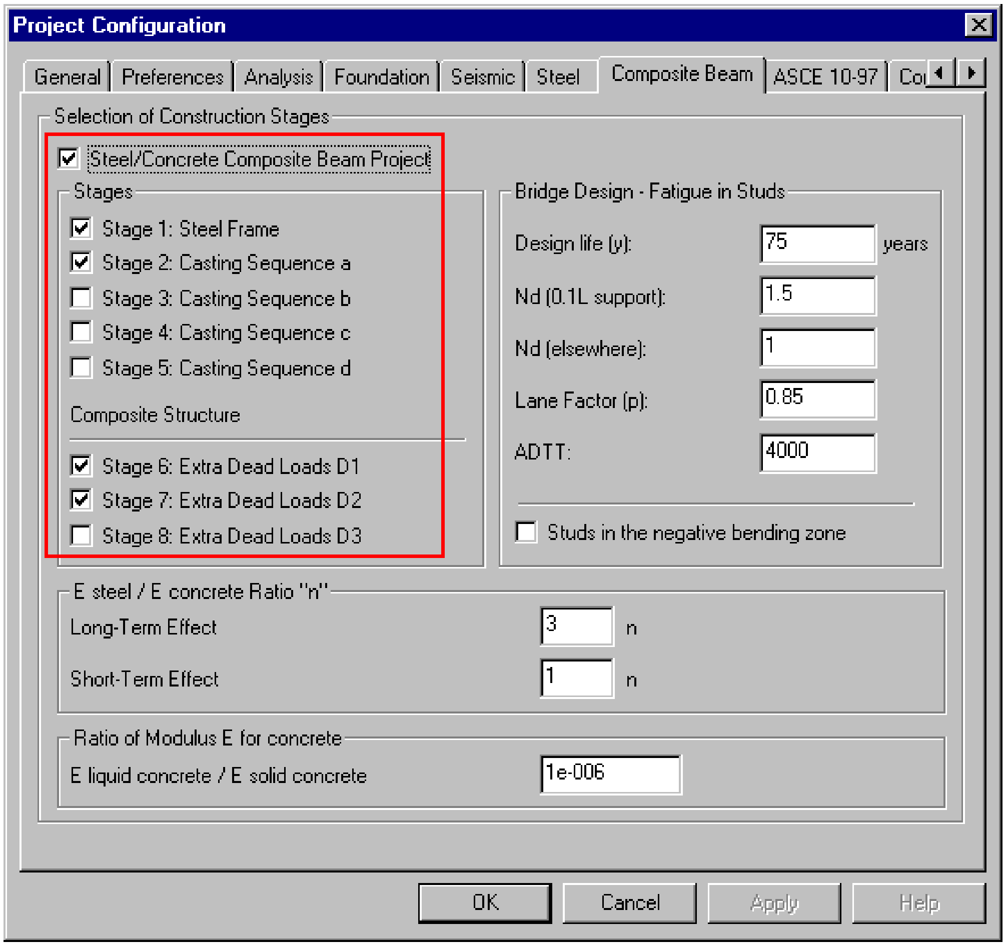


Project Configuration

- Go to **File** menu and select **Project Configuration**. Define construction stages in the **Composite Beam** tab, as shown in the figure below.

  Stage 1 represents the dead load of the steel beam. Stage 2 represents the casting of the slab; Stage 6, the dead load of wearing surface; and stage7, the dead load of new jerseys.

  This composite beam is part of a bridge and we will keep the default values entered in section ***Bridge Design – Fatigue in Studs***.



- Press OK to save data and exit the dialog box.

Moving Load Cases

- Go to **Loads / Moving Load Cases / Definition** and select the CL-625-25 moving load. The second line corresponds to the fatigue case: The overload is not considered.

Moving Load Cases

Cases | Moving Load Cases Components

2	Number	Truck	Envelope	Moving Load Axis	Traffic on axis	DLA (Truck)	DLA (Truck/Lane)	Add Overload
1	CL-625-25	[2D]-CL1-625-25	Truck : Lm01	Axis 1	>> & <<	0.25	0.00	[x]
2	CL-625-25_F	[2D]-CL1-625-25	Truck : Lm02	Axis 1	>> & <<	0.25	0.00	[ ]

- In the **Components** tab, the factor that will be applied to punctual loads and overload is equal to 1.0.

Moving Load Cases

Cases | Moving Load Cases Components

CL-625-25
CL-625-25_F

1	Factor	Offset m
1	1.00	0.00
2		

- Open the **2D Axles Factors** spreadsheet (**Loads / 2D Dynamic Allowance factors).** All factors will be equal to 1.0 for spans and supports.

2D Axle Factors

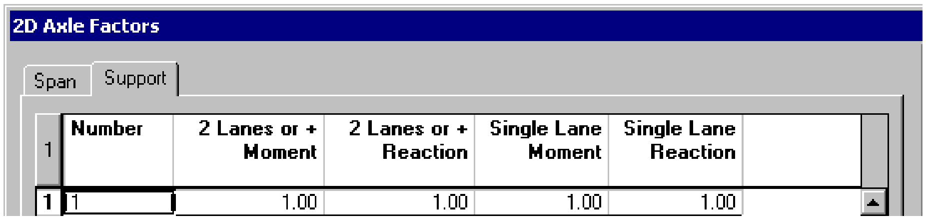
Span | Support

1	Number	2 Lanes or + Mx +	2 Lanes or + Mx -	2 Lanes or + Vy	2 Lanes or + My, Vx, Nz, Tz	2 lanes or + Displac.
1	1	1.00	1.00	1.00	1.00	1.00

2D Axle Factors

Span | Support

1	Single lane Mx+	Single lane Mx-	Single lane Vy	Single Lane My, Vx, Nz, Tz	Single Lane Displac.
1	1.00	1.00	1.00	1.00	1.00



Notes:

The 2D axle factors specified for spans must be assigned to the beam through the **Member Characteristics** dialog box, as you will see further on.

The 2D axle factors specified for supports must be assigned to supports through the Supports tab of **Node Characteristics** dialog box

Modelling

Nodes

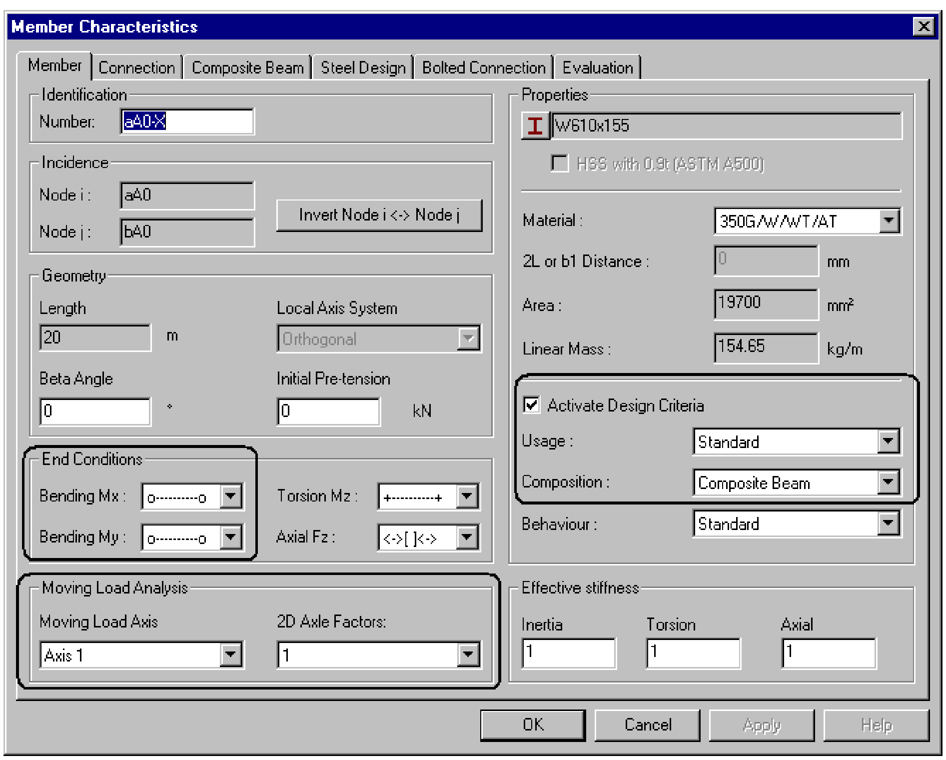
- Activate the Structure mode and the Node icon. Open the **Nodes** spreadsheet through the **Structure** menu. Add two lines, enter nodes coordinates and choose option *Support* as type of nodes.

Nodes Spreadsheet

2	Number	Type	Coord. X m	Coord. Y m	Coord. Z m	ID Master No.
1	aA0	Support	0.00	0.00	0.00	0
2	bA0	Support	20.00	0.00	0.00	0
3						

Member

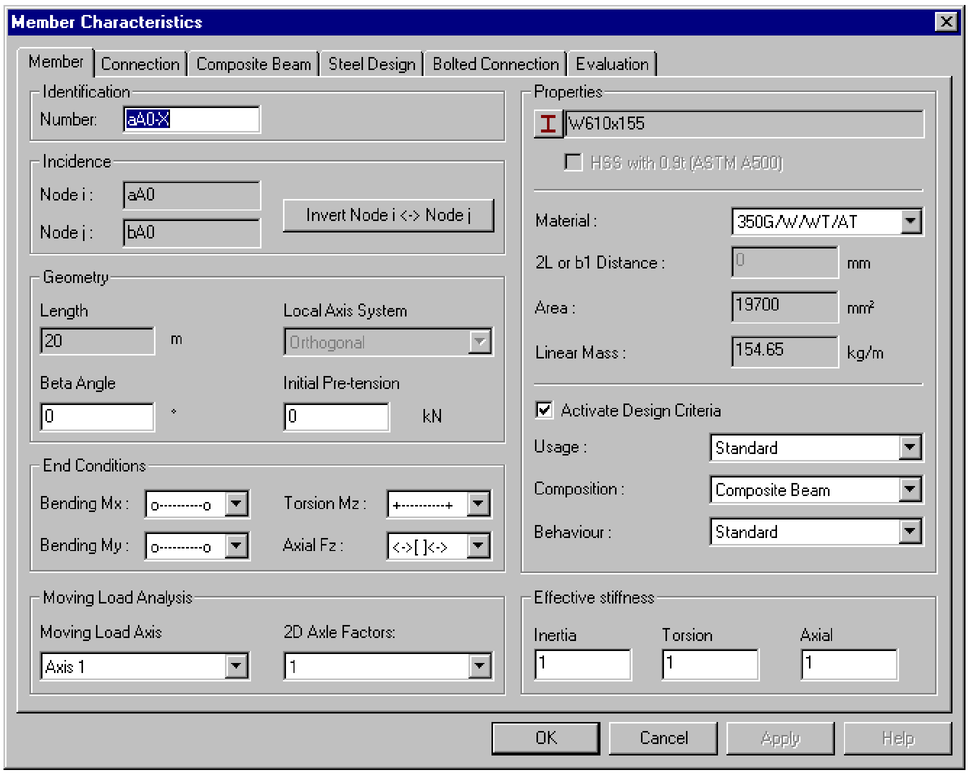
- To create the member, activate the Member icon and the **Add** mode. Click on the first node (i) and the end node (j). The **Member Characteristics** dialog box will appear on screen.



- Select a preliminary steel shape and a steel grade. The member end conditions are hinged. Select option *Composite Beam* in the "Composition" field. Activate design criteria.

- In the "Moving Load Analysis" section, select Axis 1 and the 2D axle factors for spans. Press OK.

  N. B. To learn more about moving load analysis and 2D axle factors, refer to examples included further on in this document.

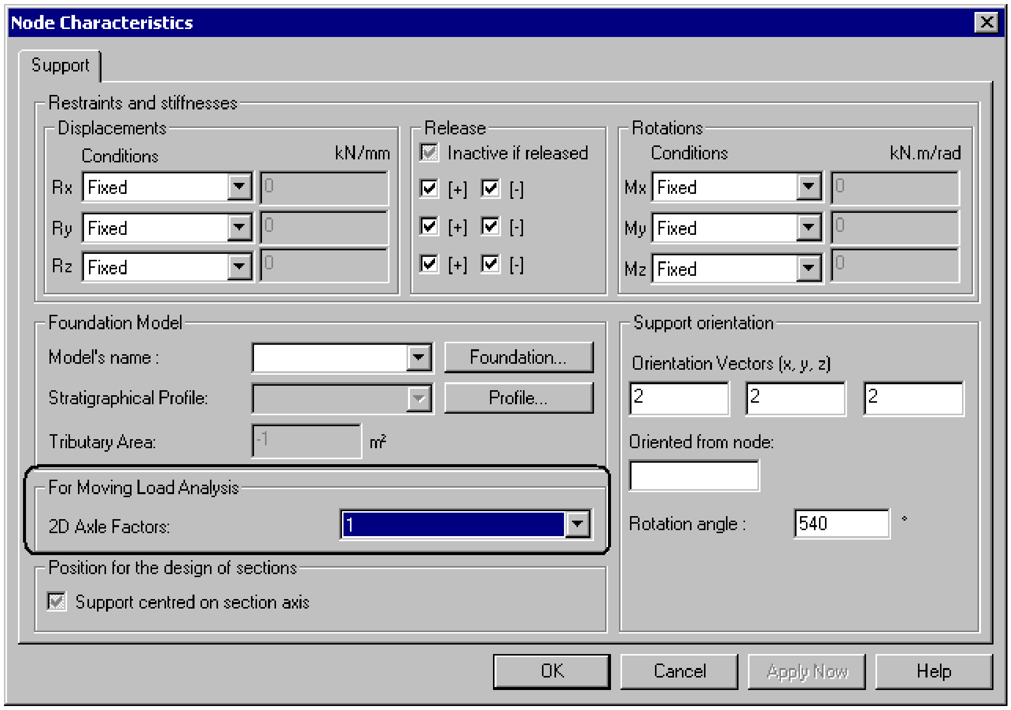


- Press OK.

- Activate the **Restricted Window**  selection mode to exit the **Add** mode.

Supports

- Activate the Support icon. Select the two supports and press the **Properties** icon to open the **Node Characteristics** dialog box. Define the support restraints and select 2D axle factors for supports. Press OK.



Slab

- Go to **Structure / Slabs**. Insert a line and enter the slab dimension and other parameters. This slab will be selected further on when defining the composite beam.

Slabs Spreadsheet

	Number	Steel deck	Direction	tc mm	hd mm	to mm	Rebar, top
1	Slab	Null	Perpendicular	200.00	0.00	200.00	15M
2							

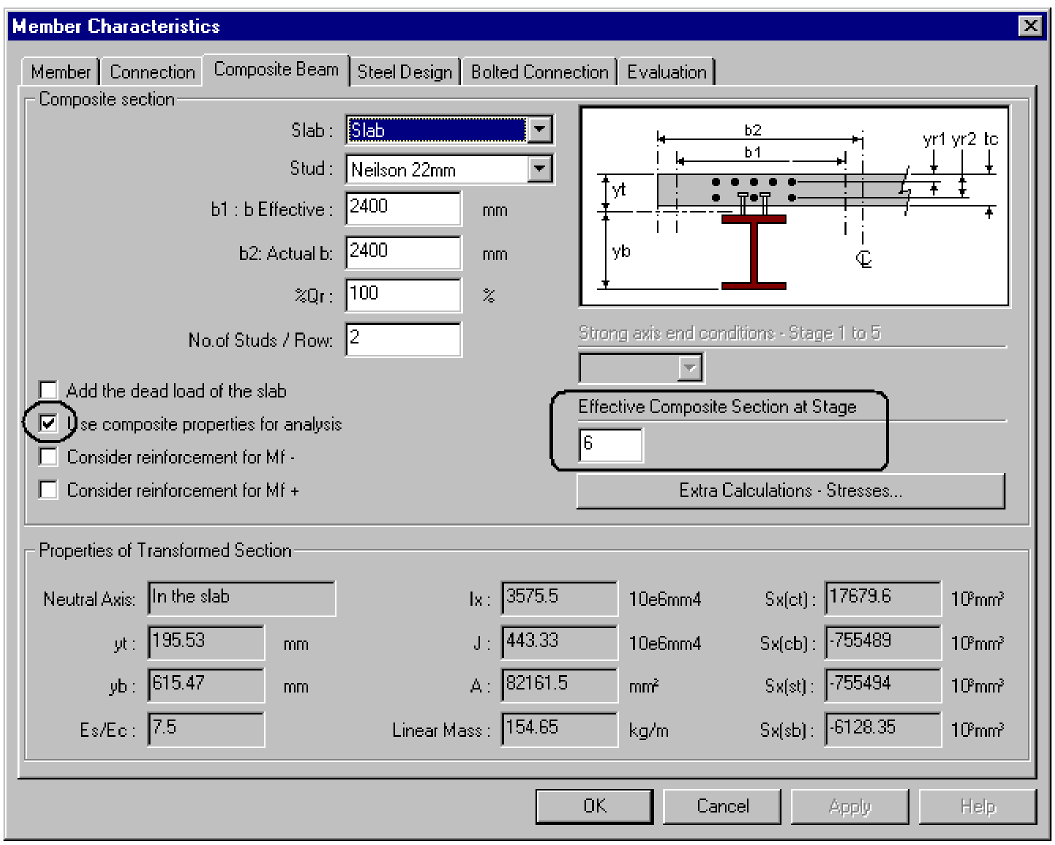
Slabs Spreadsheet

	s, top mm	d, top mm	Rebar, bot.	s, bot. mm	d, bottom mm	Material Rebar	Material Concrete
1	200.00	167.00	15M	300.00	167.00	G30.18-400R	Con030

Press the control key **F1** when a spreadsheet is open. It will open the On-Line Help right at this topic. It is useful to look at the definition of parameters included in spreadsheets.

Defining the Composite Beam

- Activate the member icon and double click on the beam. Go to the **Composite Beam** tab.
- Select the slab that you created and choose a stud among the drop-down list box. Enter effective and actual widths of slab. By default, the shear connectors located between the point of maximum and zero bending moments are 100% effective.
- Activate the option that considers the transformed-section properties for analysis: *Use composite properties for analysis*.
- Enter the stage number where composite effects will be effective.
- Close the dialog box.



Options:

Add the dead load of the slab: Do not activate this option when you have construction stages.

Consider Reinforcement for Mfx - or Mfx +: Activate the option to consider the slab reinforcement in the zone of negative or positive bending moment. The position of neutral axis will be calculated as a result.

Extra Calculations – Stresses: This function opens a tool that allows defining other points on the steel beam, where stress calculations are required.

Properties of the Transformed-section

Open the **Members** spreadsheet and consult the transformed-section properties that will be used to calculate short-term and long-term deformations, considering ratios "n" (where n = Es/Ec) that were specified in the **Composite Beam** tab of **Project Configuration**. Default values are n=1 for short-term and n=3 for long-term. These default values are also used for those not owning the *Steel Design* module.

Short-term

Members Spreadsheet

Member | Connection | Composite Beam | Composite Beam for Short-term | Composite Beam for Long-term | Filled HSS | Behaviour | Steel Design | Bolted Con

1	Composition	Neutral Axis	yt mm	yb mm	Es/Ec	Ix 10e6mm4	J 10e6mm4	Area mm²	Sx(ct) 10³mm³	Sx(cb) 10³mm³	Sx(st) 10³mm³	Sx(sb) 10³mm³
1	Composite Beam	Below slab	261.92	591.08	7.5	6771.38	457.28	102990.40	25035.57	103782.71	103782.73	-12039.97

Long-term

Members Spreadsheet

Member | Connection | Composite Beam | Composite Beam for Short-term | Composite Beam for Long-term | Filled HSS | Behaviour | Steel Design | Bolted Co

1	Composition	Neutral Axis	yt mm	yb mm	Es/Ec	Ix 10e6mm4	J 10e6mm4	Area mm²	Sx(ct) 10³mm³	Sx(cb) 10³mm³	Sx(st) 10³mm³	Sx(sb) 10³mm³
1	Composite Beam	Below slab	376.11	476.89	22.5	5032.08	163.03	60396.80	13055.76	27459.34	27459.35	-10963.32

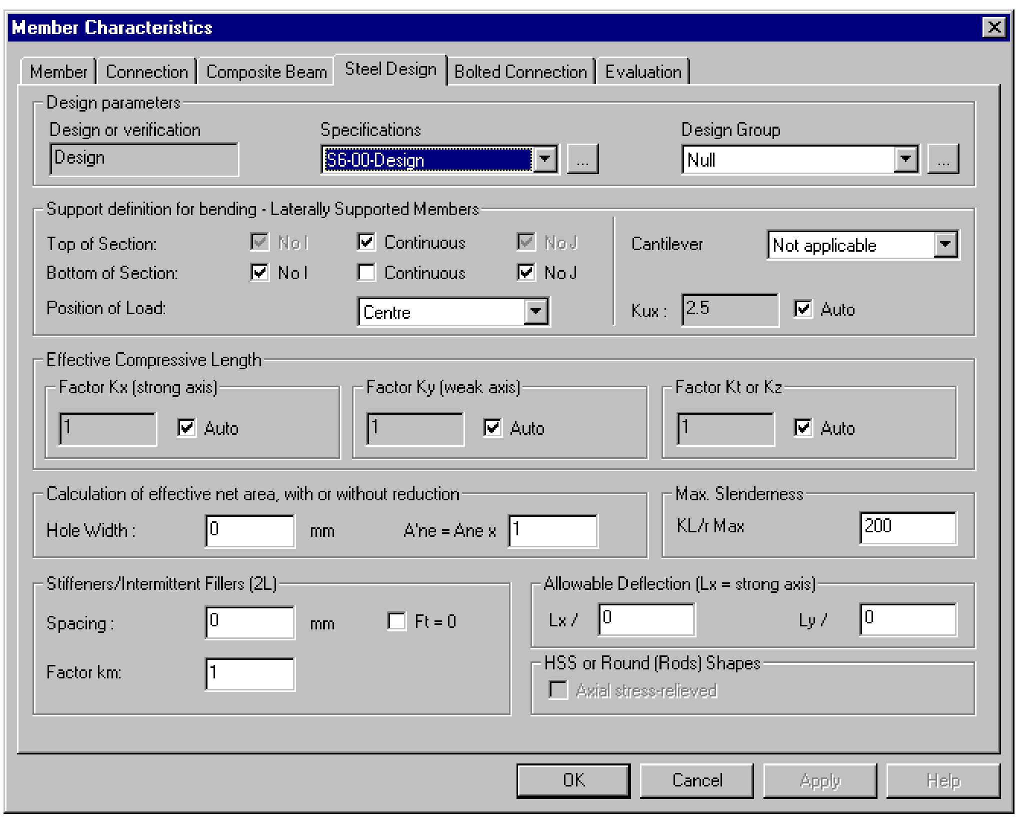
Design Criteria and Specifications

Steel Specifications

- Go to **Structure** / **Specifications** / **Steel** and consult default values used in the *S6-00-Design* steel specification.

The Steel Design tab

- Go back to the **Member Characteristics** dialog box (double click on the beam) and select the **Steel Design** tab. Select specification *S6-00-Design* among the drop-down list box. Specify a continuous lateral support at the top of the steel section.

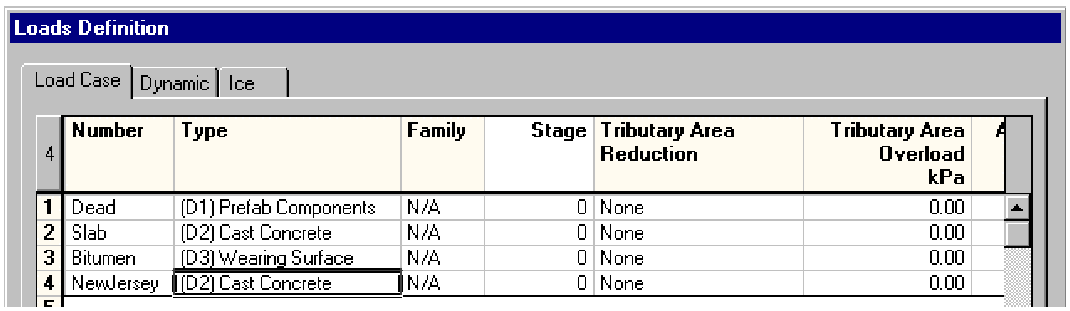


- Press OK to save data and exit the dialog box.

Load Cases and Combinations

Load Cases

- Go to **Loads / Load Cases / Definition**. Insert lines and define load case titles and types according to standard S6-00. The "Stage" column will be completed after the analysis to inform the engineer about the stage where the load case was used.

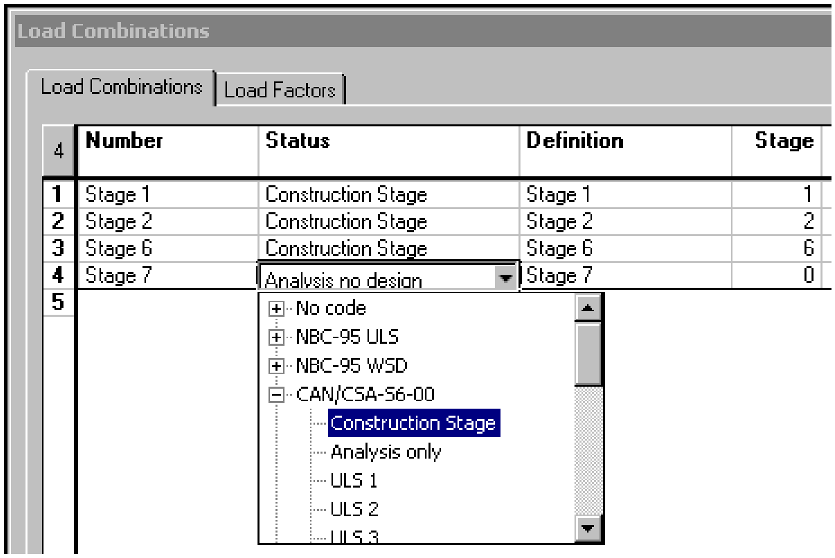


4	Number	Type	Family	Stage	Tributary Area Reduction	Tributary Area Overload kPa
1	Dead	(D1) Prefab Components	N/A	0	None	0.00
2	Slab	(D2) Cast Concrete	N/A	0	None	0.00
3	Bitumen	(D3) Wearing Surface	N/A	0	None	0.00
4	NewJersey	(D2) Cast Concrete	N/A	0	None	0.00

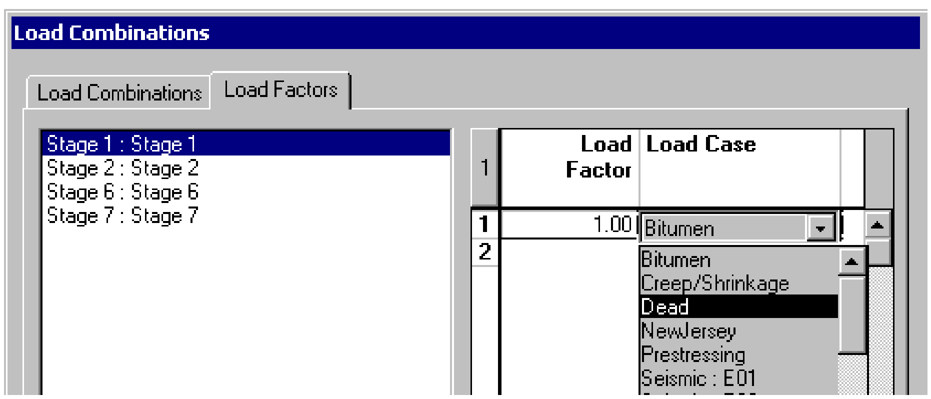
- Press OK.
- Apply load cases on the beam.

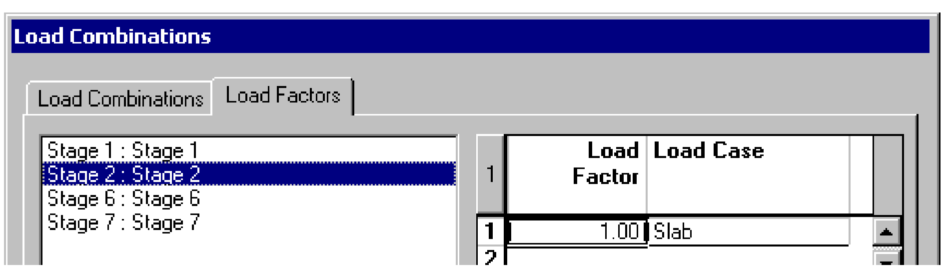
Load Combinations (Stages)

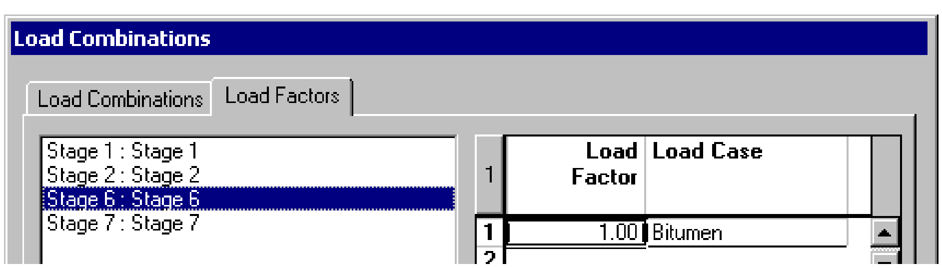
- Go to **Loads / Load Combinations / Definition**. Insert four lines; each one will represent a construction stage. Select a *Construction Stage* status and indicate the stage number in the "Stage" cell.

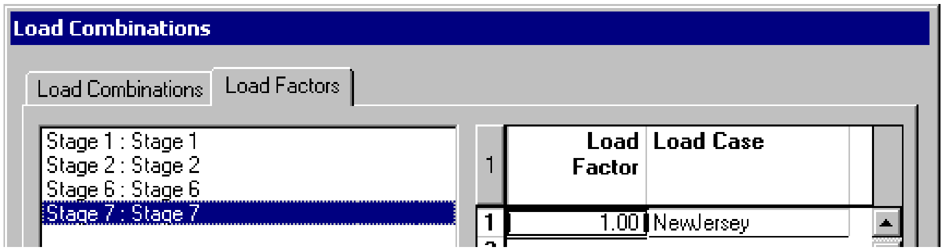


- Go to the **Load Factors** tab. Highlight a load combination in the left part of the dialog box and insert a line in the right part. Select the proper load case in the "Load Case" column. The load factor must be equal to 1.0 (service loads). Do the same for each construction stage load combination.







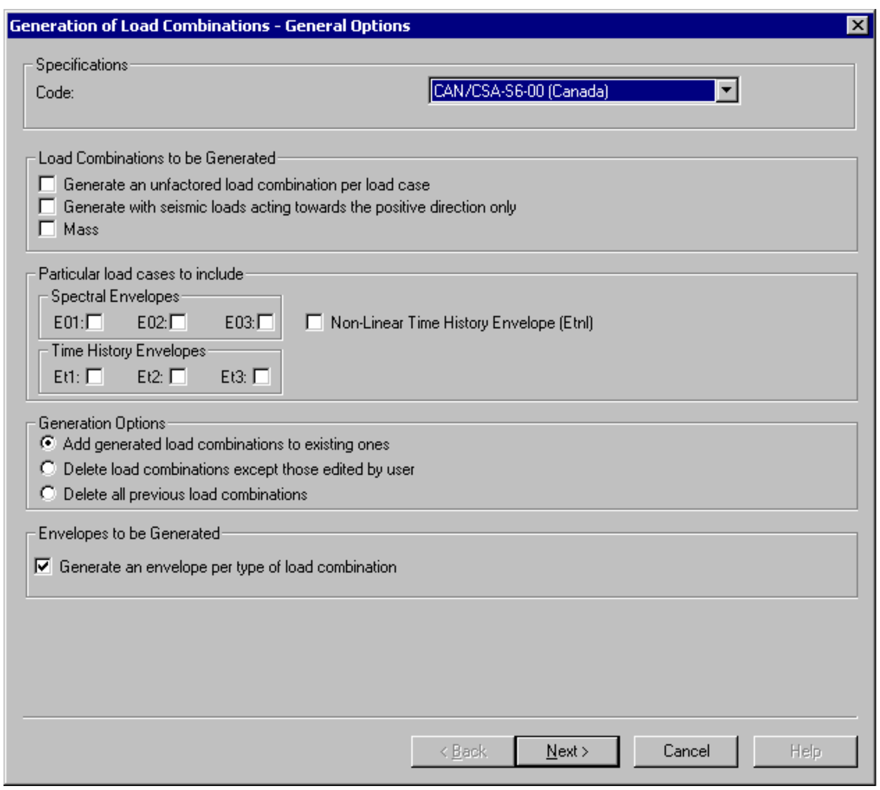


- Press OK.

We are going to use the **Load Combination Generation Wizard** to create other load combinations that are required per code S6-00.

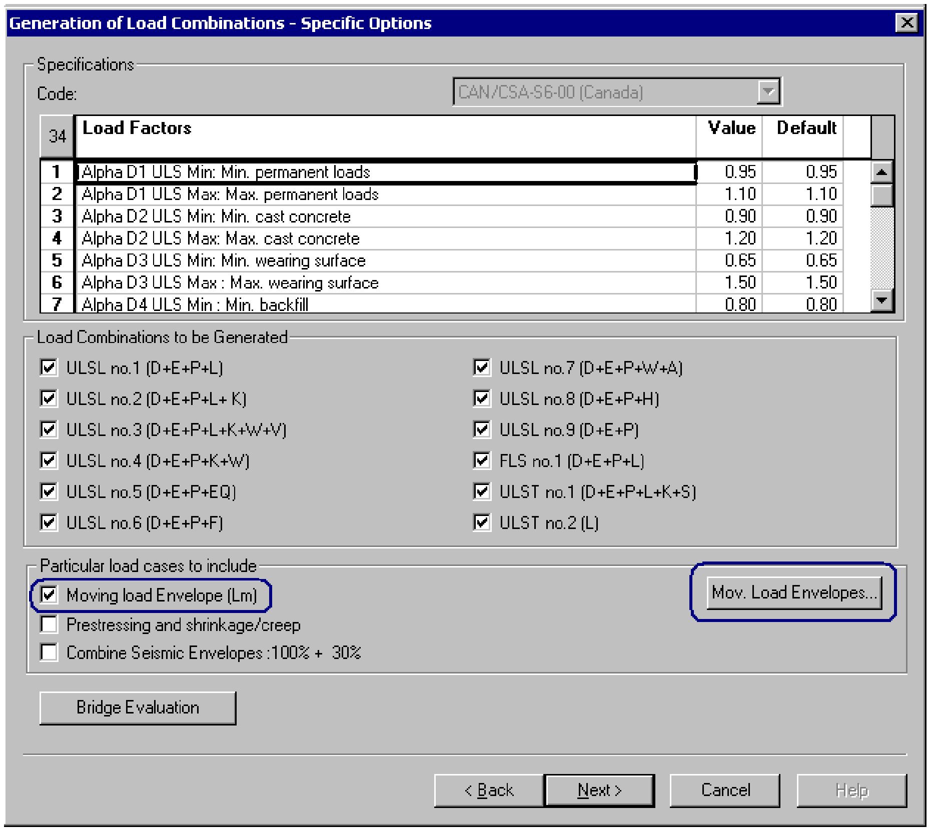
Load Combination Generation Wizard

- Go to **Loads / Load Combinations / Generation Wizard**.
- In the **General Options** page, select a code among the drop-down list box. Activate option *Add generated load combinations to existing ones* to avoid the deletion of construction stage load combinations. Activate the generation of envelopes.

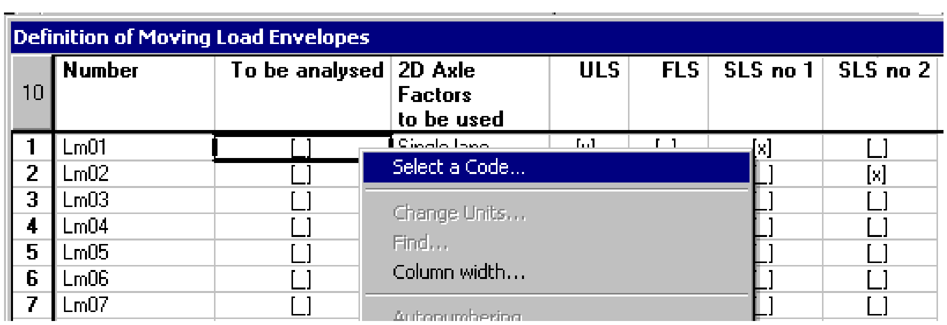


- Click "Next".

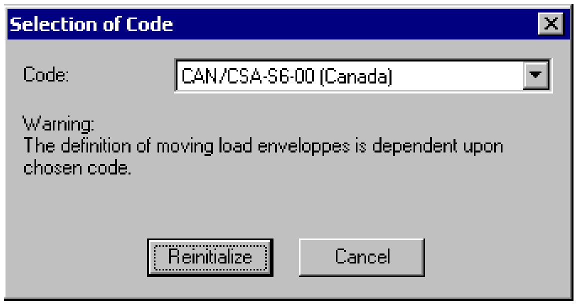
- In the **Specific Options** page, include the moving load envelopes in the generation by checking the box *Moving load Envelope*.



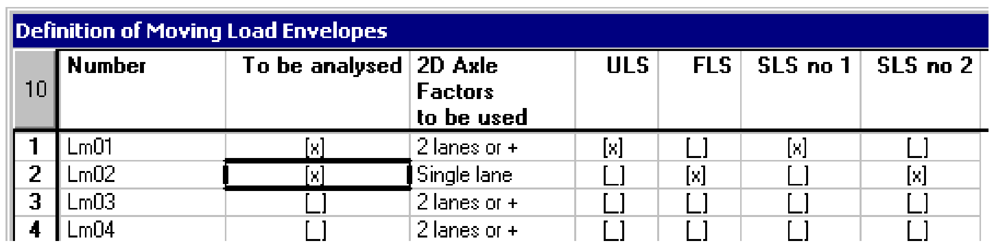
- Press the button **Moving Load Envelopes** to open the spreadsheet **Definition of Moving Load Envelopes**. Click in a cell, right click to open the contextual menu and select the function **Select a Code**.



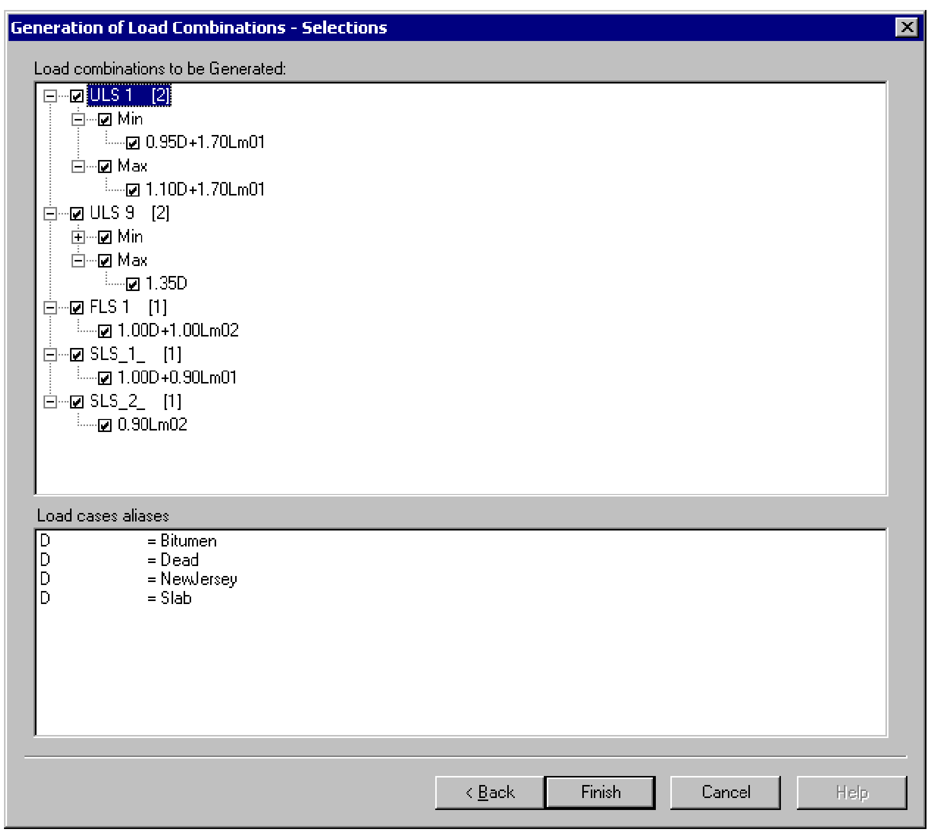
- Select the code to be considered for the generation of moving load cases and press the "Reinitialize" button.



- The spreadsheet appears with default values (2D axle factors and required load combinations) that are based on code S6-00.



- Activate the basic envelopes Lm01 and Lm02 and press OK to exit the spreadsheet. You will come back to the **Specific Options** page. Click the *Next* button.



- Press *Finish*. The **Load Combination Definition** spreadsheet will be displayed on screen.

Load Combinations

Load Combinations | Load Factors

11	Number	Status	Definition	Stage
1	Stage 1	Construction Stage	Stage 1	1
2	Stage 2	Construction Stage	Stage 2	2
3	Stage 6	Construction Stage	Stage 6	6
4	Stage 7	Construction Stage	Stage 7	7
5	ULS 1:max2	ULS 1	1.10D+1.70Lm01	0
6	ULS 1:min1	ULS 1	0.95D+1.70Lm01	0
7	ULS 9:max4	ULS 9	1.35D	0
8	ULS 9:min3	ULS 9	1.35D	0
9	FLS 15	FLS 1	1.00D+1.00Lm02	0
10	SLS_1_6	SLS 1	1.00D+0.90Lm01	0
11	SLS_2_7	SLS 2	0.90Lm02	0

Graphic results for composite beams (**Results / Load Combination / Stresses – Composite Beams**) will be available for load combinations that have a status such as *Construction Stages* and *SLS 1* (Serviceability Limits States) only.

Add "*Special*" Load Combinations

If you want to obtain steel design results for each construction stage, you can do as follows:

- Copy construction stage load combinations along with load factors using the **Duplicate** function available in the contextual menu. The duplicated lines will be inserted at the end of the spreadsheet. Change the names and modify statuses to *Ultimate*. Do not delete stage numbers.

Load Combinations

Load Combinations | Load Factors

15	Number	Status	Definition	Stage	D
1	Stage 1	Construction Stage	Stage 1	1	
2	Stage 2	Construction Stage	Stage 2	2	
3	Stage 6	Construction Stage	Stage 6	6	
4	Stage 7	Construction Stage	Stage 7	7	
5	ULS 1:max2	ULS 1	1.10D+1.70Lm01	0	
6	ULS 1:min1	ULS 1	0.95D+1.70Lm01	0	
7	ULS 9:max4	ULS 9	1.35D	0	
8	ULS 9:min3	ULS 9	1.35D	0	
9	FLS 15	FLS 1	1.00D+1.00Lm02	0	
10	SLS_1_6	SLS 1	1.00D+0.90Lm01	0	
11	SLS_2_7	SLS 2	0.90Lm02	0	
12	Stage 1_design	Ultimate	Stage 1	1	
13	Stage 2_design	Ultimate	Stage 2	2	
14	Stage 6_design	Ultimate	Stage 6	6	
15	Stage 7_design	Ultimate	Stage 7	7	

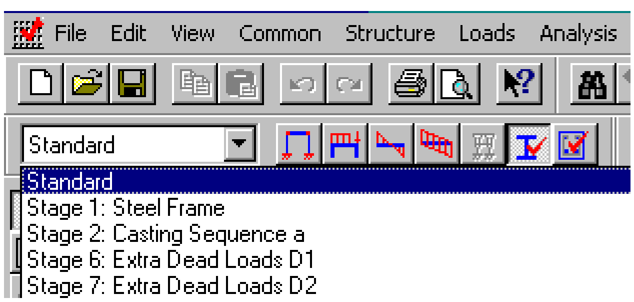
- Close the dialog box.

Analysis and Results

Design

- Launch the steel design by pressing the icon of Tools toolbar. Click on the "Analyse" button posted in the **Analysis and Design** dialog box. When the design is completed, close the dialog box.

The *Design Results* icon is automatically activated when the design results are available. Click on the arrow on Activation toolbar. You will find the "Standard" steel design results, obtained from ultimate load combinations, and special steel design results for construction stages.

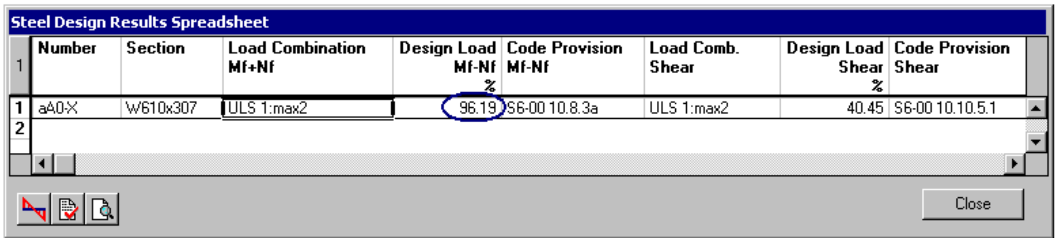


Steel Design Results Spreadsheet

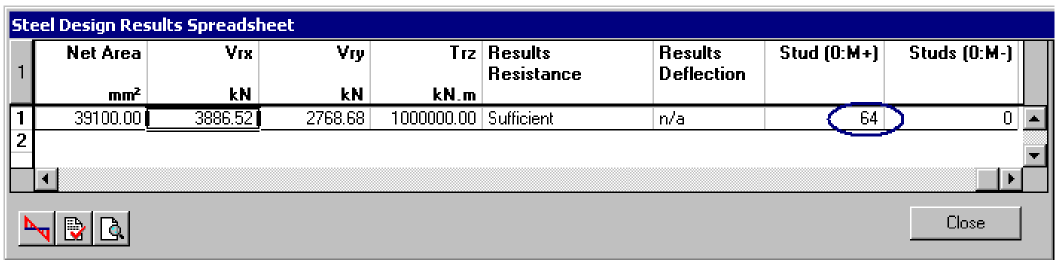
Standard

- Open the **View Options** dialog box by pressing icon of View toolbar. Display the beam's design load through the **Results** tab. We can see that the beam is working at about 90% and 100% of its capacity.

- Open the spreadsheet by double clicking on the beam. The chosen steel shape is **W610x307.**

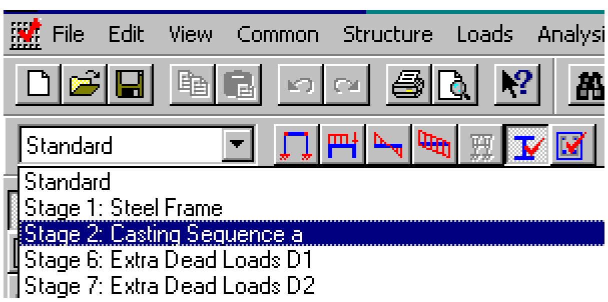


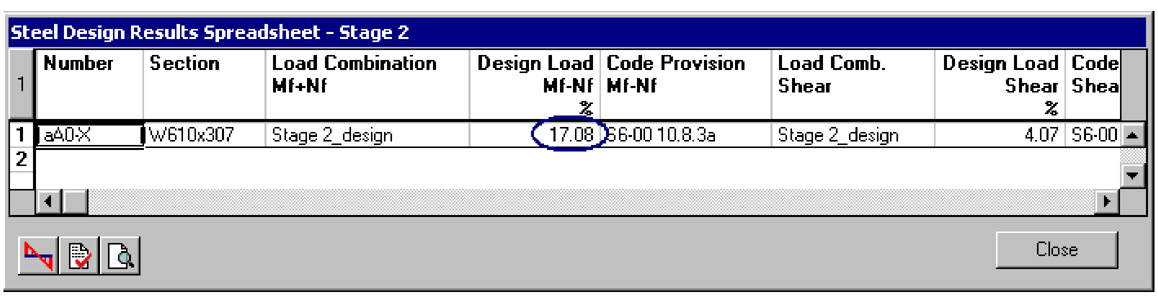
Do not forget that results are supplied for the most critical load combination, which controlled the design for this member. At the far right of the spreadsheet, you will find the required number of studs in the positive bending zone.



Construction Stage

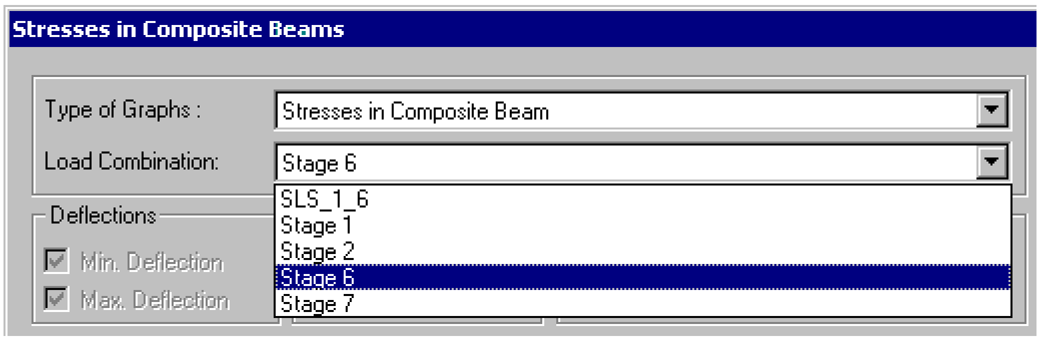
- Select a construction stage load combination on Activation toolbar and look at the design load in the steel design results spreadsheet.



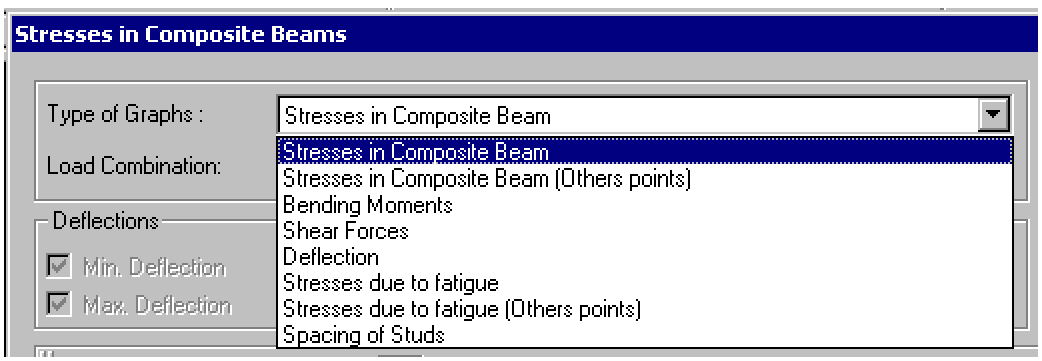


Graphical Results (Stresses) for Composite Beams

- Select the member on screen and go to **Results / Load Combinations / Stresses – Composite Beams**.
- Select a serviceability (construction stage or SLS) load combination.

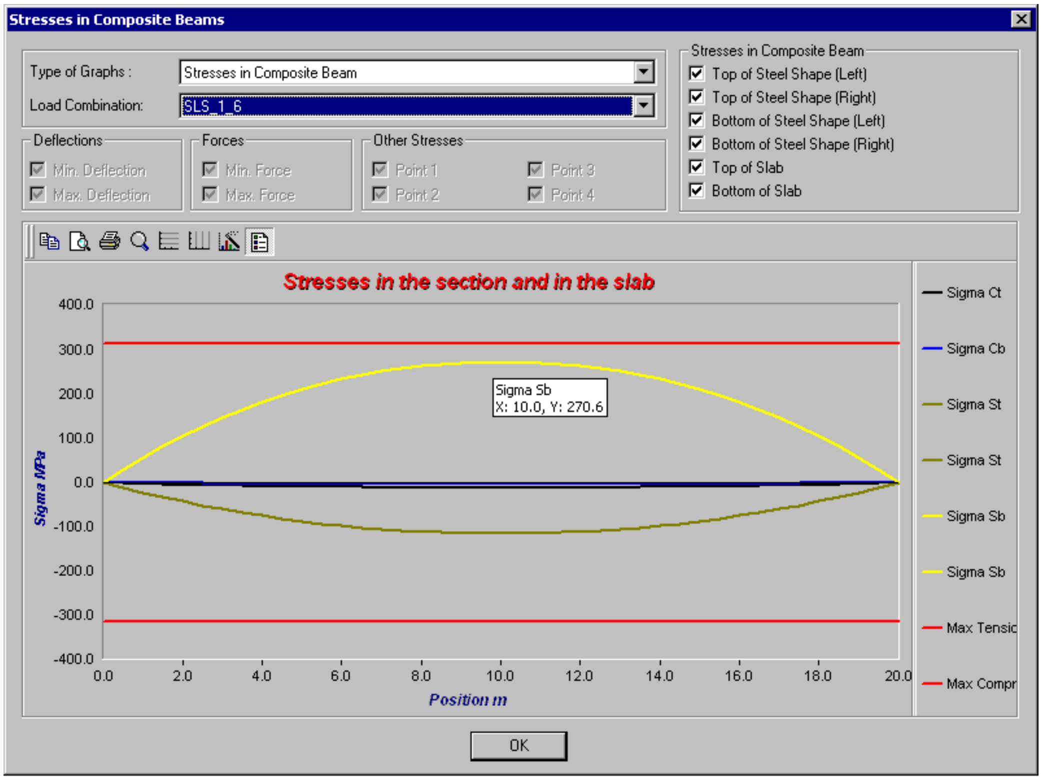


- Select a diagram among the list box.



Note: The graphs that include stresses at other points are calculated stresses at other points on the steel beam. These points are specified by the user through a tool that is accessible in the **Composite Beam** tab of the **Member Characteristics** dialog box. The tool will be described further on in this example.

Stresses - Load Combination SLS_1_6:

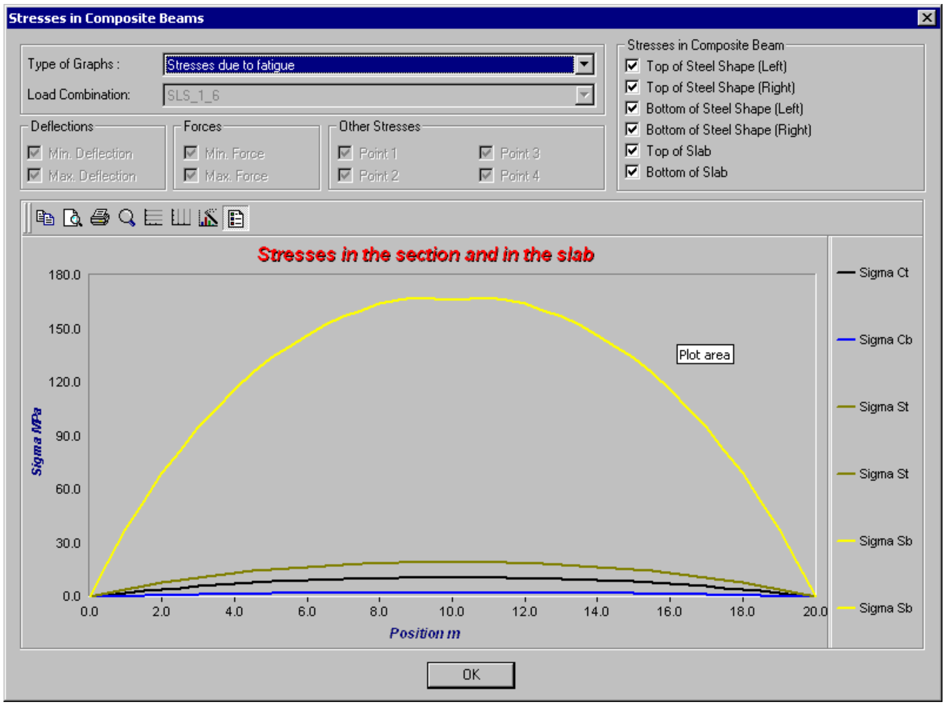


Curves must be located within the limits of maximum tension and compression stresses, which are around +/- 300 MPa.

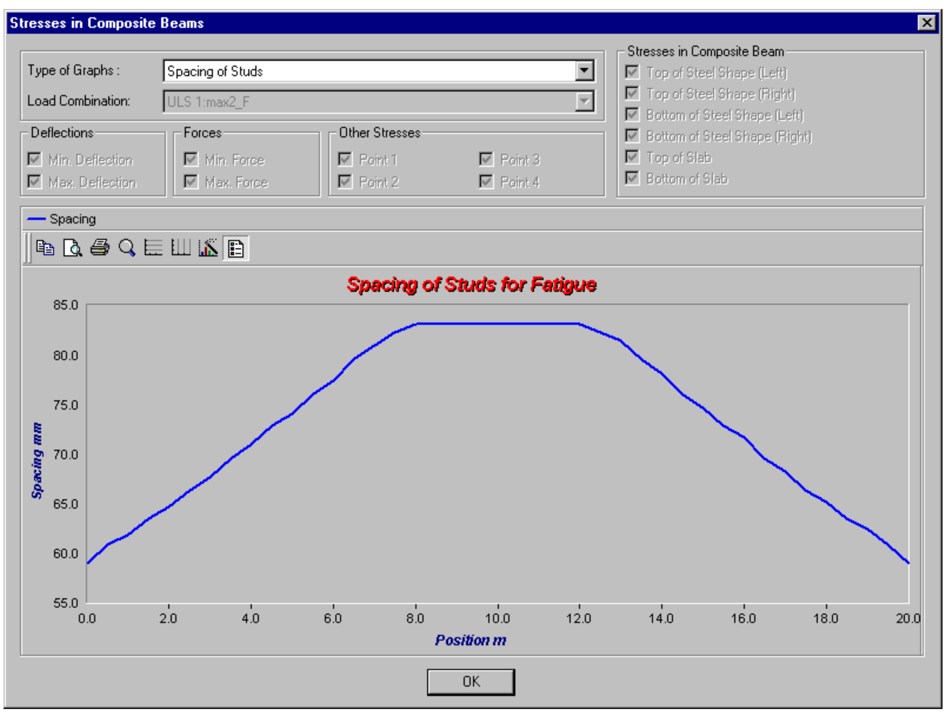
The maximum stress (Sigma Sb) occurs at the bottom flange of the **W610x307**. Place your cursor on the curve to display the coordinates. The maximum stress is equal to 270.6 MPa.

Stresses due to fatigue:

N. B. At least one "fatigue" envelope or 2 "fatigue" load combinations are required to obtain such results.



Spacing of Studs

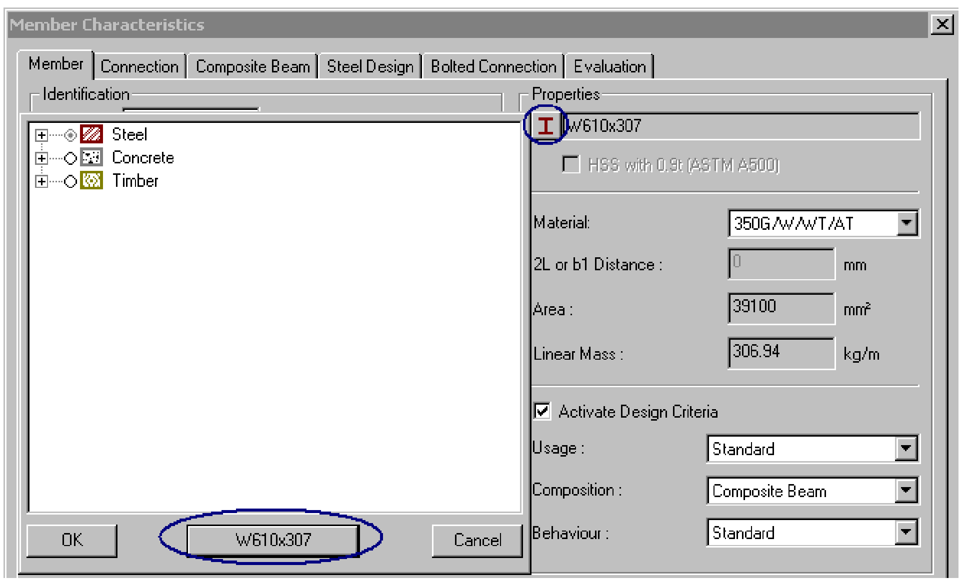


Stresses at Other Locations on the Beam

VisualDesign has a tool that consists in calculating stresses anywhere on the steel shape. Up to four points can be defined by the user. We are going to use this tool, which is available in the **Composite Beam** tab of **Member Characteristics** dialog box.

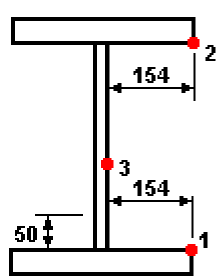
Procedure:

- Before activating this tool, we have to take not of the shape dimensions.
- Activate the Structure mode and Member icon and double click on the member to open the **Member Characteristics** dialog box.
- Click on the *I Beam* icon and click on the button "W610x307" to open the **W610x307** dialog box.

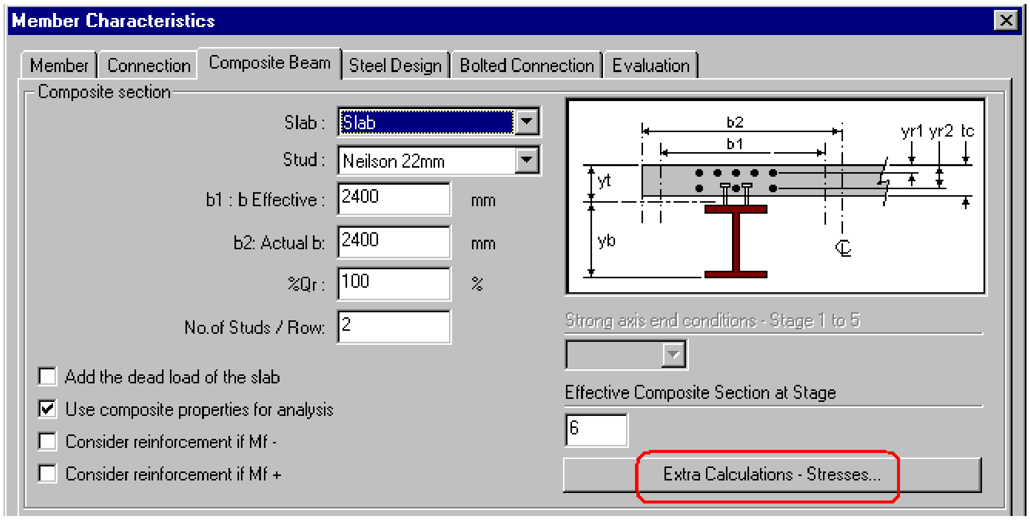


Take note of dimensions d, b, t, and w (653mm, 330mm, 39.9mm, and 22.1mm). Close the **W610x307** dialog box and press the [Cancel] button to close the *Shape* selection tree.

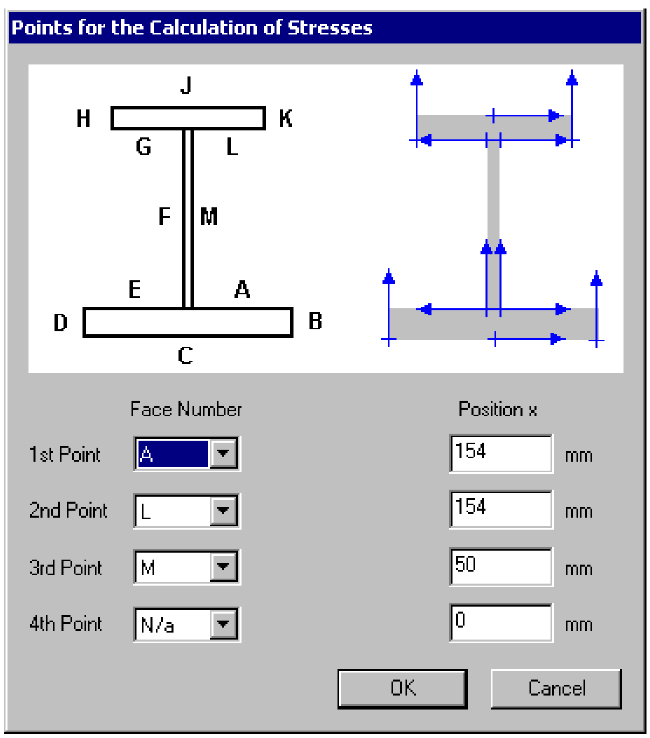
We want to get stress results at these points on the beam:



- Go to the **Composite Beam** tab of **Member Characteristics** dialog box.
- Click on the button "Extra Calculations – Stresses".



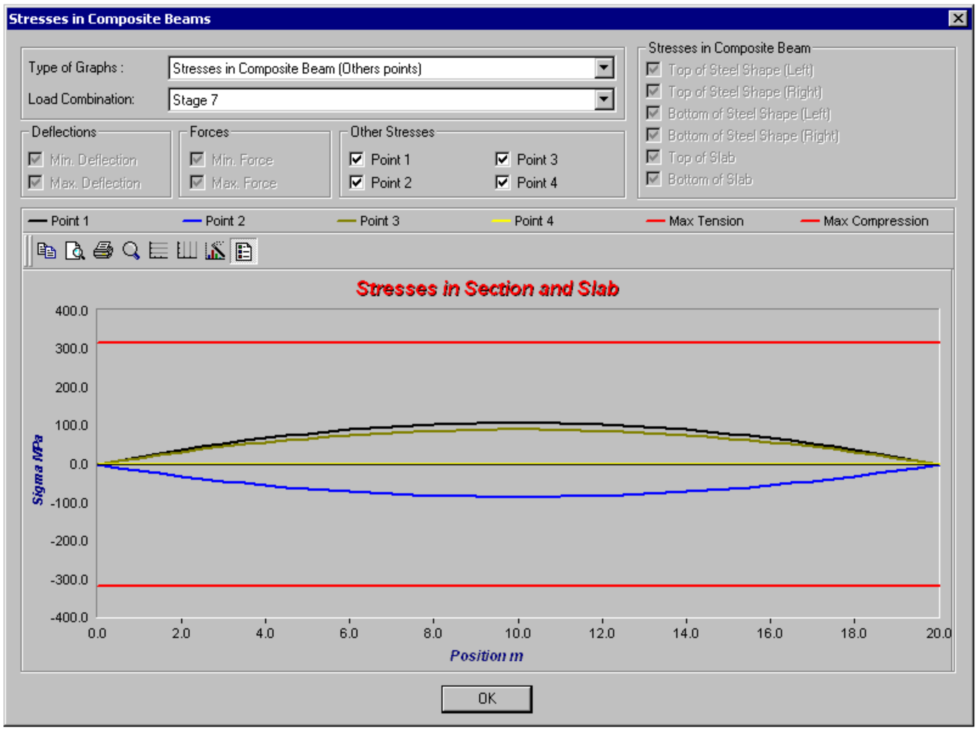
- The following dialog box will appear on screen. Select faces on the beam and specify the distance to each point with respect to the convention shown in the right part of the dialog box. Distances can be positive or negative.



Click OK to save data and exit the dialog box.

- Launch the design again.

- Select the member and go to **Results / Load Combinations** / **Stresses – Composite Beams**. Select the diagram called *Stresses in Composite Beam (Other Points)*:



- Close the dialog box.

EXAMPLE 5

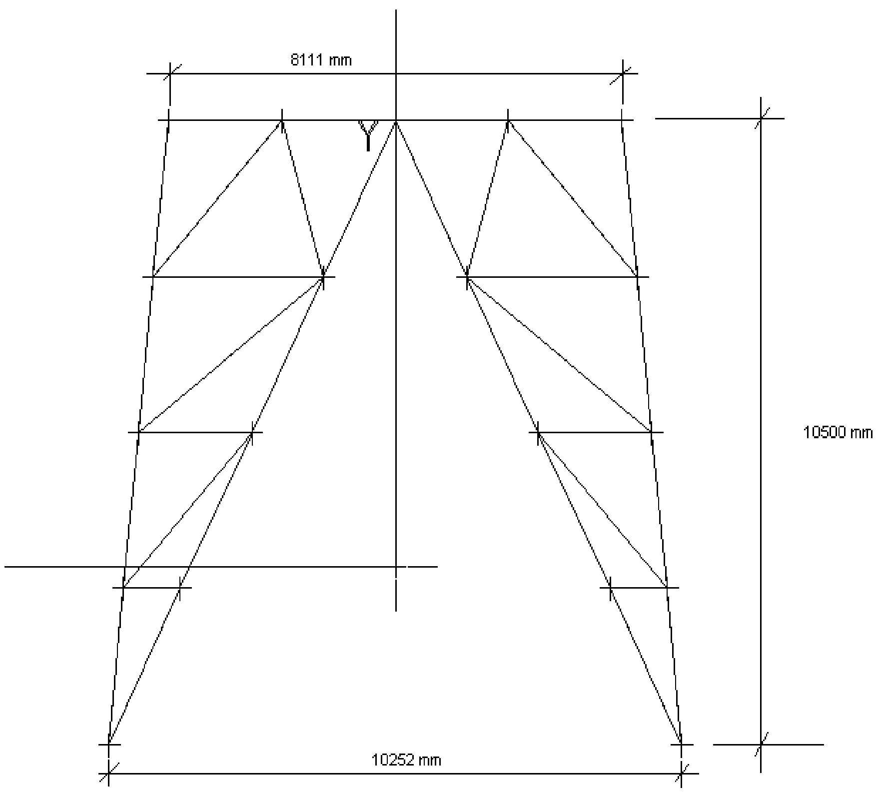
Advanced Modelling

Modelling Tower Legs

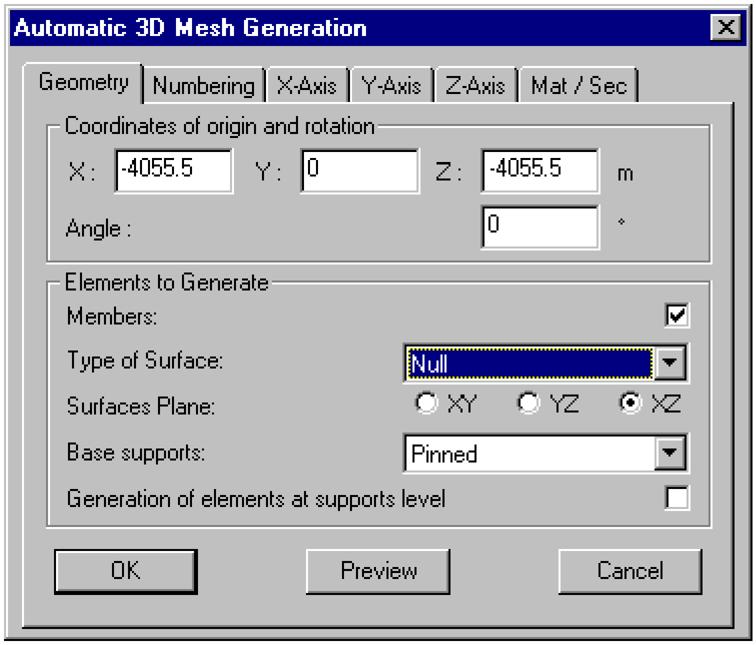
We are going to model 3D tower legs as shown in the 2D view below.

Modelling a tower is like any other modelling, except that there are some particularities that we are going to explain in this example.

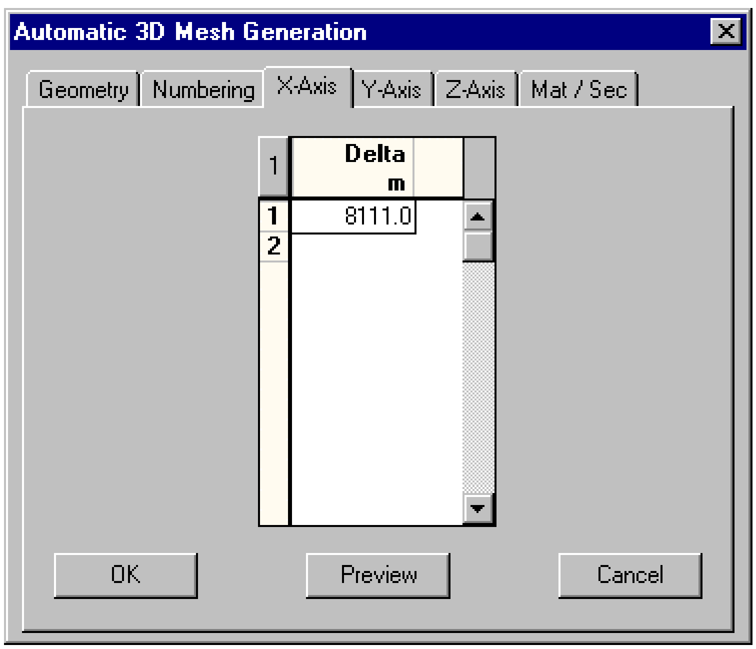
First, we are going to model one leg and then, copy and paste it to complete the final model. Dimensions are as follows (members are subdivided in equal segments):



- Go to **Project Configuration** and select the **Units** tab. Expand the *Length* root and the *Dimensions* branch, and activate the mm unit.
- Generate a cube using the **Automatic Mesh Generation** tool (**Structure/Tools** menu). Insert the following coordinates: (-4.0555, 0, -4.0555) in order to position the centre of tower at (0,0,0).

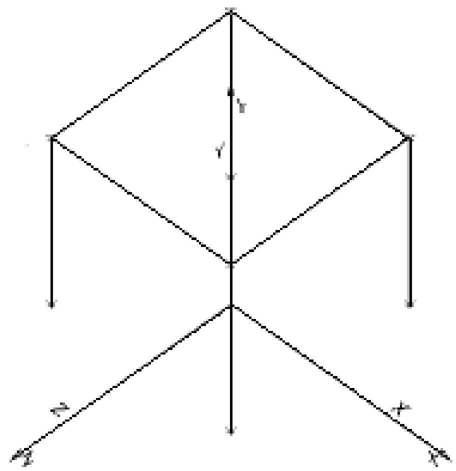


- Select the **X Axis** tab and enter 8111.00 mm as x coordinate.

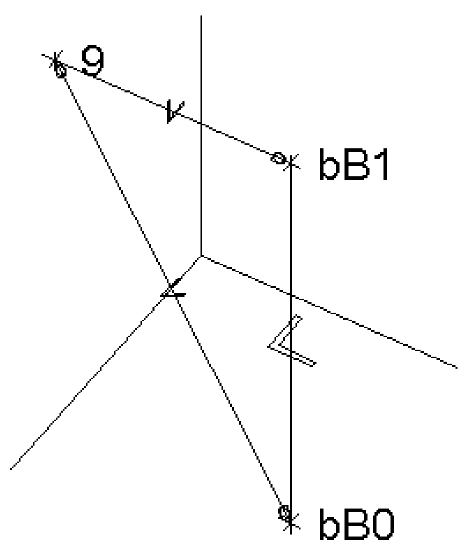


- Select the **Y Axis** tab and enter the y coordinate 10500 mm.
- Select the **Z Axis** tab and enter 8111 mm as z coordinate.
- Click OK.

This is the cube you are going to see on your screen:



- Display node numbers using the **View** tab of **View Options**.
- Display the shape outlines and end conditions using the **Attributes** tab of **View Options**.
- Select the horizontal member.
- Split this member in two using function **Multiple Split** of **Split** toolbar.
- Keep nodes 9, bB1 and bB0. Delete other nodes.



- Double-click on node bBO.
- The **Node Characteristics** dialog box will appear on your screen. Enter the following coordinates: (5126, -3000, 5126).
- Add a member between nodes bB0 and 9 and insert the following information in the **Member Characteristics** dialog box:

  Shape: L89X89X7.9

  Material: 350G/W/WT/AT

  End conditions: Pinned-Pinned

Note: Before splitting a member or using the Copy/Paste function, members must be well defined (end conditions, shapes, materials, etc.) to keep the properties of original member after splitting and copying.

- Double-click on the horizontal member and insert parameters as follows:

  Shape: L89X89X7.9

  Material: 350G/W/WT/AT

  End Conditions: Fixed-Pinned

  (Make sure that the pinned end is located right, at vertical member level.)

- For the vertical member:

  Shape: L152X152X19

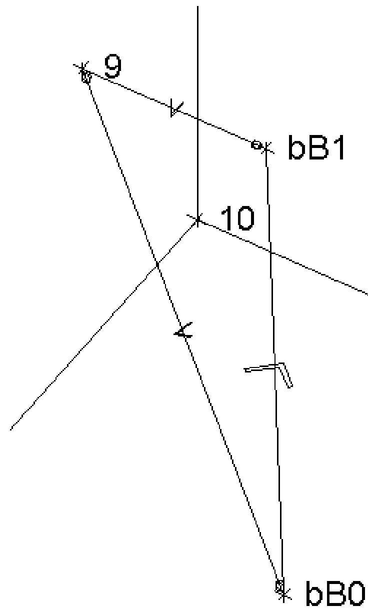
  Material: 300W/WT

  End Conditions: Fixed-Fixed

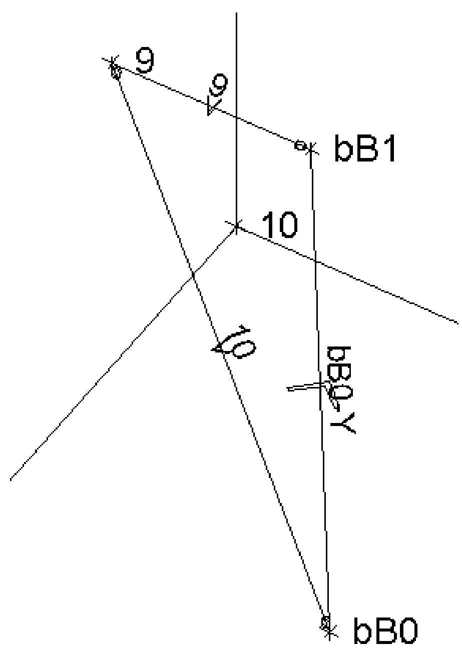
Before splitting members, it is recommended to correct the beta angle orientation, if needed.

Orientation of Beta Angle

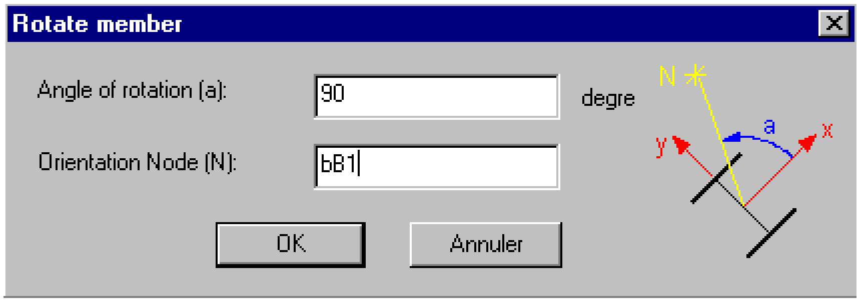
- Create node No.10 at (0,0,0).



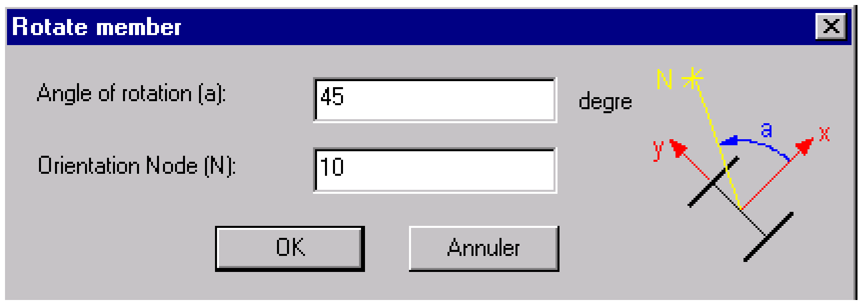
- Beta angle is very important when modelling towers. It is useful to orient steel angles for upright and secondary members (bracings).
- Display member numbers.



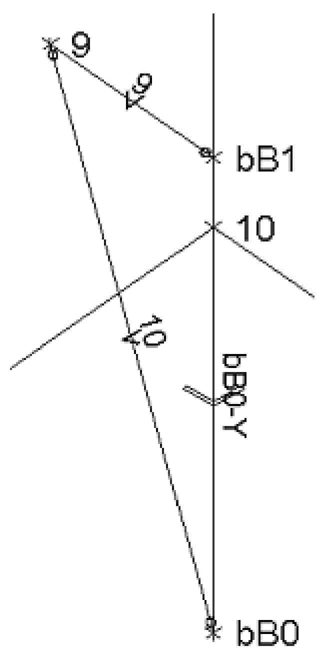
- Orient member 10's strong axis towards node bB1 using the **Rotate** function.



- Orient member bBOY strong axis at 45 degrees relatively to node 10, using the **Rotate** function.

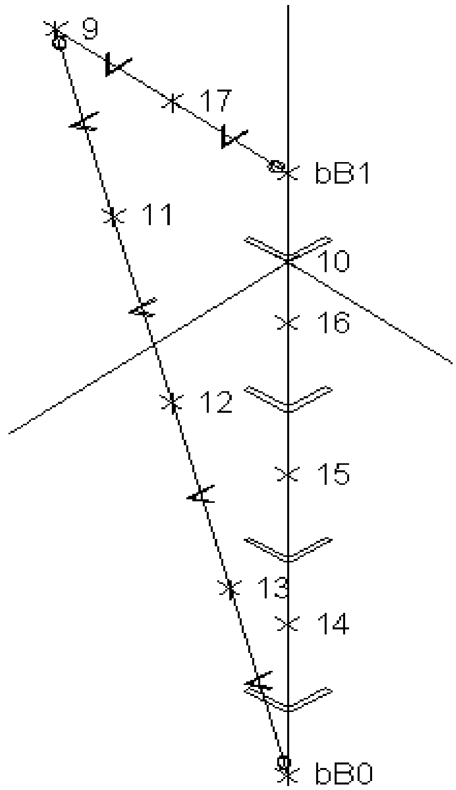


Members are correctly oriented. Steel angles can be oriented according to an orthogonal axis system or major/minor axis system.



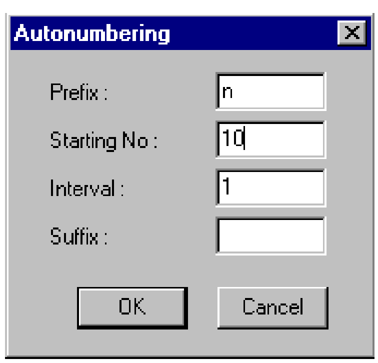
- Split member 9 in two.
- Split member 10 and bBOY in four equal segments.

You will obtain the following structure:



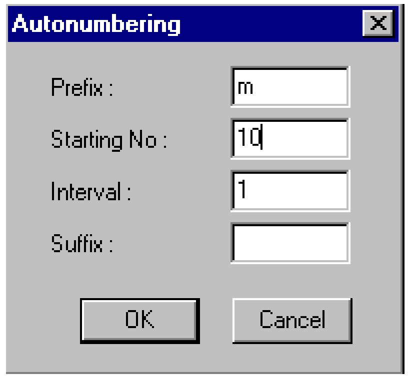
The numbering of new elements (nodes, members, floors, etc.) is automatic. However, you can renumber them, as you like.

- Select the **Nodes** spreadsheet and select the whole *Number* column. Right click and select function **Auto numbering** in the contextual menu. Enter a prefix, a starting number for numbering and an interval.



- Click OK.

- Do the same in the **Members** spreadsheet:



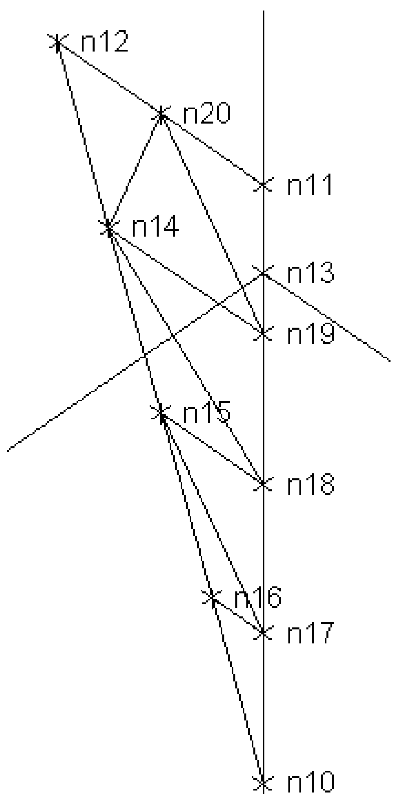
- Activate the **Add** mode and add a member between node 16 and 17 (bracing).
- Insert this information in the dialog box:

  Shape: L51X51X4.8

  Material: 300W/WT

  End Conditions: Pinned-Pinned

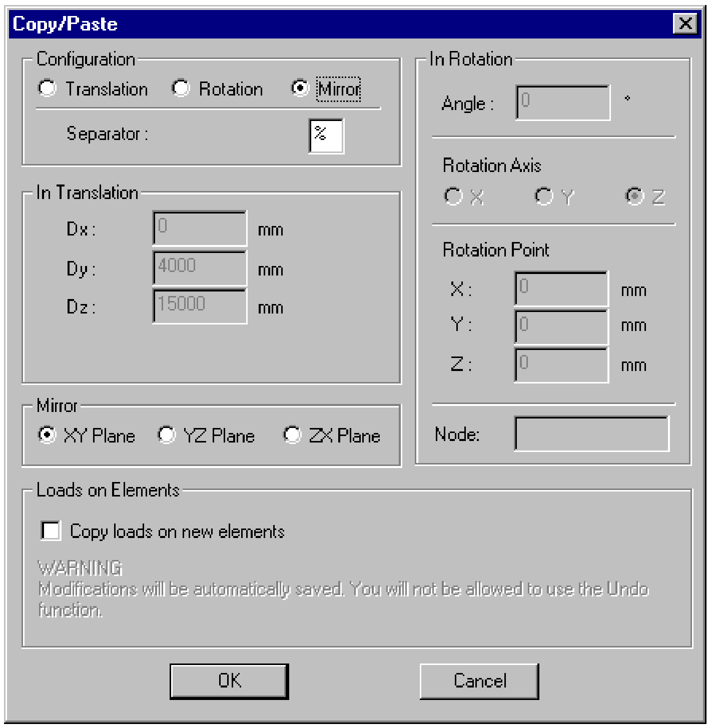
- Do the same to create other bracings.
- All secondary members (bracings) are pinned at each end.



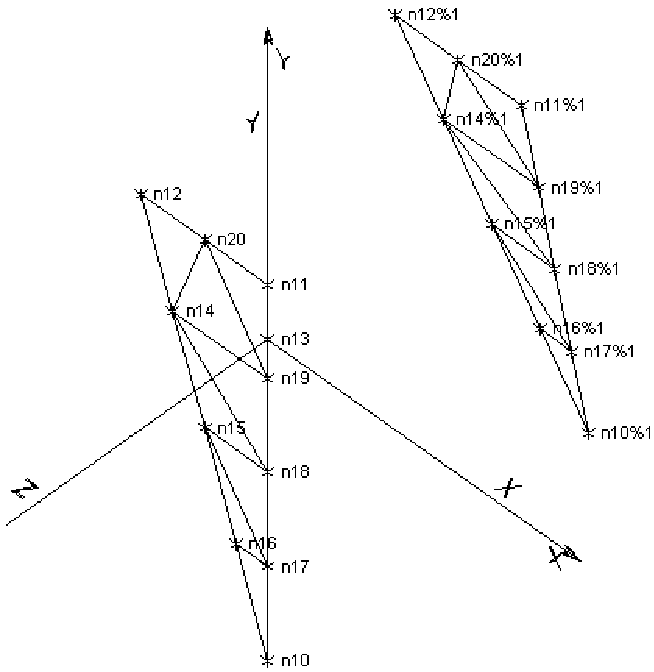
Use the "Copy/Paste" Function

Create other tower legs with the **Copy/Paste** function:

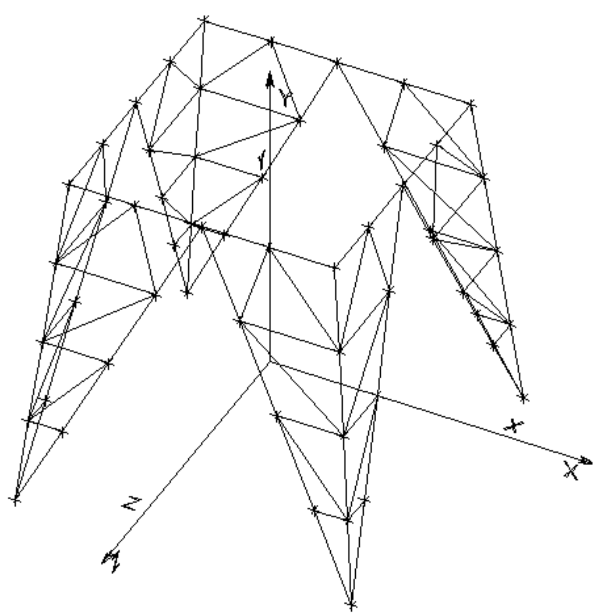
- Activate the "Extended Window" selection mode and select the whole tower leg.
- Press the short cut keys **[Ctrl] + C** to copy this selection. The Copy/Paste dialog will open on screen.
- Do a mirror copy relative to the XY plane. Insert the following information in the **Copy/Paste** dialog box. Click OK.



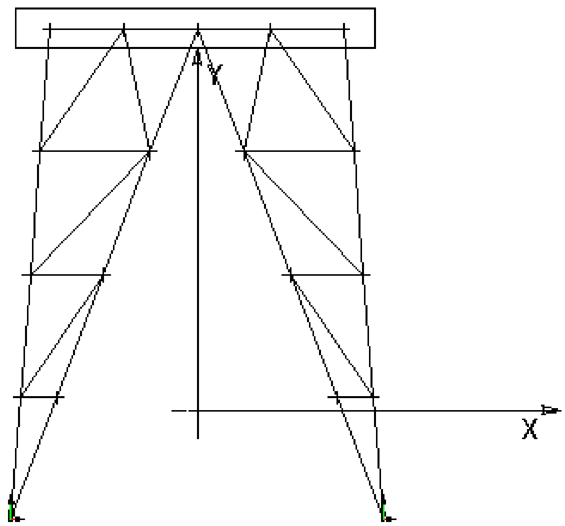
- Press the short cut keys **[Ctrl] + V** to paste the selection.



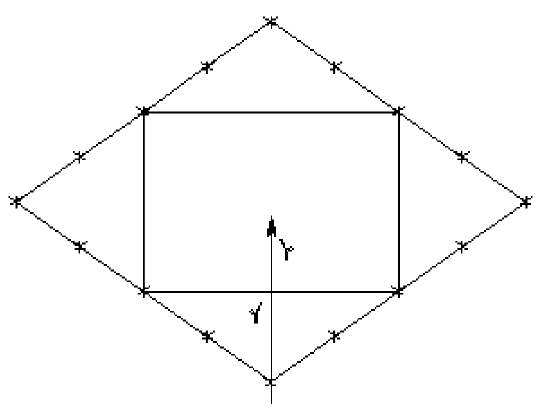
- Select all.
- Copy the selection ([**Ctrl] + C**).
- In the Copy/Paste dialog box, activate the "Rotation" configuration. The rotation axis is Y and the angle is set to 90 degrees.
- Press the short cut keys [**Ctrl] + V** three times.
- Remove the display of node number in the **View Options** dialog box.



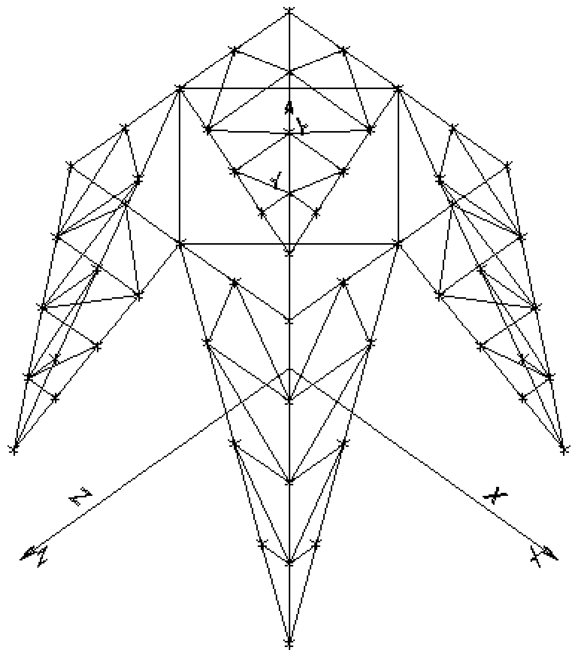
- Select an XY view using the **Camera** function.
- Select top members.
- Mask the rest of the structure using the **Mask** function.



- Select a 45 degrees view.
- Re-number nodes as explained before.
- Activate the **Add** mode and the Member icon. Add members as indicated in the figure below:



- Define new members properties.
- Unmask the rest of the structure.


Y
Z
X

E X A M P L E 6

Timber Design

Basic Principles

Timber Properties

Pre-defined materials are listed in the **Timber Properties** spreadsheet, which is located in **Common / Materials / Timber** menu.

Timber Properties Spreadsheet

92	Number	Classification	Grade	Species	Available Cuts	E MPa	E05 MPa
10	V1A_Northern_s	Visual	Select	Northern	[A]2x4...4x16	7500.00	5500.00
11	V1A_Northern_1	Visual	No.1	Northern	[A]2x4...4x16	7000.00	5000.00
12	V1A_Northern_3	Visual	No.3	Northern	[A]2x4...4x16	6500.00	4000.00
13	V1B_DFirL_c	Visual	Construction	D Fir-L	[B]2x2, 2x4, 4x4	10000.00	5500.00
14	V1B_DFirL_st	Visual	Standard	D Fir-L	[B]2x2, 2x4, 4x4	9000.00	5000.00
15	V1B_HemFir_c	Visual	Construction	Hem-Fir	[B]2x2, 2x4, 4x4	10000.00	6000.00
16	V1B_HemFir_st	Visual	Standard	Hem-Fir	[B]2x2, 2x4, 4x4	9000.00	5500.00
17	V1B_SPF_c	Visual	Construction	S-P-F	[B]2x2, 2x4, 4x4	9000.00	5500.00
18	V1B_SPF_st	Visual	Standard	S-P-F	[B]2x2, 2x4, 4x4	8000.00	5000.00
19	V1B_Northern_c	Visual	Construction	Northern	[B]2x2, 2x4, 4x4	6500.00	4000.00
20	V1B_Northern_s	Visual	Standard	Northern	[B]2x2, 2x4, 4x4	6000.00	3500.00
21	V1C_DFirL_ss	Visual	Select	D Fir-L	[C]6x10, 8x12, 10x14...	12000.00	8000.00
22	V1C_DFirL_1	Visual	No.1	D Fir-L	[C]6x10, 8x12, 10x14...	12000.00	8000.00
23	V1C_DFirL_2	Visual	No.2	D Fir-L	[C]6x10, 8x12, 10x14...	9500.00	6000.00

The following parameters are sometimes confusing, so here is a description of each of them:

Classification:

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, 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.

Nomenclature

The name that appears in the in the **Timber Properties** spreadsheet is explained below.

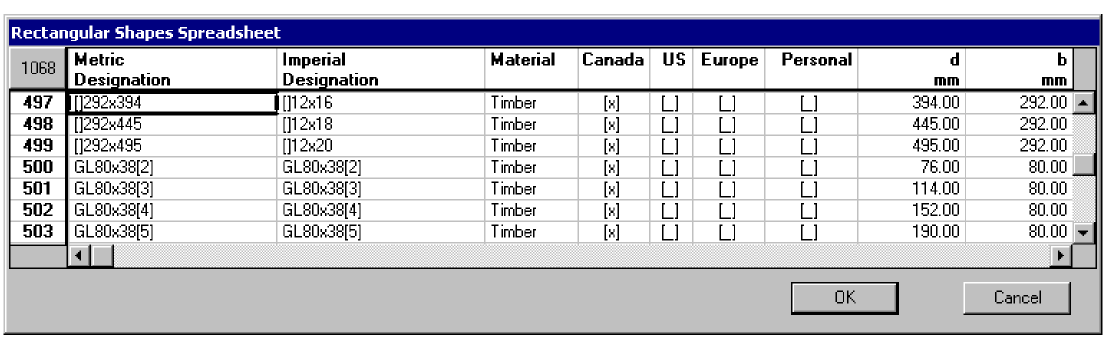
The first term represents the classification, the second term, its species, and the third, its grade (quality).

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*.

Shape Designation

Sections are listed in the **Rectangular Sections** spreadsheet (**Common / Shapes**) for timber materials. Shapes beginning with this symbol [ ] are sawn timber (standard) sections. Glue laminated sections begin with letters GL.


Rectangular Shapes Spreadsheet

1068	Metric Designation	Imperial Designation	Material	Canada	US	Europe	Personal	d mm	b mm
497	[]292x394	[]12x16	Timber	[x]	[ ]	[ ]	[ ]	394.00	292.00
498	[]292x445	[]12x18	Timber	[x]	[ ]	[ ]	[ ]	445.00	292.00
499	[]292x495	[]12x20	Timber	[x]	[ ]	[ ]	[ ]	495.00	292.00
500	GL80x38[2]	GL80x38[2]	Timber	[x]	[ ]	[ ]	[ ]	76.00	80.00
501	GL80x38[3]	GL80x38[3]	Timber	[x]	[ ]	[ ]	[ ]	114.00	80.00
502	GL80x38[4]	GL80x38[4]	Timber	[x]	[ ]	[ ]	[ ]	152.00	80.00
503	GL80x38[5]	GL80x38[5]	Timber	[x]	[ ]	[ ]	[ ]	190.00	80.00

Glue laminated sections:

A "Glulam" composition is selected for glue-laminated sections. The direction of laminations is indicated using the member local axes. The numbers of laminations are also specified.

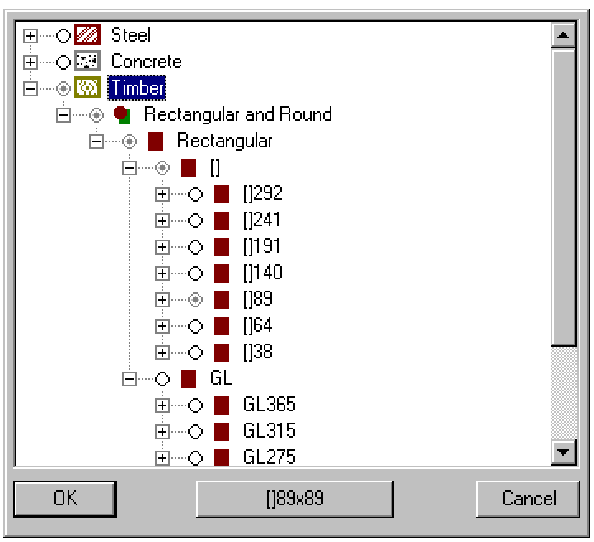
Rectangular Shapes Spreadsheet

1068	ry mm	Zy 10^3mm^3	J 10e6mm4	Composition	Dir.laminations	Nos. Laminat.	Perimeter mm
580	64.95	6252.19	1318.61	Glulam	X Local	13	1438.00
581	64.95	6733.12	1462.86	Glulam	X Local	14	1514.00
582	64.95	7214.06	1609.94	Glulam	X Local	15	1590.00
583	64.95	7695.00	1759.44	Glulam	X Local	16	1666.00

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. This *Material* selection tree is accessible through the **Member Characteristics** dialog box by pressing the I-Beam Icon.



Compatibility of Material and Section

VisualDesign does not accept some combinations of material and section. For example, a GLT type of material must be assigned to glue-laminated sections, 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 be warn by VisualDesign and the design will be stopped.

During the design process, 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.

Particular case: VisualC and VisualD Classifications

The compatibility "material and section" is tested. If some sections are incompatible, a message will be displayed to warn you. There is an exception case with Visual C and Visual D classifications. In this particular case, 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.

Design Criteria

Design criteria are activated in the **Member** tab of the **Member Characteristics** dialog box. Then, the **Timber Design** tab appears in the dialog box. In this tab, a specification or design group must be assigned to the member. The species and grade must be specified for the design. Lateral supports are applied to the member weak axis. Effective compression length factors can be entered manually. Otherwise, VisualDesign automatically detects lateral members if the project is in 3D. Bearing conditions are optional. The deflection criteria are specified in this tab. A "deflection" load combination must be created to consider deflection criteria during the design.

Specifications

VisualDesign needs specifications to design elements according to a Standard and a type of analysis (design or verification), specified shape, type of optimization (inertia, area, or height), minimum and maximum heights, service conditions (wet, dry), and treatment (Not treated, Treated incised, Treated and not incised, or treated with a fire retardant).

Design Groups

Design groups are useful for grouping elements for which you want to obtain identical sections. A specification must be assigned to a design group. The **Timber Design Groups** spreadsheet is located in **Structure / Groups / Timber**.

Insert lines in this spreadsheet and give a name to each design groups. Then, select the appropriate specifications. Design groups must then be assigned to members through the **Timber Design** tab (**Member Characteristics** dialog box).

Design groups can be created quickly using the **Group** function located in the **Structure / Groups** menu. Members that are going to be grouped must be selected first.

Analysis and Design

The timber design is launched by pressing the icon of Tools toolbar. Click on the "Analyse" button posted in the **Analysis and Design** dialog box. When the design is completed, close the dialog box.

The *Design Results* icon is automatically activated when the design results are available.

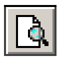
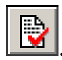
Results

View Options

Open the **View Options** dialog box by pressing icon of View toolbar. Display the colour legend and corresponding beam's design load through the **Results** tab by activating the option "Design Load". To display numerical data, activate the "Numerical" option in the same tab.

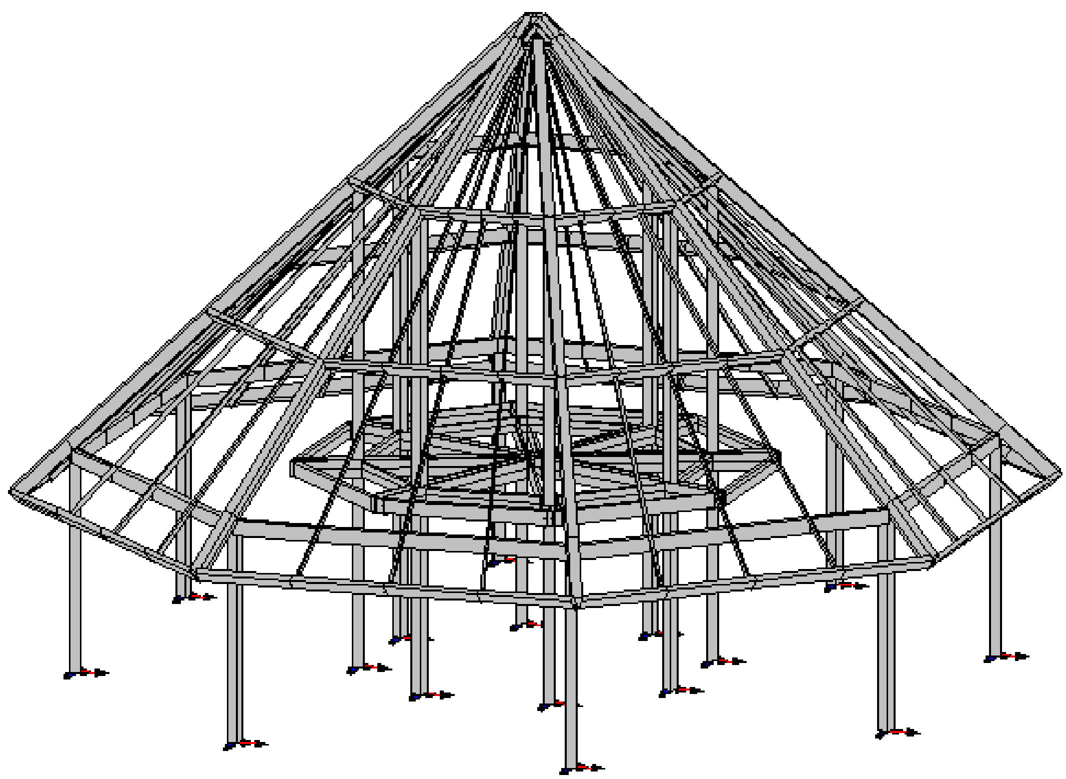
The Timber Design Results Spreadsheet

Open the **Timber Design Results** spreadsheet through **Results / Structure Design / Timber**. To look at results for only one member, double click on it. To look at results for some members only, select them and press the **Properties** icon.

A design brief is available for each member by clicking on icons located at the bottom of the spreadsheet. However, only one design brief at a time can be viewed  and printed .

Design of a Wood Structure

We are going to design a roof composed of sawn timber elements and some glulams. The design and verification will be done according to CAN/CSA-O86.1 Standard.



Modelling

Please refer to the first examples in this tutorial.

Sections and Materials

Design and Verification:

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 process.

New Sections

New sections called MT will be added in the database because the client wants to use particular glue-laminated sections made up of 1½in laminations. The availability of other pre-defined glulam will be modified in the database because we want VisualDesign to choose among the MT sections only during the design process.

- Go to **Common / Shapes** and open the **Rectangular Sections** spreadsheet.

- Insert lines, enter names using the prefix MT and specify dimensions b and d. Other properties will be automatically calculated. At the right end of the spreadsheet, set the composition to "Glulam" and specify the laminations direction and number.

- Uncheck the column "Canada" for glue-laminated sections, except the MT ones.

Note:

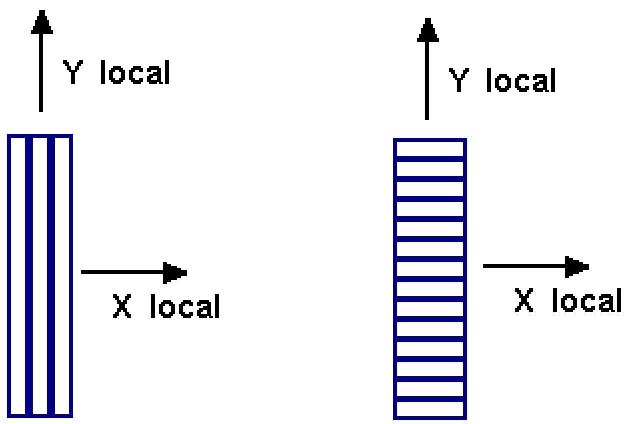
Availability: When the software was installed, we specified Canada as our country, so VisualDesign looks into the "Canada" column to find available sections during a design. In our case, only MT glue-laminated sections will be available for the design.

Rectangular Shapes Spreadsheet

1118	Metric Designation	Imperial Designation	Material	Canada	US	Europe	Personal	d in	b in	Composition	Dir.laminations	Nos. Laminat.
1092	GL275x19[97]	GL275x19[97]	Timber	[ ]	[ ]	[ ]	[ ]	72.56	10.83	Glulam	X Local	97
1093	GL275x19[98]	GL275x19[98]	Timber	[ ]	[ ]	[ ]	[ ]	73.31	10.83	Glulam	X Local	98
1094	MT4½x10	MT4½x10	Timber	[x]	[ ]	[ ]	[ ]	10.00	4.50	Glulam	Y Local	3
1095	MT3x08	MT3x08	Timber	[x]	[ ]	[ ]	[ ]	8.00	3.00	Glulam	Y Local	2
1096	MT7½x7½	MT7½x7½	Timber	[x]	[ ]	[ ]	[ ]	7.50	7.50	Glulam	Y Local	5
1097	MT6x09	MT6x09	Timber	[x]	[ ]	[ ]	[ ]	9.00	6.00	Glulam	Y Local	4
1098	MT4½x08	MT4½x08	Timber	[x]	[ ]	[ ]	[ ]	8.00	4.50	Glulam	Y Local	3
1099	MT1½x11	MT1½x11	Timber	[x]	[ ]	[ ]	[ ]	11.00	1.50	Glulam	X Local	7
1100	MT6x06	MT6x06	Timber	[x]	[ ]	[ ]	[ ]	6.00	6.00	Glulam	Y Local	4
1101	MT1½x09	MT1½x09	Timber	[x]	[ ]	[ ]	[ ]	9.00	1.50	Glulam	X Local	6
1102	MT12x1½	MT12x1½	Timber	[x]	[ ]	[ ]	[ ]	1.50	12.00	Glulam	Y Local	8
1103	MT6x10	MT6x10	Timber	[x]	[ ]	[ ]	[ ]	10.00	6.00	Glulam	Y Local	4
1104	MT4½x4½	MT4½x4½	Timber	[x]	[ ]	[ ]	[ ]	4.50	4.50	Glulam	Y Local	3
1105	MT6x11	MT6x11	Timber	[x]	[ ]	[ ]	[ ]	11.00	6.00	Glulam	Y Local	4
1106	MT3x09	MT3x09	Timber	[x]	[ ]	[ ]	[ ]	9.00	3.00	Glulam	Y Local	2
1107	MT3x11	MT3x11	Timber	[x]	[ ]	[ ]	[ ]	11.00	3.00	Glulam	Y Local	2
1108	MT6x08	MT6x08	Timber	[x]	[ ]	[ ]	[ ]	8.00	6.00	Glulam	Y Local	4
1109	MT1½x08	MT1½x08	Timber	[x]	[ ]	[ ]	[ ]	8.00	1.50	Glulam	X Local	5
1110	MT1½x10	MT1½x10	Timber	[x]	[ ]	[ ]	[ ]	10.00	1.50	Glulam	X Local	7
1111	MT4½x09	MT4½x09	Timber	[x]	[ ]	[ ]	[ ]	9.00	4.50	Glulam	Y Local	3
1112	MT4½x11	MT4½x11	Timber	[x]	[ ]	[ ]	[ ]	11.00	4.50	Glulam	Y Local	3
1113	MT6x12	MT6x12	Timber	[x]	[ ]	[ ]	[ ]	12.00	6.00	Glulam	Y Local	4
1114	MT4½x12	MT4½x12	Timber	[x]	[ ]	[ ]	[ ]	12.00	4.50	Glulam	Y Local	3
1115	MT9x09	MT9x09	Timber	[x]	[ ]	[ ]	[ ]	9.00	9.00	Glulam	Y Local	6
1116	MT3x12	MT3x12	Timber	[x]	[ ]	[ ]	[ ]	12.00	3.00	Glulam	Y Local	2
1117	MT3x10	MT3x10	Timber	[x]	[ ]	[ ]	[ ]	10.00	3.00	Glulam	Y Local	2
1118	MT1½x12	MT1½x12	Timber	[x]	[ ]	[ ]	[ ]	12.00	1.50	Glulam	X Local	8
1119												

Member Local Axes and Laminations

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



Member Properties

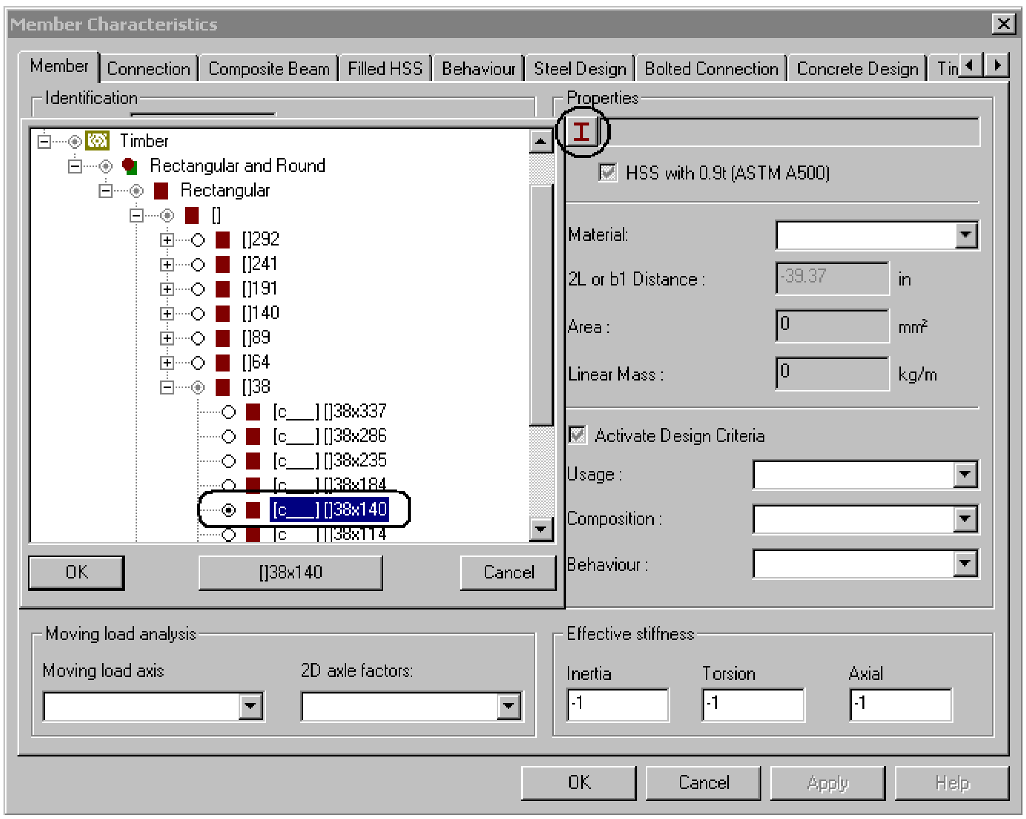
- While in the Structure mode, activate the Member icon on Elements toolbar and select the members for whom you want to assign the same section and material. Click the **Properties** icon to open the **Member Characteristics** dialog box.

Note: To select many elements of the same type, such as members, keep the [Ctrl] key down while clicking on each element.

When many elements are selected, blank fields appear in the dialog box. They mean that the selected data will be assigned to all selected members. Sometimes, values (negative or else) are indicated in blank fields but they are not "real" values. Shaded boxes appear to be activated (checked) but they are not. For example, to activate design criteria for the selected members, the box must be checked.

Top of Roof – Standard sections

- Click the I-Beam icon to open the Shape selection tree. Expand the *Timber* root, *Rectangular* branch, and activate the radio button to select a shape. Click OK to close the selection tree.

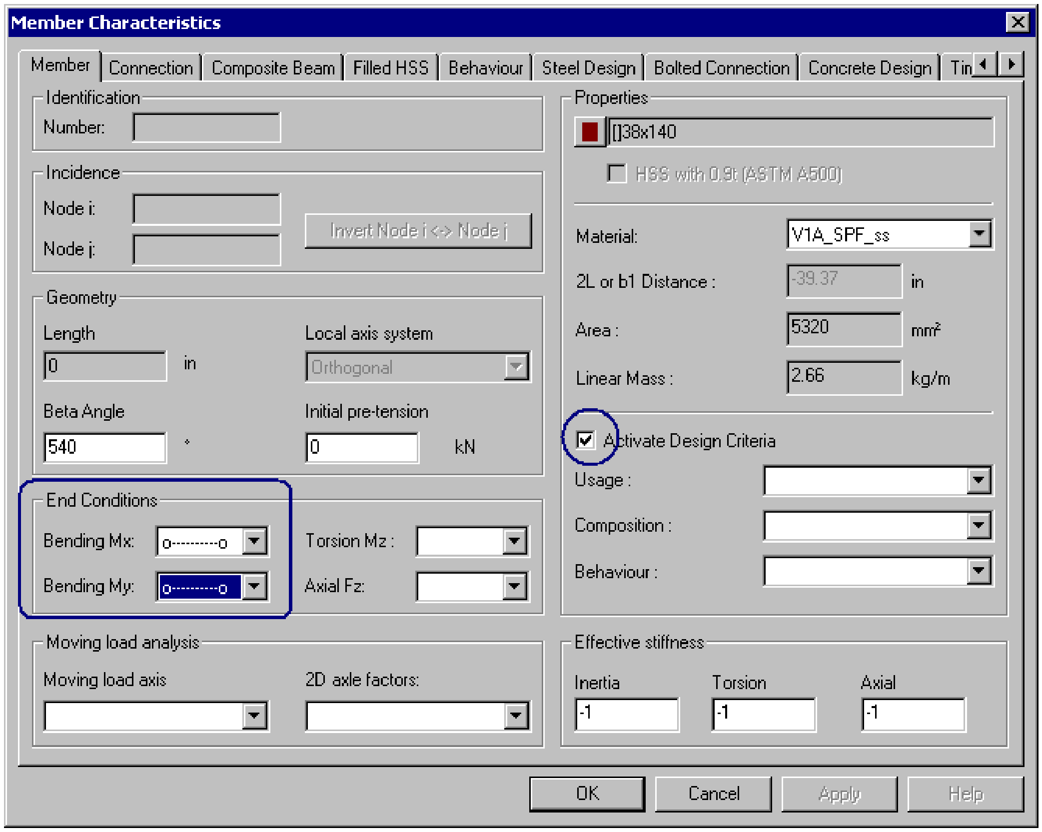


This section is glulam so a glulam material must be assigned.

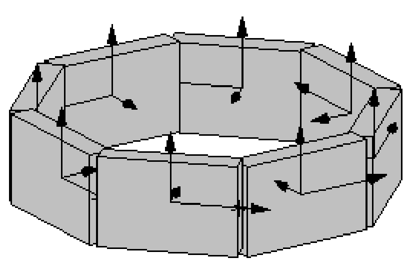
- Go to the "Material" field and click the arrow to open the list box. It is still a selection tree. Expand the *Timber* root, *GLT* and *SPF* branches and double click on material V1A_SPF_ss to select it.

- Activate design criteria.

- Go to the "End Conditions" section and select hinges at node i and j (0-----0) on the member strong and weak axes.

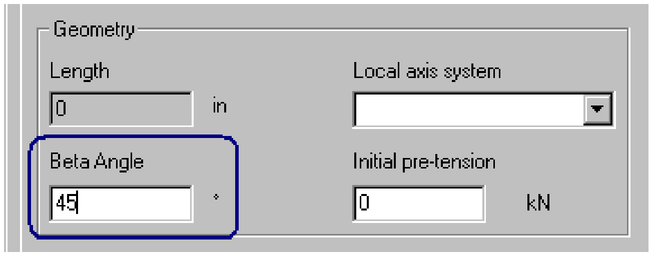


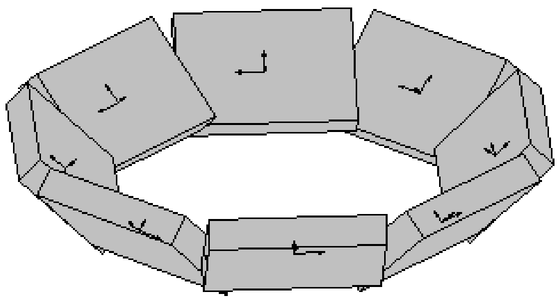
- Click OK.
- Open the **View Options** dialog box. Display the member local axes and activate the *3D Display* option in the **Attributes** tab.



The members are not oriented properly. They are located at the top of the roof, which is sloped at an angle of 45 degrees. We are going to modify the beta angle for these members (the default beta angle is equal to 0).

- Select the members and click the **Properties** icon. Enter a beta angle of 45 degrees. Click OK.





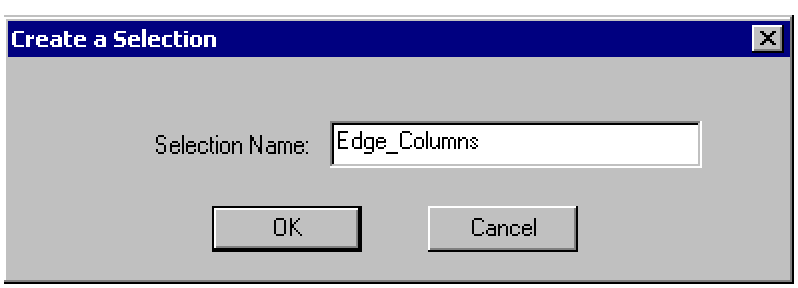
Create Selections

Use the selection functions in **Edit / Select** menu (**Create a selection**, **Update a current selection**, **Choose a selection**, and **Edit selections**) to create your own selections of miscellaneous elements. These selections can be called back anytime for modelling, applying loads, looking at results, etc.

When a selection (or more than one) is activated, use the **Mask** function to mask the rest of the structure.

Example: We want to create a selection of edge columns.

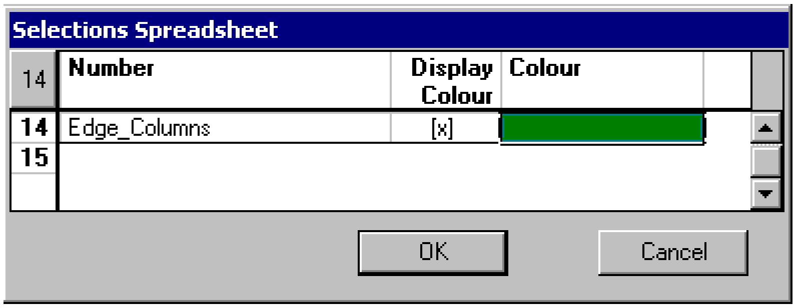
- Activate the Structure mode and the Member icon.
- Select edge columns (click on each column with the cursor while keeping the [Ctrl] key down.
- Go to **Edit / Select / Create a selection**. This dialog box will appear on screen.



- Enter a name and click OK.

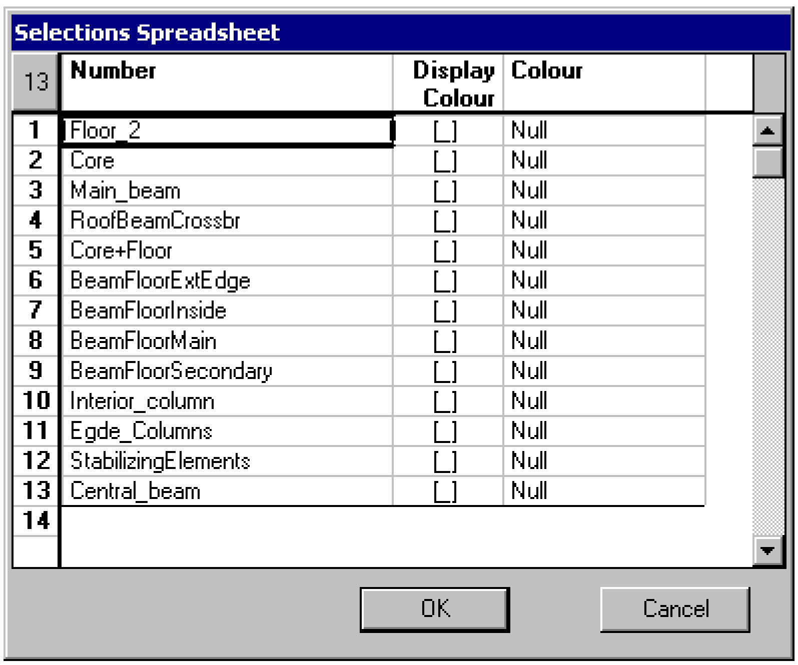
Assign a colour to this selection

- Go to **Edit / Select / Edit selections**. This dialog box will appear on screen.



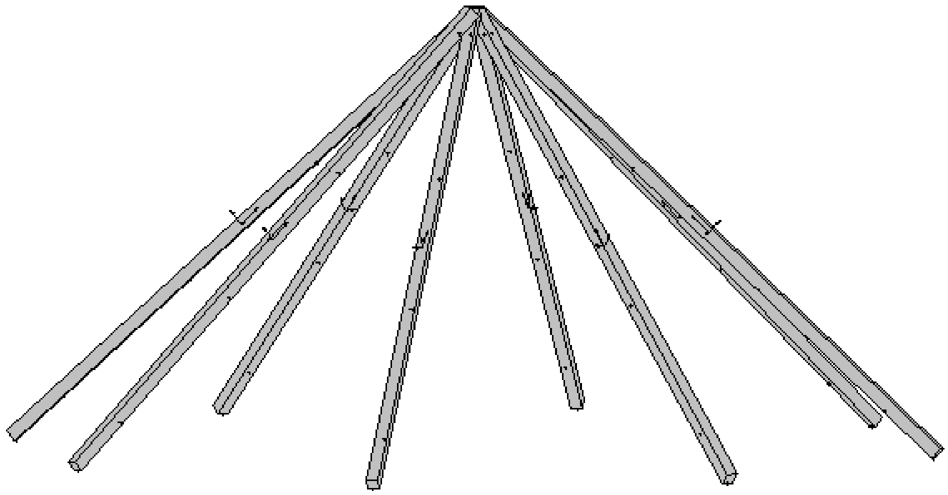
Double-click in the "Colour" column and select a colour among the list box. To see the colour on screen, activate the option [x] by double clicking.

We created the following selections:

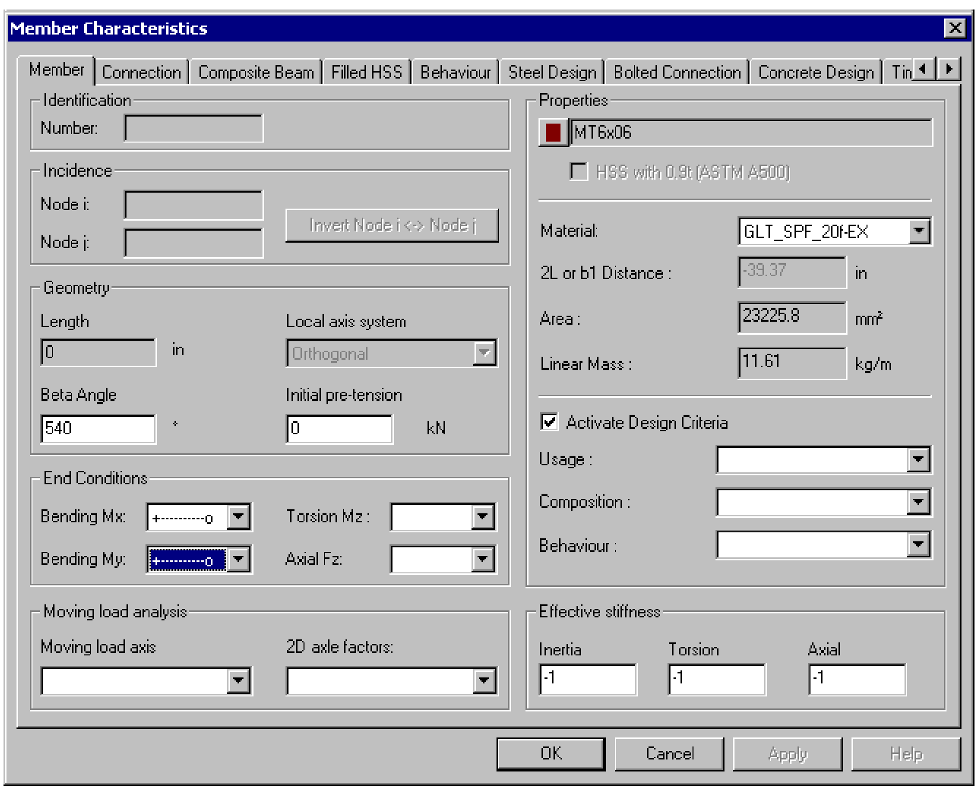


Main Beams – Customized Glulam Sections

Call back the Main_Beam selection and mask the rest of the structure.



- Activate the Structure mode and the Member icon. Select members and click the **Properties** icon.



- Click the I-beam icon to open the Shape selection tree. Select any customized MT section. A design will be done so VisualDesign will select the adequate member among the MT sections.

- Select a GLT material.

- Activate design criteria.

- End conditions: Display the member local axis system to see the position of node i and j. The local z-axis always points towards node j. The member is hinged at the top and fixed at the bottom of the roof.

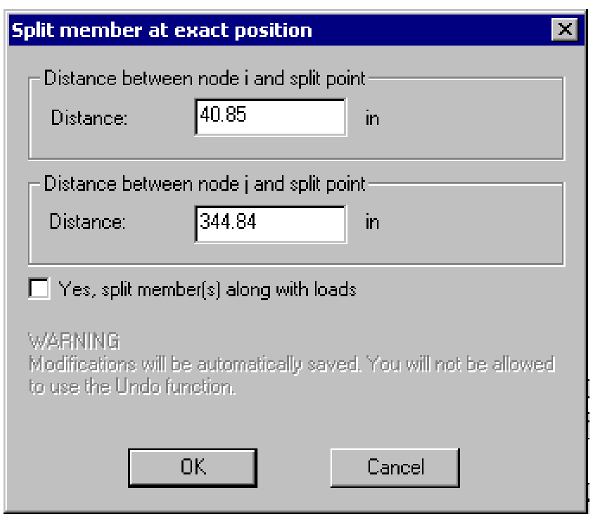
Split members

These members have to be split at the junction of transverse members. Transverse members will be modelled afterwards using these nodes. **Split** functions are available in the **Edit / Split** menu and on the **Split/Join** toolbar.

From the bottom node, which is node i, the first split will be located at 40.85in.

- Select the member that you want to split and got to **Edit / Split / Exact position**.

- Enter 40.85 as the distance between node i and the split point. Click OK.



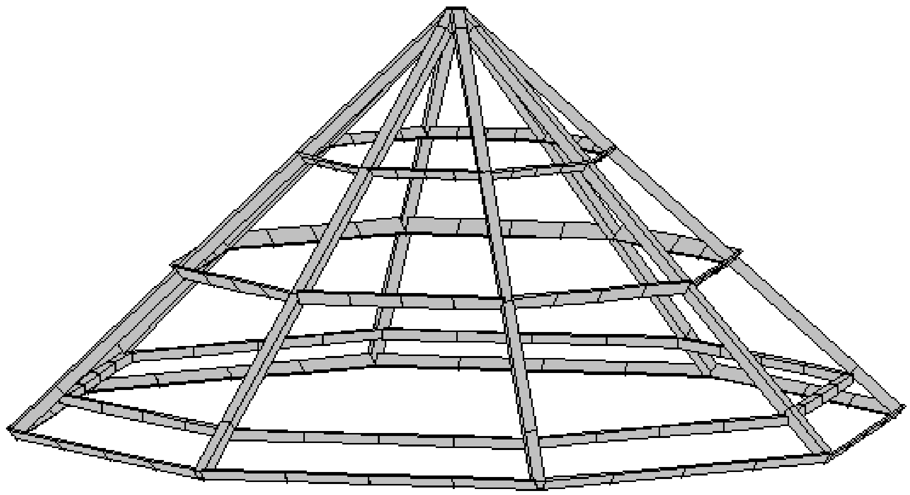
VisualDesign will ask you to merge nearest nodes. Answer Yes.

- Do the same to split the member three other times.

- Do the same for other member or use the **Copy/Paste** function in rotation.

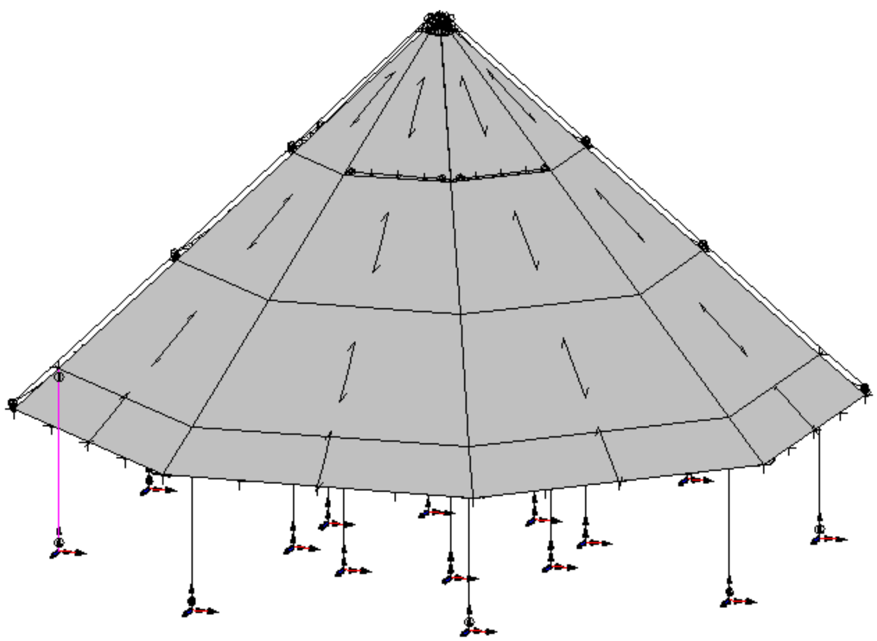
Transverse members

Use the split nodes and model transverse members.



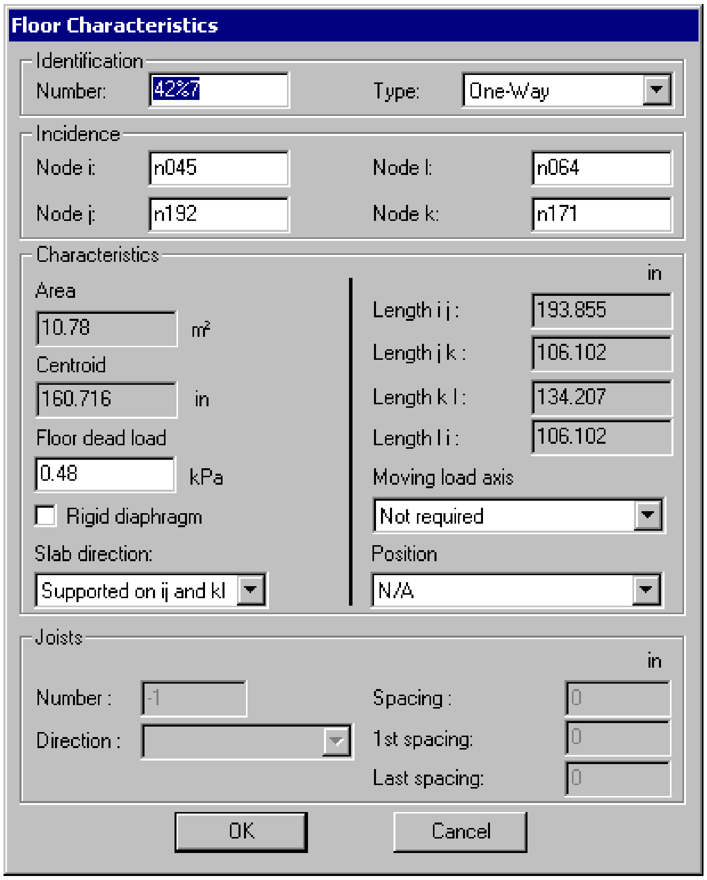
Floors

Floors are modeled to transfer the loads on adjacent members, according to selected tributary areas. Therefore, adjacent beams are required on the floor outline. Sometimes, it is necessary to put "dummy" members, which have a small inertia and a density of zero. One-way floors were specified.



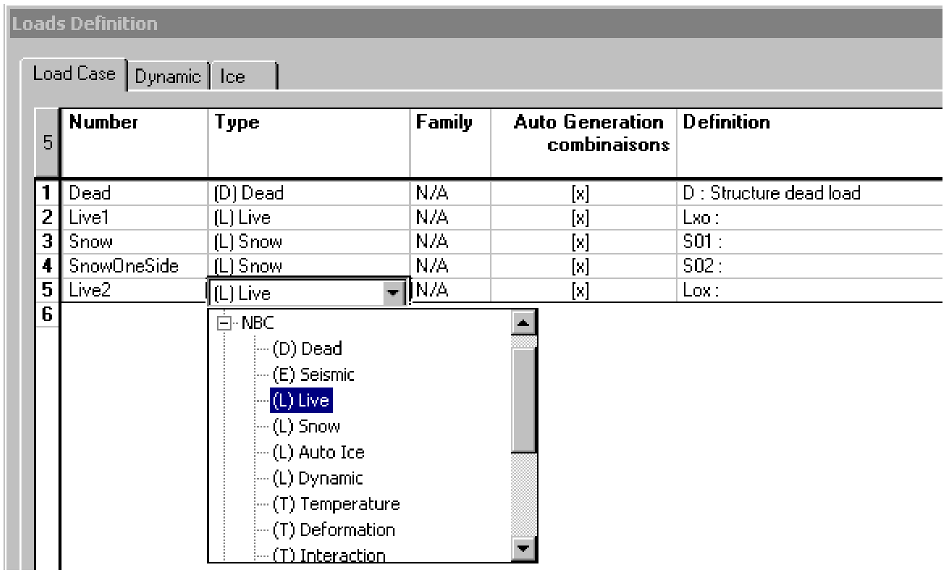
To create floor, activate the Structure mode, the Floor icon and the Add mode. Then, click one each corner node clockwise or counter clockwise. Try to keep in mind that the floor local axes are depending upon incidence nodes. If the first floor is defined in a clockwise direction, do the same for other floors.

If the floor dead load is specified in the **Floor Characteristics** dialog box, it will be automatically considered in the dead load of the structure. If the floor dead load is specified in the **Loads Definition** spreadsheet, it will have to be graphically applied to the structure, as other types of loads.



Loads Definitions

Load case titles and types must be defined in the **Loads Definition** spreadsheet (**Loads / Load Cases / Definition**). Types of loads (wind, dead, live, snow, etc) are important if you plan to generate load combinations with the **Load Combination Generation Wizard**. Double-click in the "Type" column and expand the root that is corresponding to the appropriate building code. Double-click on a type of load.



Note: Some columns are masked in this spreadsheet.

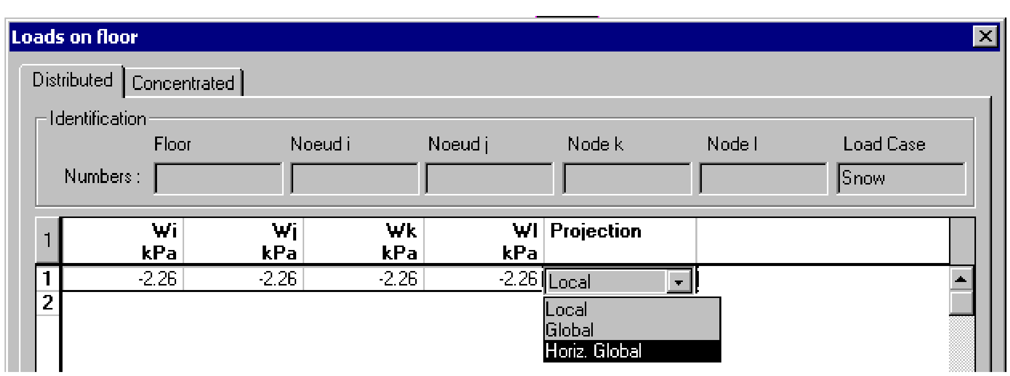
Applying Loads

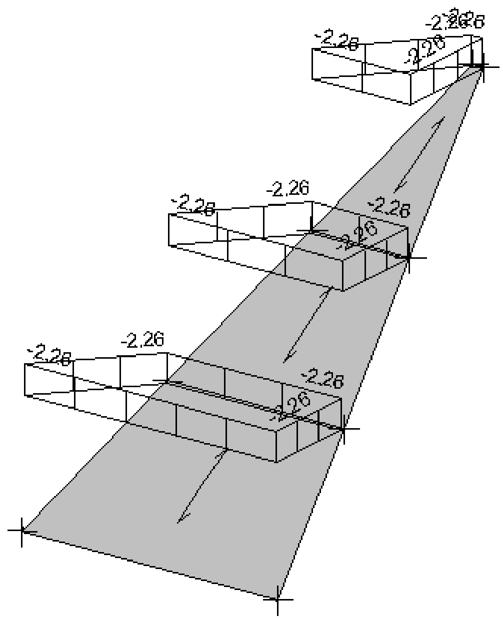
Loads are applied graphically because it is easier this way. Loads spreadsheets are useful to sort data and modify common values using the contextual menu.

Snow Load:

Snow load must be applied using the horizontal global projection.

- Activate the Load case mode and select the snow load case among the Title list box.
- Activate the Floor element and select all floors. Click the **Properties** icon.
- Insert a line in the spreadsheet and enter –2.26 kPa. Double-click in the "Projection" column and select *Horiz. Global*. Click OK.





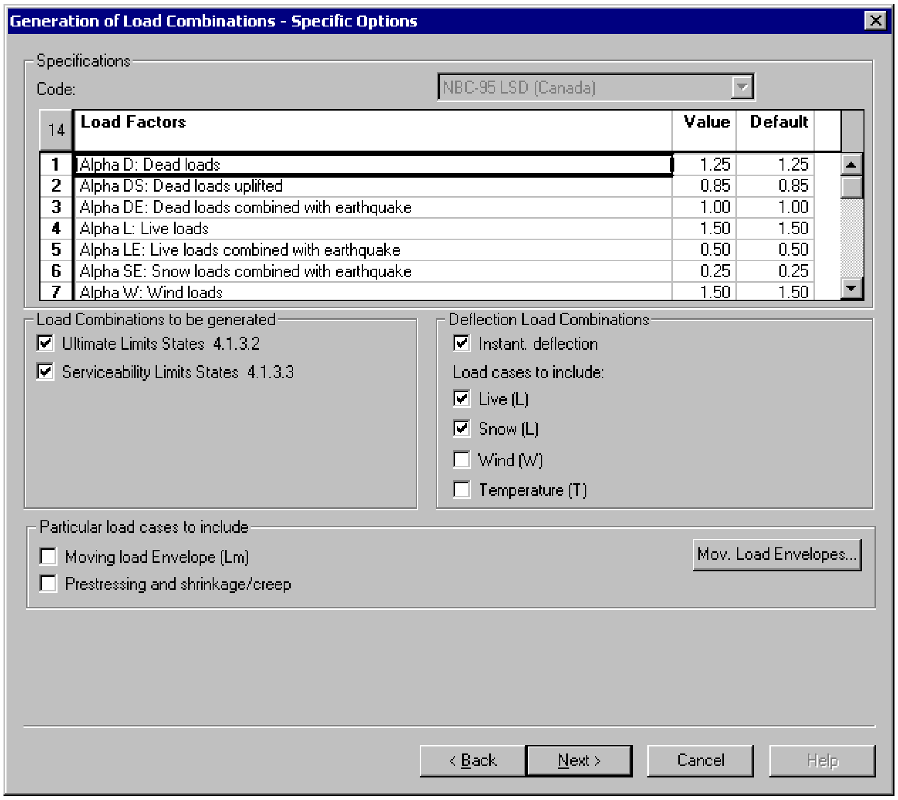
- Load Case: Snow

Follow the same procedure to graphically apply other load cases.

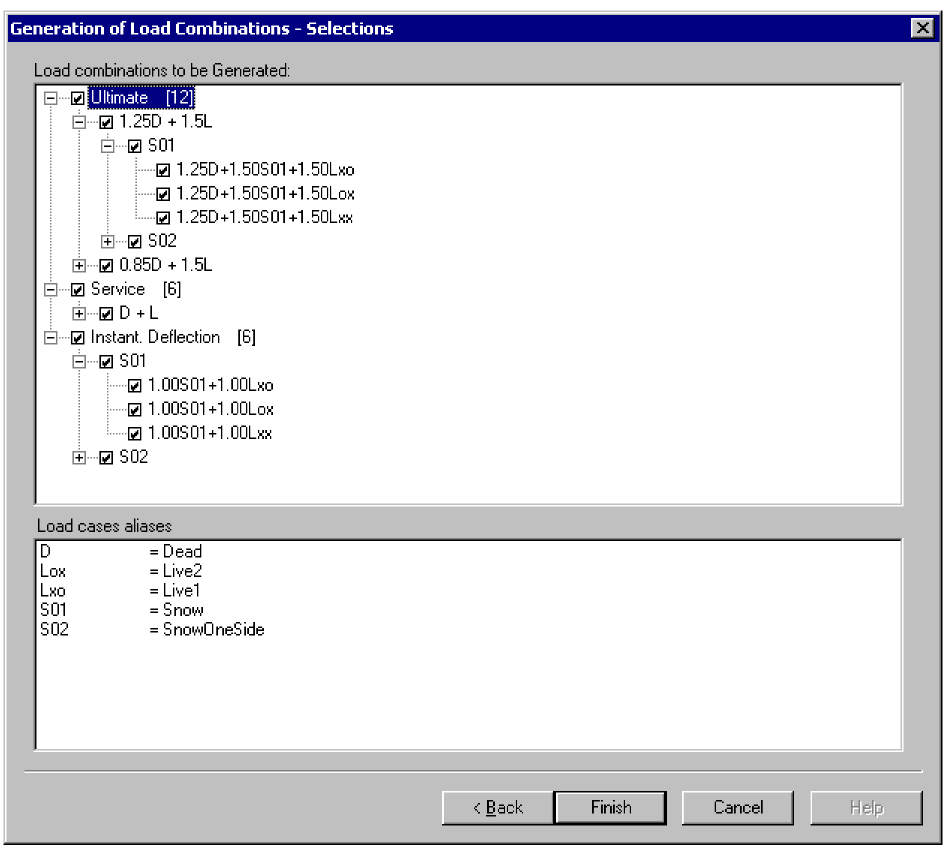
Load Combinations

The **Load Combination Generation Wizard** will be used to automatically generate all the required load combinations according to the CNB-95 code.

- Go to **Loads / Load Combinations / Generation Wizard**.
- Select the NBC-95 Code among the list box. Activate the option that will generate envelopes.
- Click the "Next" button.
- Activate the ultimate and serviceability limits states load combinations. We want the deflections to be considered during the design process so these load combinations are also activated.



- Click the "Next" button.



This last page shows the load combinations that will be generated when the "Finish" button will be clicked. If you do not want to analyse all load combinations, uncheck some of them.

- Click the "Finish" button. The **Load Combinations** spreadsheet will appear on screen.

Load Combinations

Load Combinations | Load Factors

24	Number	Status	Definition	Duration Kd
1	DL01	Ultimate	1.25D+1.50S01+1.50Lxo	1.00
2	DL02	Ultimate	1.25D+1.50S01+1.50Lox	1.00
3	DL03	Ultimate	1.25D+1.50S01+1.50Lxx	1.00
4	DL04	Ultimate	1.25D+1.50S02+1.50Lxo	1.00
5	DL05	Ultimate	1.25D+1.50S02+1.50Lox	1.00
6	DL06	Ultimate	1.25D+1.50S02+1.50Lxx	1.00
7	DL07	Ultimate	0.85D+1.50S01+1.50Lxo	1.00
8	DL08	Ultimate	0.85D+1.50S01+1.50Lox	1.00
9	DL09	Ultimate	0.85D+1.50S01+1.50Lxx	1.00
10	DL10	Ultimate	0.85D+1.50S02+1.50Lxo	1.00
11	DL11	Ultimate	0.85D+1.50S02+1.50Lox	1.00
12	DL12	Ultimate	0.85D+1.50S02+1.50Lxx	1.00
13	DL13	Service	1.00D+1.00S01+1.00Lxo	1.00
14	DL14	Service	1.00D+1.00S01+1.00Lox	1.00
15	DL15	Service	1.00D+1.00S01+1.00Lxx	1.00
16	DL16	Service	1.00D+1.00S02+1.00Lxo	1.00
17	DL17	Service	1.00D+1.00S02+1.00Lox	1.00
18	DL18	Service	1.00D+1.00S02+1.00Lxx	1.00
19	L19	Instant. Deflection	1.00S01+1.00Lxo	1.00
20	L20	Instant. Deflection	1.00S01+1.00Lox	1.00
21	L21	Instant. Deflection	1.00S01+1.00Lxx	1.00
22	L22	Instant. Deflection	1.00S02+1.00Lxo	1.00
23	L23	Instant. Deflection	1.00S02+1.00Lox	1.00
24	L24	Instant. Deflection	1.00S02+1.00Lxx	1.00

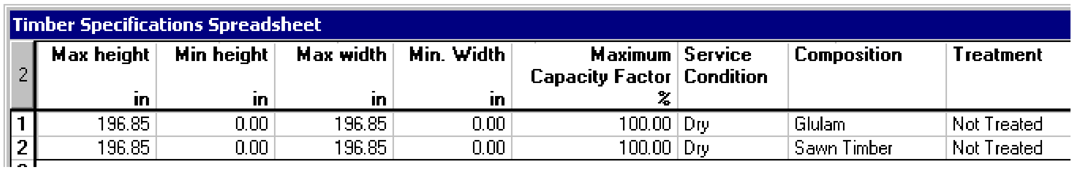
- Close the spreadsheet.

Specifications

- Go to **Structure / Specifications / Timber** to open the spreadsheet. Two specifications are already entered: one for design and the second for verification. In our project, only glulam members will be design so, in the "Composition" column, select *Glulam*. Sawn timber elements will be verified. Service condition is *Dry* and elements are not treated.

Timber Specifications Spreadsheet

5	Number	Code	Type of analysis	Optimization	Interaction	Shape
1	CSA-086-1-Design	CSA-086-01	Design	Area	Standard	Bar (Rect.)
2	CSA-086-1-Verif.	CSA-086-01	Verification	Area	Standard	Bar (Rect.)



Timber Specifications Spreadsheet

2	Max height in	Min height in	Max width in	Min. Width in	Maximum Capacity Factor %	Service Condition	Composition	Treatment
1	196.85	0.00	196.85	0.00	100.00	Dry	Glulam	Not Treated
2	196.85	0.00	196.85	0.00	100.00	Dry	Sawn Timber	Not Treated

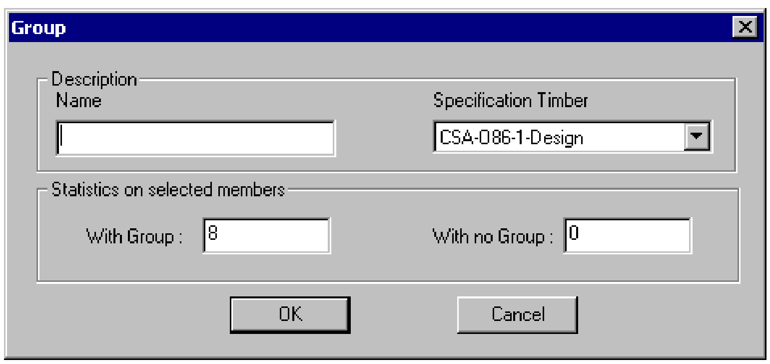
- Click OK.

Design Groups

Use the **Group** function (**Structure / Groups**) or use the short cut keys [Ctrl]+G. Members must be selected before calling this function.

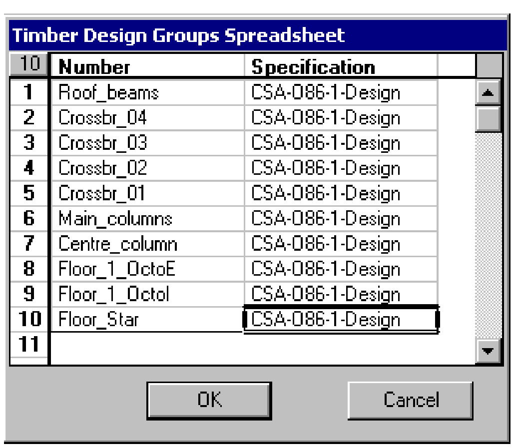
Example:

- Activate the Structure mode and Member element.
- Select columns located within the structure and press the [Ctrl]+G keys.



- Type a name and select a specification for this design group. Click OK.

We created these design groups. VisualDesign will assign one section per design group.



Timber Design Groups Spreadsheet

10	Number	Specification
1	Roof_beams	CSA-086-1-Design
2	Crossbr_04	CSA-086-1-Design
3	Crossbr_03	CSA-086-1-Design
4	Crossbr_02	CSA-086-1-Design
5	Crossbr_01	CSA-086-1-Design
6	Main_columns	CSA-086-1-Design
7	Centre_column	CSA-086-1-Design
8	Floor_1_OctoE	CSA-086-1-Design
9	Floor_1_OctoI	CSA-086-1-Design
10	Floor_Star	CSA-086-1-Design
11		

Displaying a Design Group

- Open the **View Options** dialog box and go to the **Attributes** tab. Activate the option *Design Group* in the Member section and select a group in the drop-down list box. Click OK.

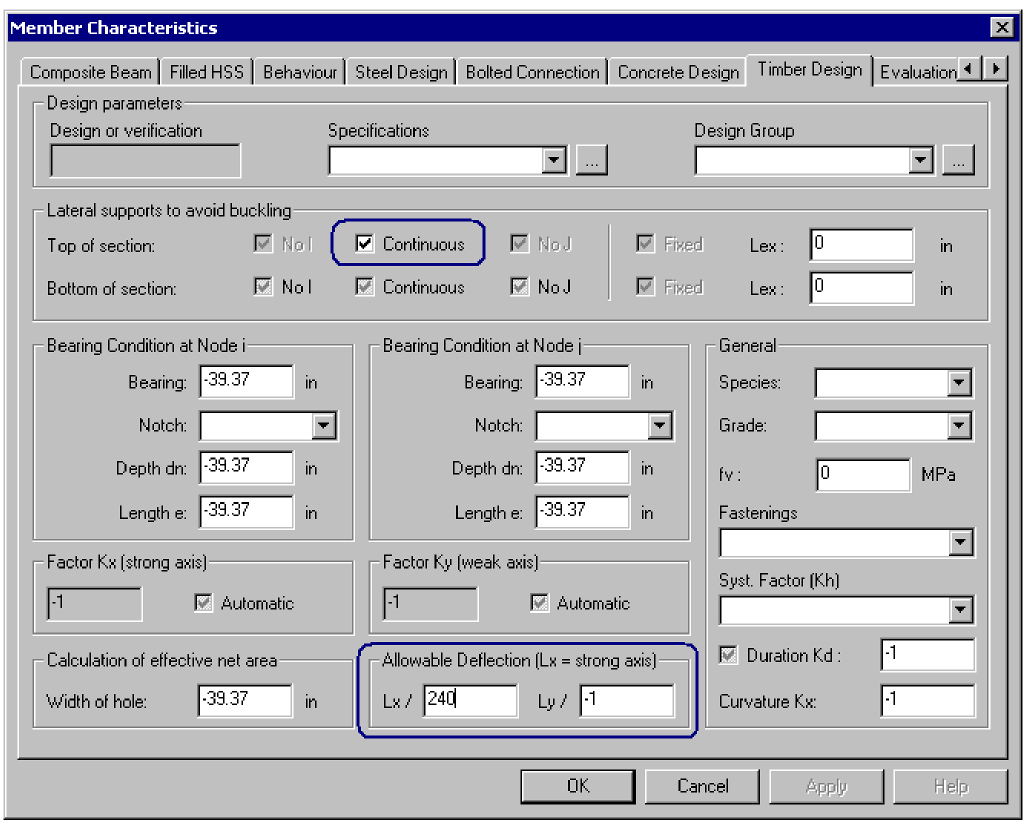
Member Design Criteria

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 in the **Timber Design** tab.

Other design criteria can be specified through the Member's **Timber Design** tab. For example, main beams will be continuously supported with plywood or other sheathing elements. The following procedures must be done:

- Activate the Structure mode and Member element. Call back the "Main_beams" selection and mask the rest of the structure. Click the **Properties** icon.

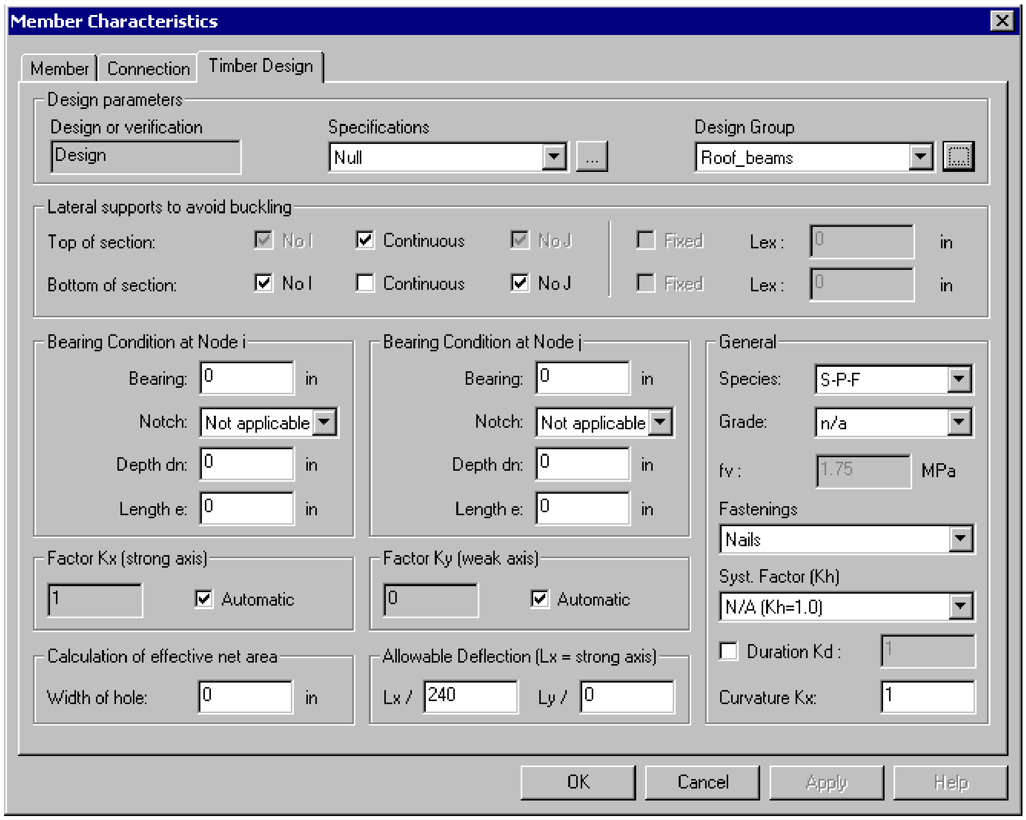
- Specify a continuous lateral support at the top of the section, as shown on the image below, and assign a deflection criterion of Lx/240 on the member strong axis.



- Click OK.

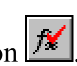
To look at default values in this dialog box, double click on only one member.

Press the **F1** control key to open VisualDesign On-line Help about this dialog box and fields that are composing it.



The effective compression length factors are automatically calculated (default setting) in this project because it is a 3D model (VisualDesign seeks for transverse members along beams and columns).

Analysis and Design

- Open the **Analysis and Design** dialog box by pressing this icon .
- Click on the "Analyse" button to launch the design.
- When the design is completed, close the dialog box.

Results

The *Design Results* icon is automatically activated when the design results are available.

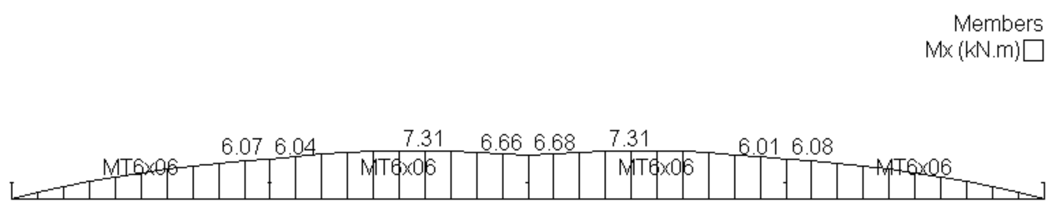
View Options – Coloured Legend for Results

Open the **View Options** dialog box by pressing this icon . Go to the **Results** tab and activate the *Design Load* option to look at coloured members. Each colour corresponds to a percentage range for the calculated member design loads. When a member is at 90% of its capacity, it will be displayed orange. This option is useful to quickly detect insufficient members, which are red.

To modify default colours, go to the **Limits** tab and select other colours for design load intervals.

Forces and Deflection Diagrams

- Activate a load combination or envelope on Activation toolbar.
- Select the **Results** tab of **View Options** dialog box to display graphic results for forces and deflections, support reactions, moments at nodes, shear stresses, axial stresses, torsion and deflection of members by ticking off the boxes. To see numerical values, tick off the "Numerical" box.



- Load Combination : DL04

Note: To get a better view, select members and mask the rest of the structure.

Timber Design Results Spreadsheet

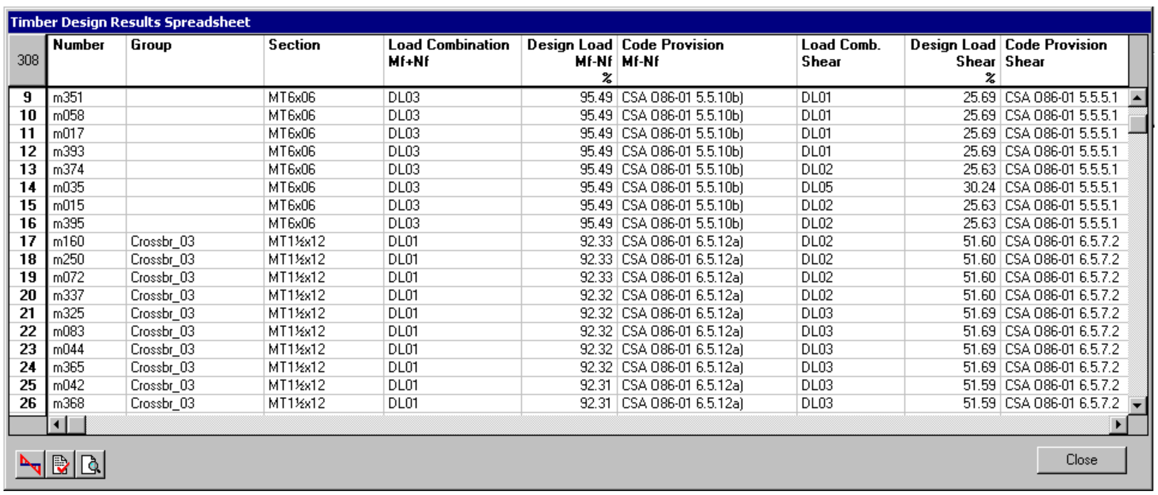
The spreadsheet is available in **Results** / **Structure Design /Timber**. It includes critical load combinations (interaction and shear) and code provisions that controlled the design for each member. It also supplies the member design load, bending, shear and axial resistances, effective compression lengths, etc.

If lines are marked with yellow, it means that the member is not adequate. Look for a red cell to find the problem. Colours don't appear on the printed sheet.

There are many ways to open this spreadsheet:

- If no members were selected, the spreadsheet will include all members;
- Double-click on a member to access the spreadsheet. The spreadsheet will include data for this member only.
- Click on many members while pressing down the [Ctrl] key and click on the **Properties** icon. Only selected members will be part of the results spreadsheet.

In the lower part of this spreadsheet, you will notice three buttons. The first gives access to the member internal forces spreadsheet, the second prints the member design brief, and the third, is a print preview of the design brief.


Timber Design Results Spreadsheet

308	Number	Group	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear	Design Load Shear %	Code Provision Shear
9	m351		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL01	25.69	CSA 086-01 5.5.5.1
10	m058		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL01	25.69	CSA 086-01 5.5.5.1
11	m017		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL01	25.69	CSA 086-01 5.5.5.1
12	m393		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL01	25.69	CSA 086-01 5.5.5.1
13	m374		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL02	25.63	CSA 086-01 5.5.5.1
14	m035		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL05	30.24	CSA 086-01 5.5.5.1
15	m015		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL02	25.63	CSA 086-01 5.5.5.1
16	m395		MT6x06	DL03	95.49	CSA 086-01 5.5.10b)	DL02	25.63	CSA 086-01 5.5.5.1
17	m160	Crossbr_03	MT1½x12	DL01	92.33	CSA 086-01 6.5.12a)	DL02	51.60	CSA 086-01 6.5.7.2
18	m250	Crossbr_03	MT1½x12	DL01	92.33	CSA 086-01 6.5.12a)	DL02	51.60	CSA 086-01 6.5.7.2
19	m072	Crossbr_03	MT1½x12	DL01	92.33	CSA 086-01 6.5.12a)	DL02	51.60	CSA 086-01 6.5.7.2
20	m337	Crossbr_03	MT1½x12	DL01	92.32	CSA 086-01 6.5.12a)	DL02	51.60	CSA 086-01 6.5.7.2
21	m325	Crossbr_03	MT1½x12	DL01	92.32	CSA 086-01 6.5.12a)	DL03	51.69	CSA 086-01 6.5.7.2
22	m083	Crossbr_03	MT1½x12	DL01	92.32	CSA 086-01 6.5.12a)	DL03	51.69	CSA 086-01 6.5.7.2
23	m044	Crossbr_03	MT1½x12	DL01	92.32	CSA 086-01 6.5.12a)	DL03	51.69	CSA 086-01 6.5.7.2
24	m365	Crossbr_03	MT1½x12	DL01	92.32	CSA 086-01 6.5.12a)	DL03	51.69	CSA 086-01 6.5.7.2
25	m042	Crossbr_03	MT1½x12	DL01	92.31	CSA 086-01 6.5.12a)	DL03	51.59	CSA 086-01 6.5.7.2
26	m368	Crossbr_03	MT1½x12	DL01	92.31	CSA 086-01 6.5.12a)	DL03	51.59	CSA 086-01 6.5.7.2

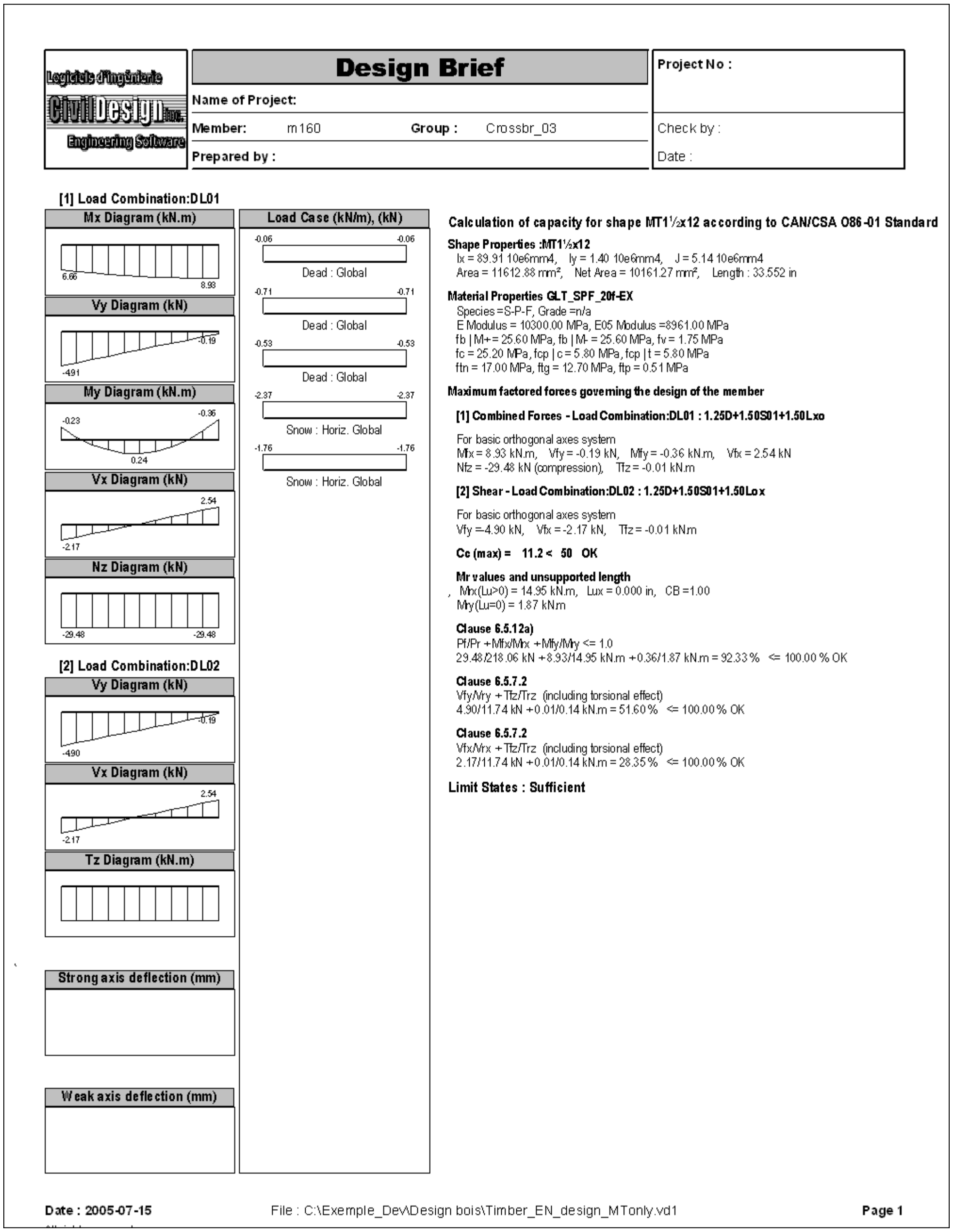
Close

Design Brief

Display the member design brief by selecting (highlight) appropriate line in the **Timber Design Results** spreadsheet.

Click the "Print Preview" icon to have a look at the member design brief. To print it, use the "Print" button available in the Print Preview menu.

Click the "Design Brief" icon to directly print it.



Design Brief

Legiciels d'Ingénierie CivilDesign Inc. Engineering Software

Name of Project:			**Project No :**
Member: m160	**Group :** Crossbr_03		Check by :
Prepared by :			Date :

[1] Load Combination:DL01

Mx Diagram (kN.m)
6.66 8.93

Vy Diagram (kN)
-0.19 -4.91

My Diagram (kN.m)
-0.23 -0.36 0.24

Vx Diagram (kN)
2.54 -2.17

Nz Diagram (kN)
-29.48 -29.48

[2] Load Combination:DL02

Vy Diagram (kN)
-0.19 -4.90

Vx Diagram (kN)
2.54 -2.17

Tz Diagram (kN.m)

Strong axis deflection (mm)

Weak axis deflection (mm)

Load Case (kN/m), (kN)

-0.06 -0.06
Dead : Global

-0.71 -0.71
Dead : Global

-0.53 -0.53
Dead : Global

-2.37 -2.37
Snow : Horiz. Global

-1.76 -1.76
Snow : Horiz. Global

Calculation of capacity for shape MT1½x12 according to CAN/CSA O86-01 Standard

Shape Properties :MT1½x12
Ix = 89.91 10e6mm4, Iy = 1.40 10e6mm4, J = 5.14 10e6mm4
Area = 11612.88 mm², Net Area = 10161.27 mm², Length : 33.552 in

Material Properties GLT_SPF_20f-EX
Species =S-P-F, Grade =n/a
E Modulus = 10300.00 MPa, E05 Modulus =8961.00 MPa
fb | M+= 25.60 MPa, fb | M- = 25.60 MPa, fv = 1.75 MPa
fc = 25.20 MPa, fcp | c = 5.80 MPa, fcp | t = 5.80 MPa
ftn = 17.00 MPa, ftg = 12.70 MPa, ftp = 0.51 MPa

Maximum factored forces governing the design of the member

[1] Combined Forces - Load Combination:DL01 : 1.25D+1.50S01+1.50Lxo

For basic orthogonal axes system
Mfx = 8.93 kN.m, Vfy = -0.19 kN, Mfy = -0.36 kN.m, Vfx = 2.54 kN
Nfz = -29.48 kN (compression), Tfz = -0.01 kN.m

[2] Shear - Load Combination:DL02 : 1.25D+1.50S01+1.50Lox

For basic orthogonal axes system
Vfy =-4.90 kN, Vfx = -2.17 kN, Tfz = -0.01 kN.m

Cc (max) = 11.2 < 50 OK

Mr values and unsupported length
, Mrx(Lu>0) = 14.95 kN.m, Lux = 0.000 in, CB =1.00
Mry(Lu=0) = 1.87 kN.m

Clause 6.5.12a)
Pf/Pr +Mfx/Mrx +Mfy/Mry <= 1.0
29.48/218.06 kN +8.93/14.95 kN.m +0.36/1.87 kN.m = 92.33 % <= 100.00 % OK

Clause 6.5.7.2
Vfy/Vry + Tfz/Trz (including torsional effect)
4.90/11.74 kN +0.01/0.14 kN.m = 51.60 % <= 100.00 % OK

Clause 6.5.7.2
Vfx/Vrx + Tfz/Trz (including torsional effect)
2.17/11.74 kN +0.01/0.14 kN.m = 28.35 % <= 100.00 % OK

Limit States : Sufficient

Date : 2005-07-15 File : C:\Exemple_Dev\Design bois\Timber_EN_design_MTonly.vd1 **Page 1**

E X A M P L E 7

Modal & Spectral Analyses and Ductile Steel Design

Modal & Spectral Analyses

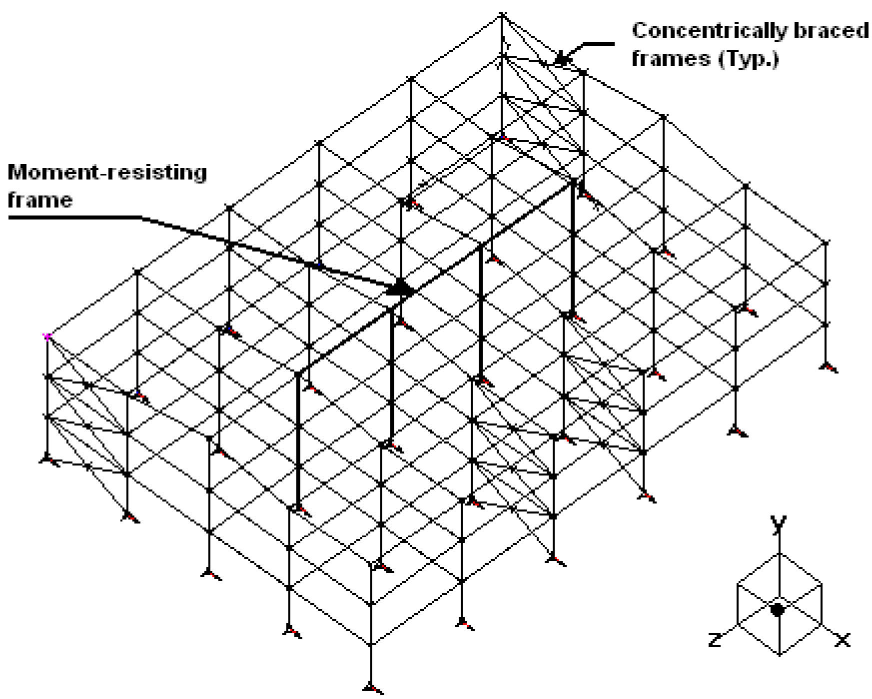
We are going to run a modal and spectral analysis according to the Canadian National Building Code 2005, for a steel building located in Montreal, and shown below.

The lateral force resisting-systems are:

Direction-x: Concentrically braced frames in tension only, type LD (R=2).

Direction-z: Moment-resisting frame located on the middle axis, type LD (R=2).

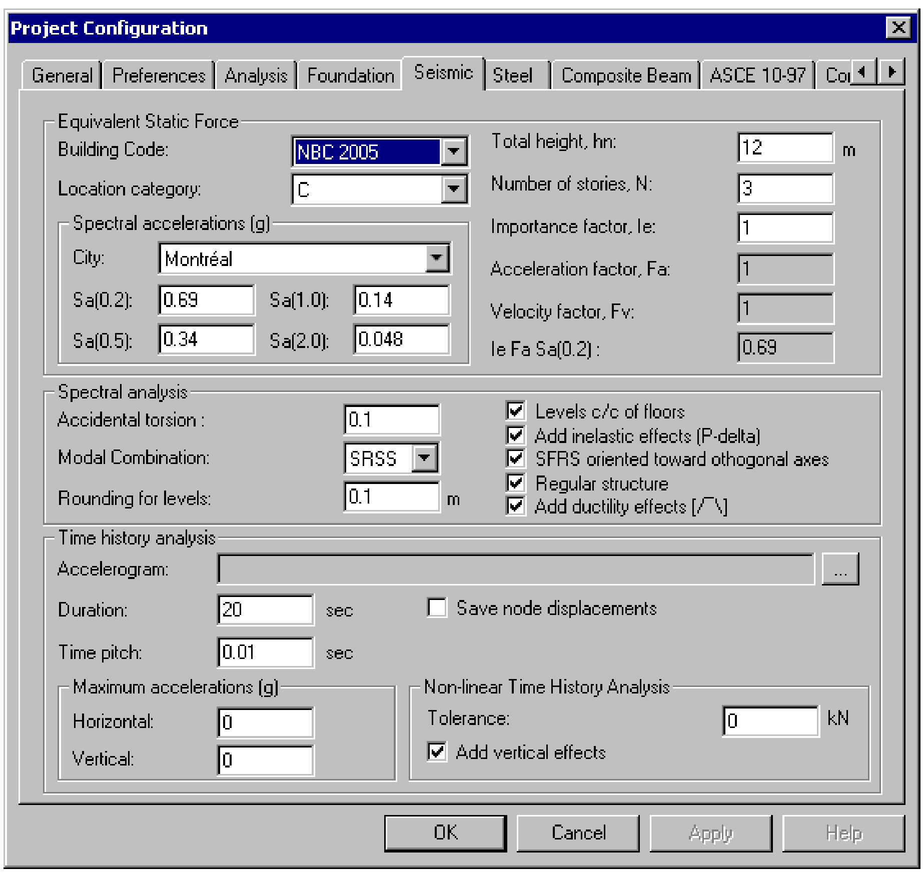
Supports' degrees of freedom in rotation are free.



Project Configuration

Seismic Tab

- Select the **Seismic** tab of **Project Configuration** dialog box (**File** menu).
- Enter the following data: Building Code: CNB-2005, Location category = C, City of Montreal, Total height = 12m, Number of stories (N) = 3, and Importance factor of 1.0.
- The SRSS (*Square-Root-of-Sum-of-Squares*) method will be used for modal combination.
- Activate the option "Add ductility effects [/‾\]" to design the ductile steel frames according to section 27 of S16-01 Standard.

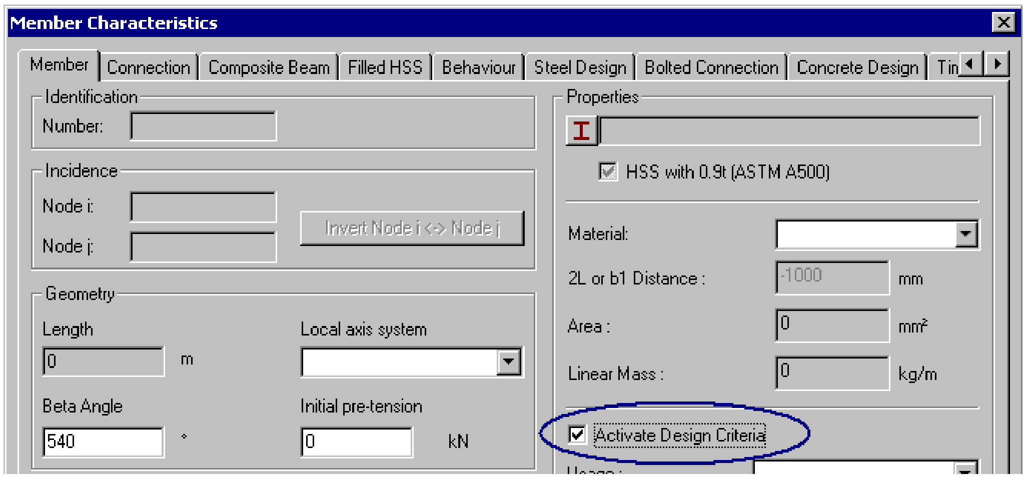


- Click OK.

Design Criteria and Specifications

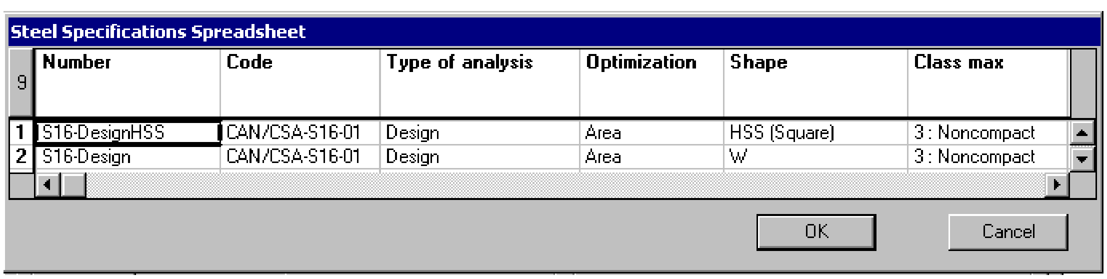
Member Tab

Select all members and activate the design criteria in the **Member** tab of **Member Characteristics** dialog box.



Specifications

Open the **Steel Specifications** spreadsheet and add one specification for tension-only bracings. Square HSS will be used for these members.


Steel Specifications Spreadsheet

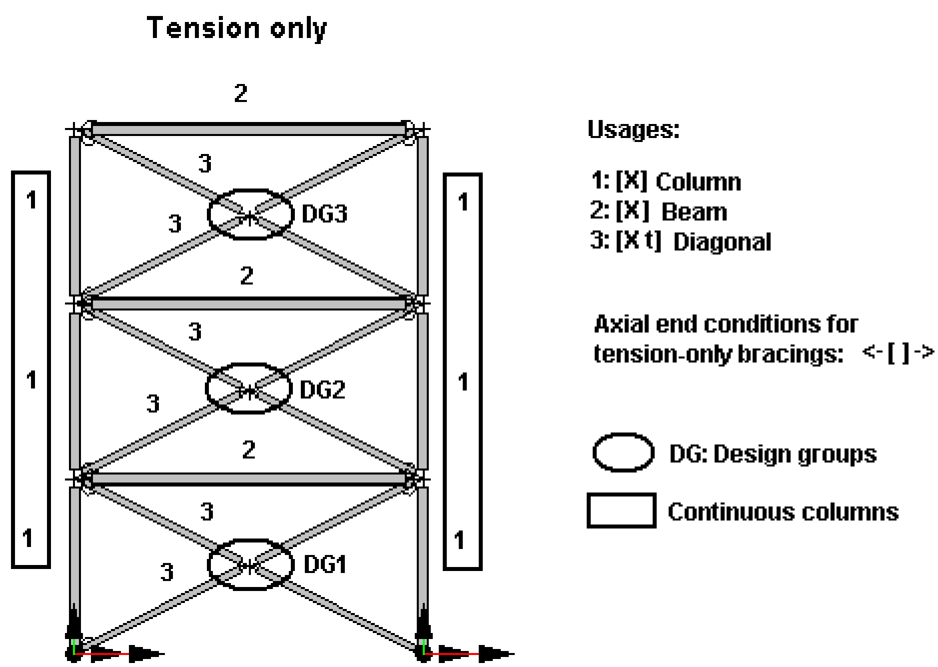
9	Number	Code	Type of analysis	Optimization	Shape	Class max
1	S16-DesignHSS	CAN/CSA-S16-01	Design	Area	HSS (Square)	3 : Noncompact
2	S16-Design	CAN/CSA-S16-01	Design	Area	W	3 : Noncompact

Do not bother with the class of the steel shape that will be design as a ductile element. The software will automatically choose a class 1 or 2, as required per S16-01 Standard.

Ductile Frames

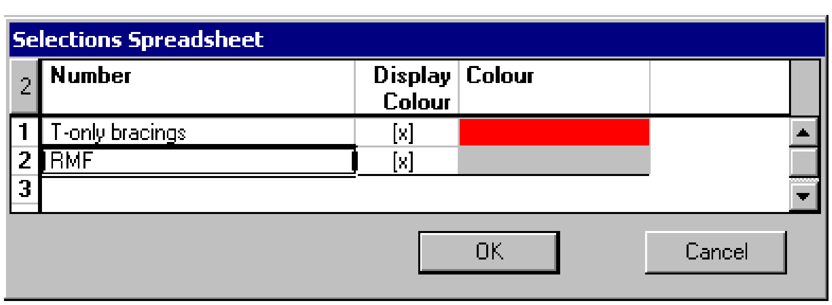
Concentrically Braces Frames in Tension-only – Limited Ductility (R=2)

The following member usages must be specified in the **Member** tab along with axial end conditions for tension-only bracing.



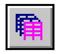
To help you while modelling the ductile frames, create selections for each type of frame. Selection functions (**Create selections**, **Choose selections**, and **Edit Sections**) are available in the **Edit / Select** menu. Learn more about these functions in VisualDesign On-line Help.

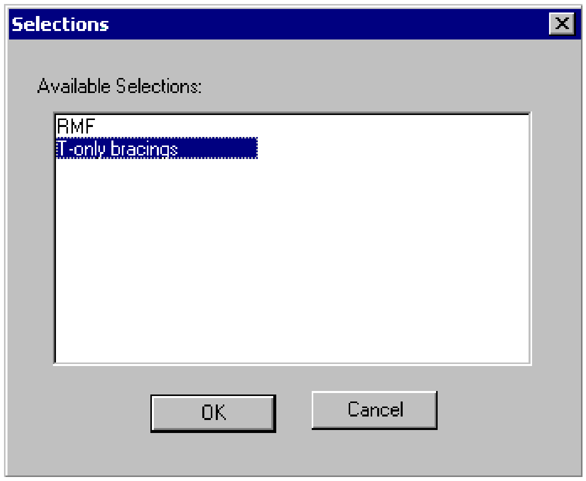
We created two selections and assigned a colour for each one:


Selections Spreadsheet

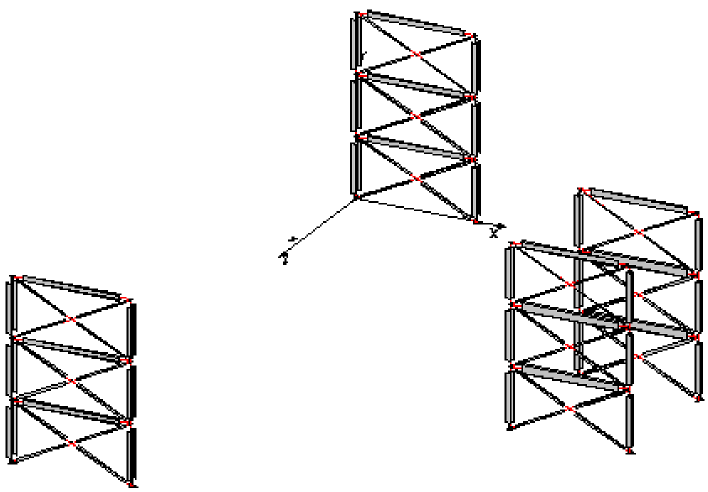
2	Number	Display Colour	Colour
1	T-only bracings	[x]	
2	RMF	[x]	
3			

OK Cancel

- Cick the icon  on Edit toolbar to open the **Selections** dialog box. Highlight the "T-only bracings" selection and click OK.

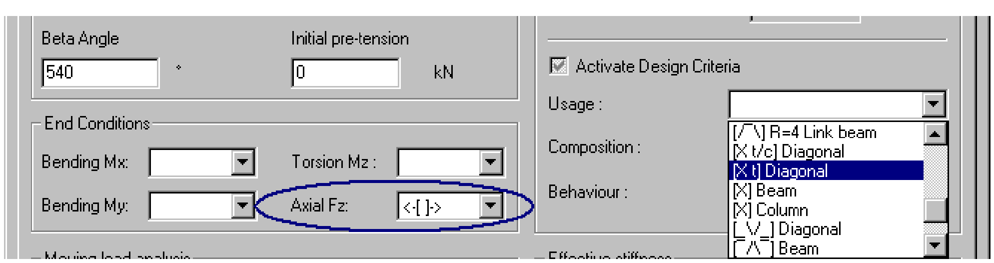


- Mask the rest of the structure by clicking this icon  on View toolbar.



- To quickly select the bracings, go to **Edit / Select / Sloped members**. Click the **Properties**  button.

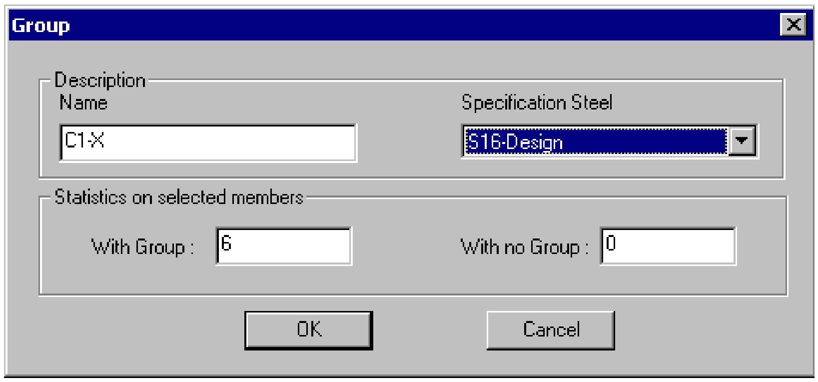
- In the **Member** tab, specify tension-only <-[ ]-> as axial end conditions and select the member usage "[Xt] Diagonal" in the Usage list box. Click OK.



- In the same manner, assign the usage "[X] Column" for columns composing these frames, and the usage "[X] Beam" for beams.

- Columns located on both sides of the bracings must be continuous. Therefore, 4 design groups are required, one per X frame.

- To create a design group, select columns on one X-frame and press the short cut keys [Ctrl]+G. Enter a name for this group and select a specification. Click OK.

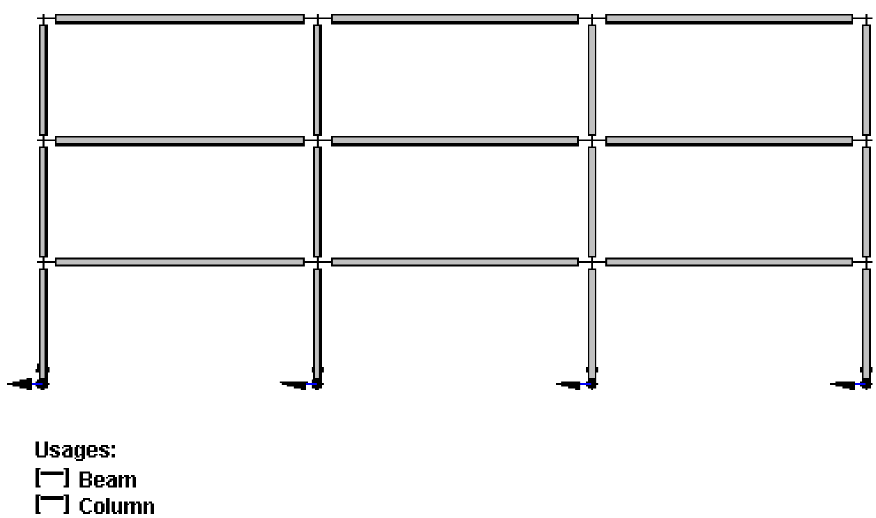


- Do the same for the columns composing other X-frames.

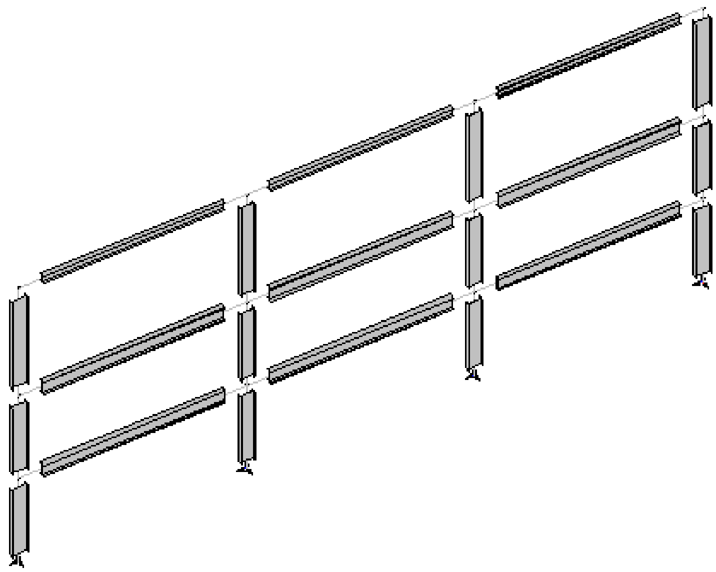
- Create design groups for bracings located on each level and each frame and assign the HSS steel specification.

Resisting-Moment Frame with a Limited Ductility (R=2)

There are two member usages for this type of ductile frame: [ ] Beam and [ ] Columns.

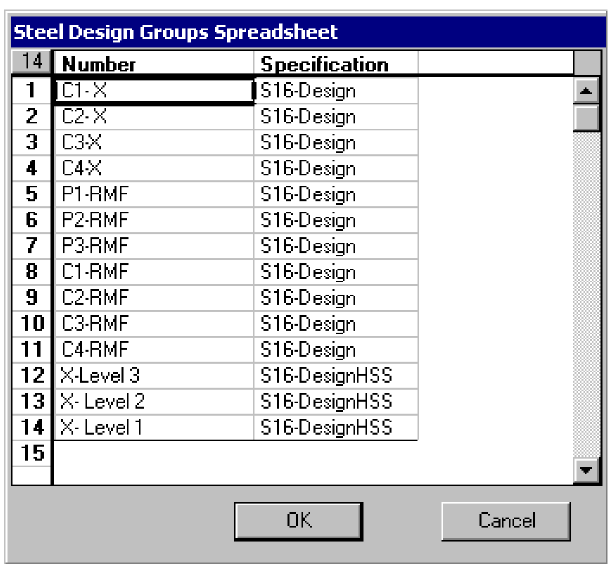


Call back the **Selections** dialog box and highlight the "MRF" selection. Mask the rest of the structure.



Columns and beams must be continuous. Design groups are created for each column and for continuous beams at each level.

The following design groups were created. The spreadsheet is available in **Structure / Groups / Steel members.**



Steel Design Groups Spreadsheet

14	Number	Specification
1	C1-X	S16-Design
2	C2-X	S16-Design
3	C3-X	S16-Design
4	C4-X	S16-Design
5	P1-RMF	S16-Design
6	P2-RMF	S16-Design
7	P3-RMF	S16-Design
8	C1-RMF	S16-Design
9	C2-RMF	S16-Design
10	C3-RMF	S16-Design
11	C4-RMF	S16-Design
12	X-Level 3	S16-DesignHSS
13	X- Level 2	S16-DesignHSS
14	X- Level 1	S16-DesignHSS
15		

OK Cancel

Load Cases

The following load case titles and types are defined in the **Loads Definition** spreadsheet:

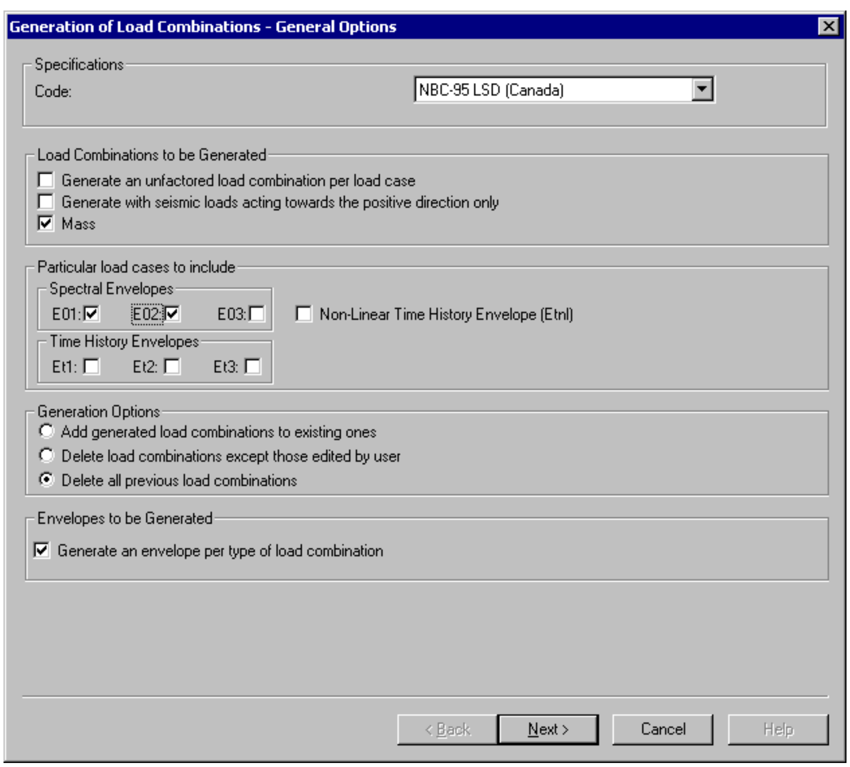
Loads Definition

Load Case | Dynamic | Ice

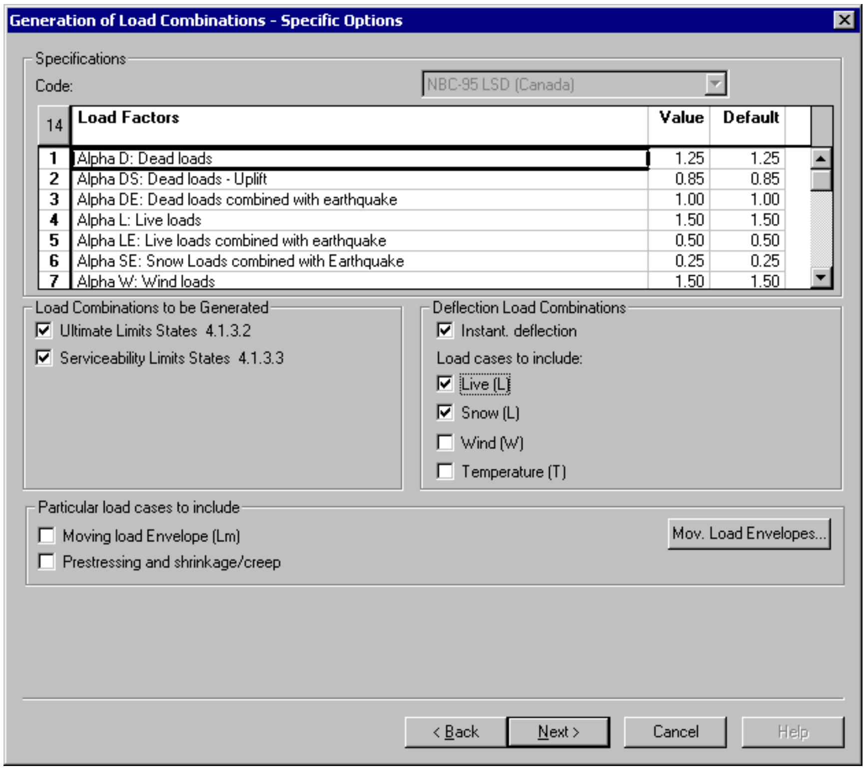
5	Number	Type	Family	Tributary Area Reduction	Tributary Area Overload kPa	Auto Generation combinaisons
1	Dead	(D) Dead	N/A	None	0.00	[x]
2	S1	(L) Snow	N/A	None	0.00	[x]
3	L1	(L) Live	N/A	None	0.00	[x]
4	D1_Roof	(D) Dead	N/A	None	0.00	[x]
5	D2_Floors	(D) Dead	N/A	None	0.00	[x]

Load Combination Generation Wizard

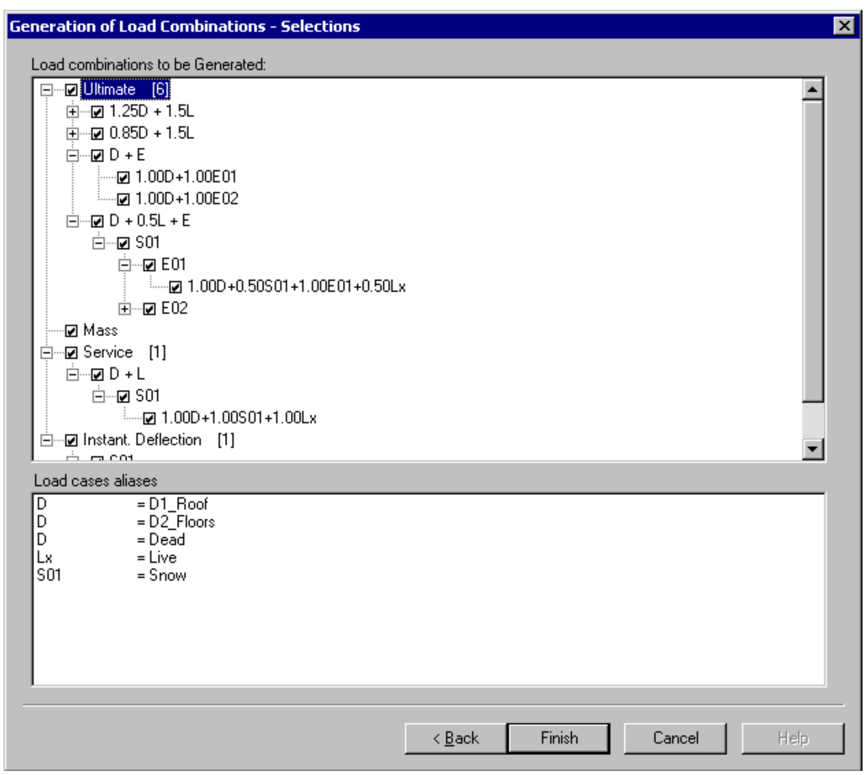
The **Load Combination Generation Wizard** will be used to generate ultimate and service load combinations according to NBC. Spectral envelopes E01 and E02 will also be included. Therefore, the "Mass" load combination will be generated by default. (According to CNB Code, this load combination must include all dead loads plus 25% of snow loads.)



- Click the "Next" button.

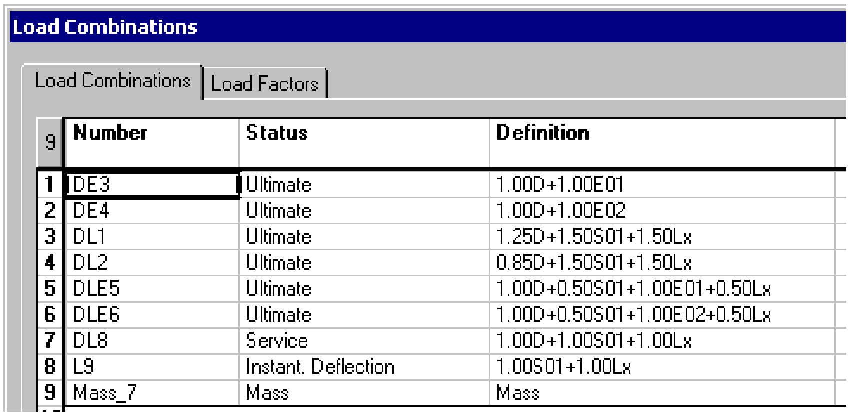


- Click the "Next" button.



- Click the "Finish" button.

The **Wizard** generated these load combinations and corresponding load factors:



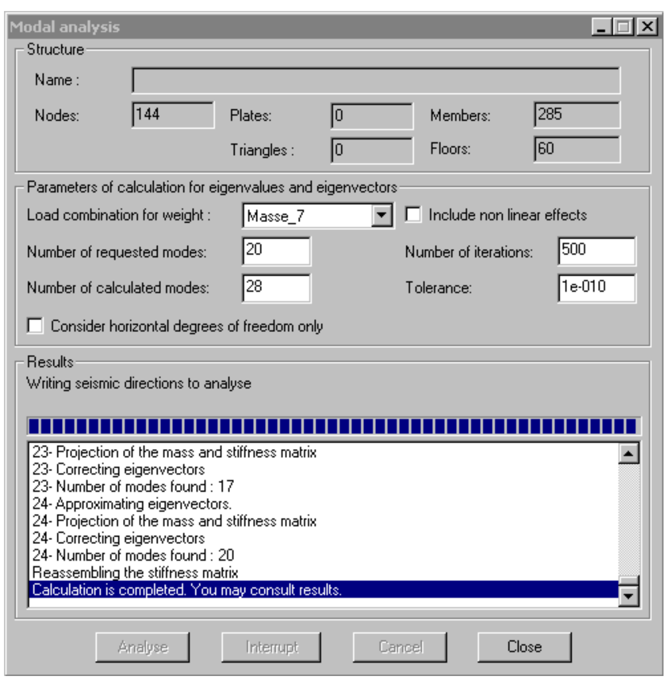
9	Number	Status	Definition
1	DE3	Ultimate	1.00D+1.00E01
2	DE4	Ultimate	1.00D+1.00E02
3	DL1	Ultimate	1.25D+1.50S01+1.50Lx
4	DL2	Ultimate	0.85D+1.50S01+1.50Lx
5	DLE5	Ultimate	1.00D+0.50S01+1.00E01+0.50Lx
6	DLE6	Ultimate	1.00D+0.50S01+1.00E02+0.50Lx
7	DL8	Service	1.00D+1.00S01+1.00Lx
8	L9	Instant. Deflection	1.00S01+1.00Lx
9	Mass_7	Mass	Mass

You are ready to run a modal analysis to obtain the structure vibration modes and corresponding frequencies.

Modal Analysis

- Click the **Modal Analysis** icon .

- In the **Modal Analysis** dialog box, select the "Mass" load combination. We specified 20 as the number of requested modes. Launch the modal analysis by clicking the "Analyse" button.



Note The CNBC suggests 2 to 3 vibration modes per building story for a spectral analysis. Two orthogonal directions must be studied.

A default value for the number of calculated modes is automatically initialized. This value represents the number of modes that have to be calculated in order to get the first *n* desired modes (*Subspace Iteration Method*). For more detail, refer to the topic "Number of Calculated Modes" in On-Line Help *Chapter 6*.

Tolerance and number of iterations are generally satisfying.

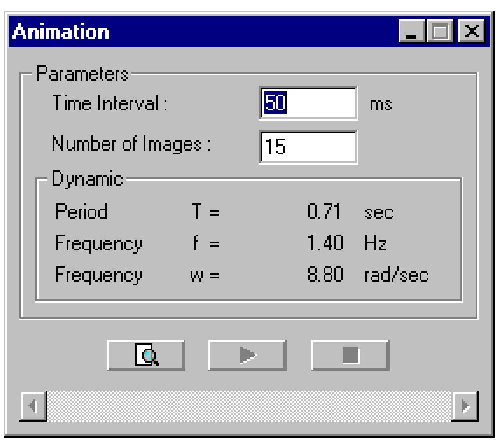
- When analysis is completed, close the dialog box.

Modal Analysis Results

Animation of a Vibration Mode

When the modal analysis is completed, the Vibration Activation mode is automatically activated. Use the **Animation** function to visualize the movement of the building under a selected vibration mode.

- Select a vibration mode in the drop-down list box of Activation toolbar.
- Select an XY, YZ or isometric view of the structure to have a better look at the structure. Use the **Camera** function or use the control key [Pg Dn], [Home], or [Pg Up].
- Click on the **Animation** icon of View toolbar.
- In the **Animation** dialog box, press the icon to capture images.



- If the diagram amplitude is too small, click on the **Increase Amplitude** icon of **Diagrams** toolbar. Click as many times as necessary. Use the **Animation** function again.

Frequencies and Vibration Modes

- Go to **Results** / **Modal & Spectral** and select the **Frequencies and Vibration Modes** spreadsheet.

In this spreadsheet, we can see that mode 1 and 2 are acting in the main directions z and x because the corresponding modal contributions (γz and γx) are the highest. A value of 1.0 for components *Dir. X* and *Dir. Z* means that 100% of the vibration mode is acting in this direction.

Frequencies and Vibration Modes Spreadsheet

20	Mode	w rad/sec	f Hz	T sec	ξ %	Shape	Dir.x	Dir.y	Dir.z	γ.x	γ.y	γ.z
1	Mode 1	4.77	0.76	1.32	5.00	Other	-0.00	0.00	1.00	-0.00	0.00	1297.03
2	Mode 2	9.99	1.59	0.63	5.00	Other	-1.00	0.00	0.00	-1330.80	0.63	0.01
3	Mode 3	10.14	1.61	0.62	5.00	Torsion	-0.99	0.00	-0.14	-7.56	0.00	-1.07
4	Mode 4	14.39	2.29	0.44	5.00	Other	-0.00	-0.00	1.00	-0.00	-0.00	458.54
5	Mode 5	23.42	3.73	0.27	5.00	Other	0.00	0.00	1.00	0.00	0.00	242.20
6	Mode 6	26.31	4.19	0.24	5.00	Other	0.67	-0.00	-0.74	0.11	-0.00	-0.12
7	Mode 7	26.34	4.19	0.24	5.00	Other	0.03	-0.00	1.00	0.00	-0.00	0.12
8	Mode 8	26.34	4.19	0.24	5.00	Other	-0.96	0.01	-0.28	-0.00	0.00	-0.00
9	Mode 9	26.34	4.19	0.24	5.00	Other	-0.16	-0.92	-0.35	-0.00	-0.00	-0.00
10	Mode 10	26.34	4.19	0.24	5.00	Other	-0.96	-0.28	0.07	-0.00	-0.00	0.00
11	Mode 11	26.34	4.19	0.24	5.00	Other	0.88	0.38	-0.27	0.00	0.00	-0.00
12	Mode 12	26.34	4.19	0.24	5.00	Other	-0.00	-0.00	-1.00	-0.00	-0.00	-6.53
13	Mode 13	26.35	4.19	0.24	5.00	Other	-0.00	0.00	-1.00	-0.01	0.00	-1.66
14	Mode 14	26.35	4.19	0.24	5.00	Other	-0.00	0.00	1.00	-0.01	0.00	2.42
15	Mode 15	26.36	4.20	0.24	5.00	Other	0.00	0.00	-1.00	0.00	0.00	-2.01
16	Mode 16	26.90	4.28	0.23	5.00	Other	-0.06	0.00	1.00	-0.09	0.00	1.42
17	Mode 17	26.91	4.28	0.23	5.00	Other	-0.01	0.00	-1.00	-0.02	0.00	-2.84
18	Mode 18	27.84	4.43	0.23	5.00	Other	1.00	0.01	0.00	402.38	3.70	0.00
19	Mode 19	28.63	4.56	0.22	5.00	Other	1.00	0.02	-0.07	1.75	0.03	-0.12
20	Mode 20	40.41	6.43	0.16	5.00	Other	-1.00	0.04	-0.00	-132.84	4.83	-0.00

When components Dir.x, Dir.y, and Dir.z are specified in the **Linear Seismic Directions** spreadsheet, VisualDesign automatically initialises the main vibration modes according to the highest modal contributions that were calculated in the **Frequencies** spreadsheet.

Spectral Analysis

Linear Seismic Directions Spreadsheet

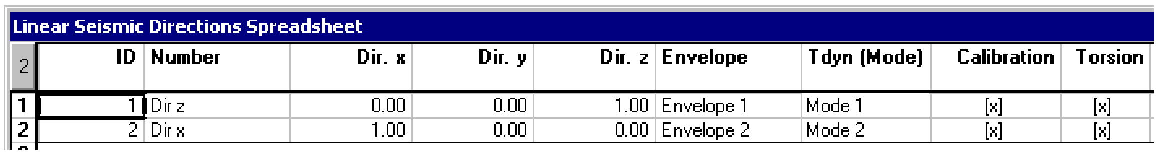
- Select the **Linear Seismic Directions** spreadsheet in **Loads** menu.

- Insert two lines and type the names Dirx and Dirz in the "Number" column. Enter a value of 1.0 as the x-component and z-component, with respect to each direction.

Note: Components Dir.x, Dir.y, and Dir. z

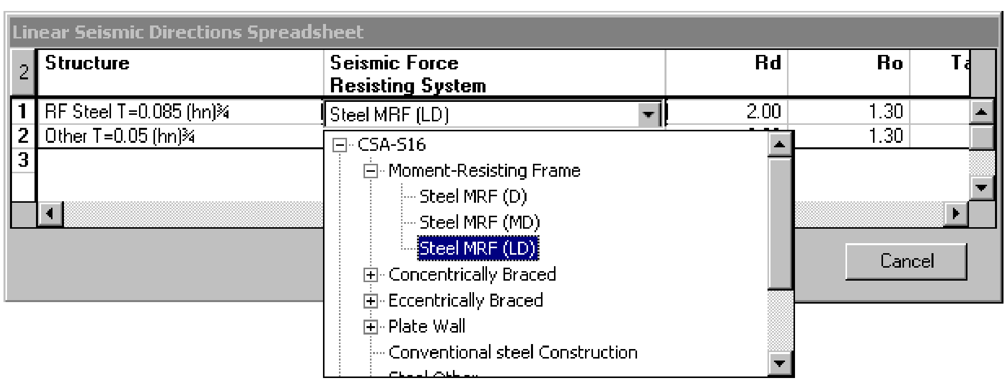
These components can be different from the value of 1.0. For example, you could have 0.67 for direction-x and 0.36 for direction-z, for a main vibration mode. (The geometry can be irregular or oriented at 45 degrees from global axes.) In this case, you should enter these values as components. The second main direction, which must be orthogonal to the first direction, will have similar components.

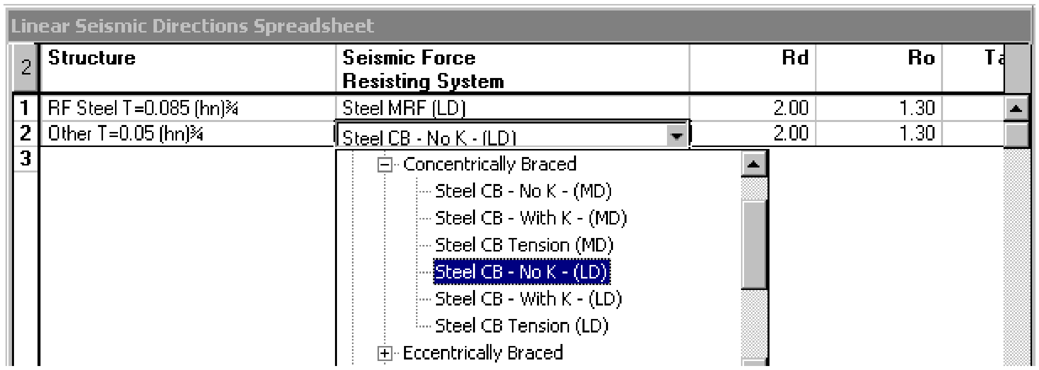
- In our example, we entered a value of 1.0 for direction x and z. Modes 1 and 2 were initialized by VisualDesign, according to the highest modal contributions. Select a different spectral envelope in the "Envelope" column, one for each direction.



- Go to the "Structure" column and select a lateral force resisting system as per NBC code, for each direction. Press the F1 control key to open VisualDesign On-line Help and learn more about this parameter.

- The next columns "Seismic Force Resisting System", "Rd", and "Ro" are relevant to the new NBC-2005 code. Double-click in the first column and expand the *CSA-S16* root. Select (double-click) the appropriate ductile frame for each direction. You will notice that the corresponding parameters Rd and Ro will be initialised.





Linear Seismic Directions Spreadsheet

2	Ta emp. sec	Ta dyn. sec	Ta chosen sec	S(Ta)	Mv	W kg	V kN
1	0.55	1.38	0.82	0.21	1.00	1894926.20	1509.37
2	0.32	0.56	0.48	0.36	1.00	1894926.20	2567.18

Linear Seismic Directions Spreadsheet

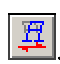
2	V kN	Ve kN	Vd kN	V chosen kN	Torsion M kN.m	Modal M/M %
1	1509.37	0.00	0.00	1207.50	0.00	0.00
2	2567.18	0.00	0.00	2053.75	0.00	0.00

- Parameters Ve, Vd, V chosen, Torsion M, and Modal M/M will be written in the spreadsheet when the spectral analysis will be completed.

Note Ve and Vd are calculated from modal analysis. Ve is the equivalent lateral force acting at the base and represents the elastic response. Vd is the seismic lateral force acting at the base from spectral analysis.

- Click OK to exit the spreadsheet.

Launch the Spectral Analysis.

- Click the **Spectral Analysis** icon . Press the "Analyse" button to start the analysis.

Important You must obtain at least 90% of participating mass in each seismic direction.

Spectral Analysis Results

Checking Participating Mass

- Reopen the **Linear Seismic Directions** spreadsheet and check the percentage of participating mass (Modal M/M), the base shear (Vd), and torsional moment acting at the base of the structure (Torsion M).

The participating mass exceeds 90% for each main direction. Therefore the spectral analysis is good. If you get less than 90%, eliminate local vibration modes, increase the number of requested modes and run another modal and spectral analysis.

Linear Seismic Directions Spreadsheet

2	W kg	V kN	Ve kN	Vd kN	V chosen kN	Torsion M kN.m	Modal M/M %
1	1957172.33	772.05	2048.74	390.24	617.64	1976.45	99.70
2	1957172.33	1767.67	5118.66	1312.48	1414.14	6363.63	99.67

Equivalent Static Loads

Torsion loads and loads due to inelastic effects (P-delta) have been generated during the spectral analysis. VisualDesign modified these loads to equivalent static loads Tx Dir. x, z and Tr Dir. x, z, which are listed in the **Loads Definition** spreadsheet. VisualDesign modified these loads to equivalent static loads. These static equivalent loads will be applied to the structure nodes when a static analysis or design will be launched.

Loads Ri:Dir. x, z, are corresponding to the retro calculations done by VisualDesign for ductile frames.

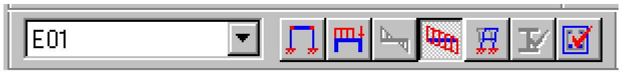
Loads Definition

Load Case | Dynamic | Ice

15	Number	Type	Auto Generation combinaisons	Auto-generated Torsion	Definition
1	Dead	(D) Dead	[x]	[_]	D : Structure Dead Load
2	Snow	(L) Snow	[x]	[_]	S01 : Snow
3	Live	(L) Live	[x]	[_]	Lx : Floor live load
4	D1_Roof	(D) Dead	[x]	[_]	D : Roof Dead Load
5	D2_Floors	(D) Dead	[x]	[_]	D : Floor Dead Load
6	Tx.Dir z	(E) Seismic	[_]	[x]	Loads on nodes for theta x
7	Tr.Dir z	(E) Seismic	[_]	[x]	Loads at nodes for torsion
8	R6:Dir z	(E) Seismic	[_]	[x]	
9	Tx.Dir x	(E) Seismic	[_]	[x]	Loads on nodes for theta x
10	Tr.Dir x	(E) Seismic	[_]	[x]	Loads at nodes for torsion
11	R6:Dir x	(E) Seismic	[_]	[x]	
12	R2:Dir x	(E) Seismic	[_]	[x]	
13	R3:Dir x	(E) Seismic	[_]	[x]	
14	R4:Dir x	(E) Seismic	[_]	[x]	
15	R5:Dir x	(E) Seismic	[_]	[x]	

Seismic Envelopes E01 and E02

The Envelope mode is automatically activated when closing the **Spectral Analysis** dialog box. Envelopes *E01* and *E02* includes the resulting seismic forces acting respectively in the z-direction and x-direction.



- To have a look at seismic forces and deflections, select the **Results** tab of **View Options** and activate a force diagram.

Information on Levels

- Go to **Results** / **Modal/Spectral** / **Levels**. Check if inter-story drifts do not exceed the allowable limit. Refer to building code.

Information on Levels (stories) according to Seismic Direction

9	Seismic Direction	Height m	Width m	F kN	V kN	W kN	δave mm	δmax mm	Bx	Δmx mm	hs m	θx
1	Dir x	12.00	45.00	539.63	539.63	4382.80	39.26	48.13	1.23	7.66	4.00	0.02
2	Dir x	8.00	45.00	532.25	1071.88	11753.21	32.41	43.18	1.33	13.92	4.00	0.04
3	Dir x	4.00	45.00	341.84	1413.72	19120.44	18.99	29.39	1.55	17.22	4.00	0.06
4	Dir x	0.00	45.00	0.42	1414.14	19193.30	1.77	9.98	5.63	1.77	0.00	0.00
5												
6	Dir z	12.00	32.00	326.01	326.01	4382.80	102.97	107.58	1.04	28.04	4.00	0.09
7	Dir z	8.00	32.00	142.95	468.96	11753.21	80.65	96.08	1.19	41.70	4.00	0.26
8	Dir z	4.00	32.00	148.17	617.14	19120.44	41.70	61.57	1.48	37.50	4.00	0.28
9	Dir z	0.00	32.00	0.51	617.64	19193.30	4.48	22.15	4.95	4.48	0.00	0.00

Column Δmx indicates inter-story drifts. The code limits this displacement to 2% of story height. In our case, this limit is equal to 0.02 x 4000mm = 80mm. Inter-story drifts are well below this limit.

Node Displacements

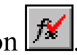
Node displacements are available for a selected vibration mode.

- Select vibration mode #1 on Activation toolbar and go to **Results** / **Modal/Spectral** / **Node Displacements.**

Node Displacements Spreadsheet

144	Number	Displ. x mm	Displ. y mm	Displ. z mm	θx °	θy °	θz °
118	cF3	-0.00	-0.00	1.02	0.00	-0.00	0.00
119	dF3	-0.00	-0.00	1.02	0.00	-0.00	0.00
120	eF3	-0.00	-0.00	1.02	0.00	-0.00	0.00
121	207	-0.00	-0.00	0.92	0.00	0.00	0.00
122	208	-0.00	-0.00	0.92	0.00	-0.00	0.00
123	209	-0.00	-0.00	0.58	0.00	0.00	0.00
124	210	-0.00	-0.00	0.58	0.00	-0.00	-0.00
125	211	-0.00	-0.00	0.18	0.00	0.00	0.00
126	212	-0.00	-0.00	0.18	0.00	-0.00	-0.00
127	213	-0.00	-0.00	0.92	0.00	-0.00	0.00
128	214	-0.00	-0.00	0.92	0.00	0.00	0.00

Steel Design with Ductile Frames

- Launch the steel design by clicking on this icon .

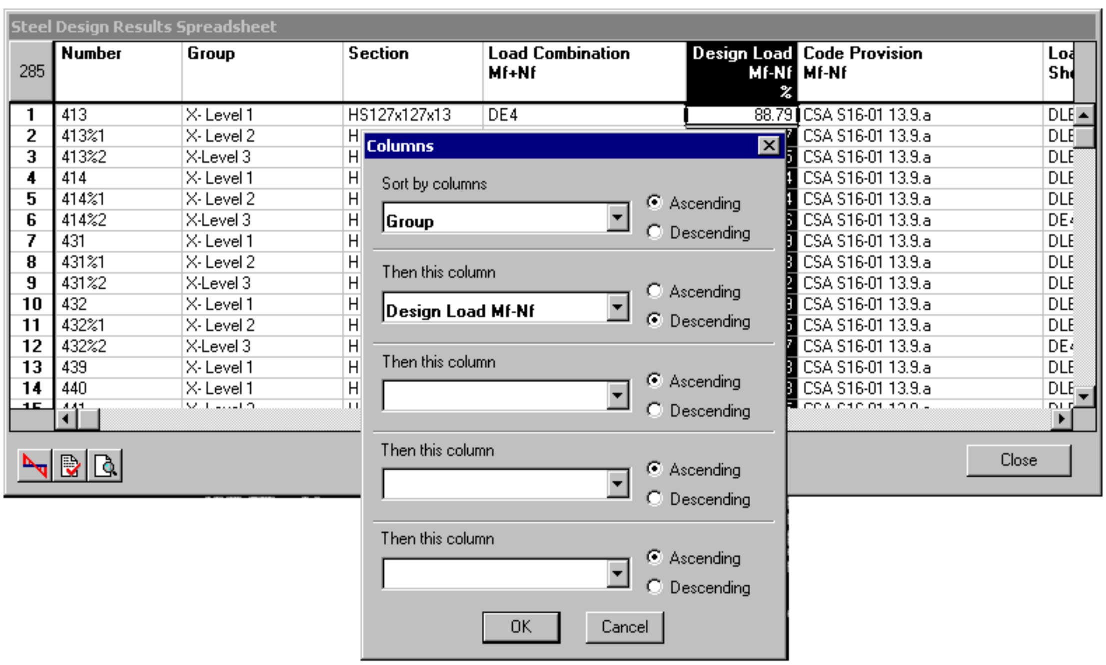
During the Design Process:

In the **Analysis and Design** dialog box, activate the option *Dynamic and Spectral analysis at each cycle of design* if you want VisualDesign™ to rerun the modal and spectral analyses at each design cycle (shapes will change and so will the seismic response).

If vibration modes and main seismic directions have changed during the design, VisualDesign will automatically initialize data in the **Linear Seismic Directions** spreadsheet during the design process.

Steel Design Results Spreadsheet

- Go to **Results / Structure Design / Steel** and verify members' design loads. Sort data using groups and design loads.



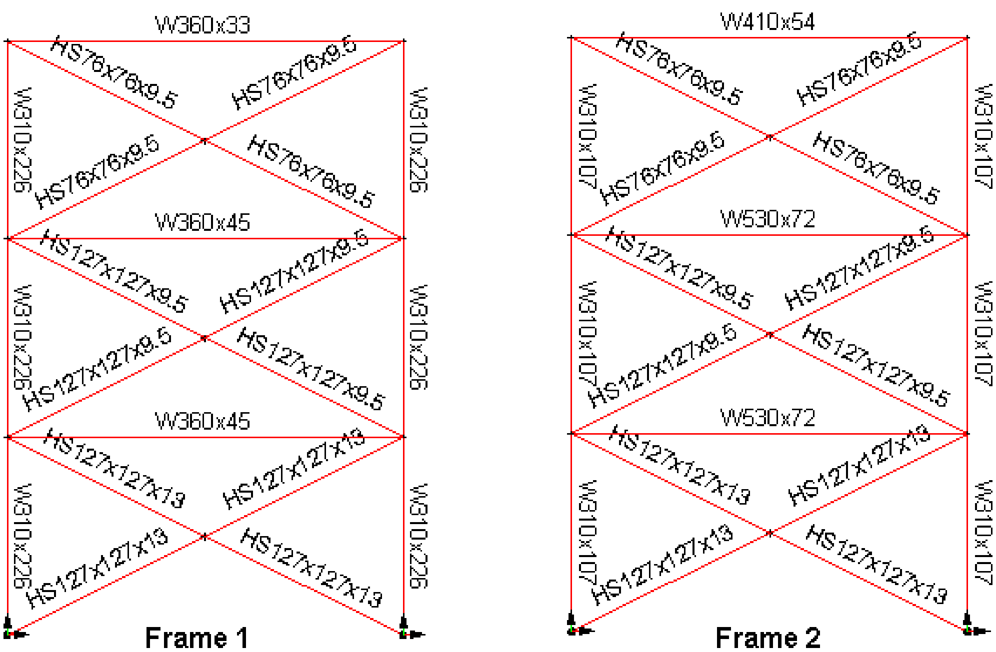
Steel Design Results Spreadsheet

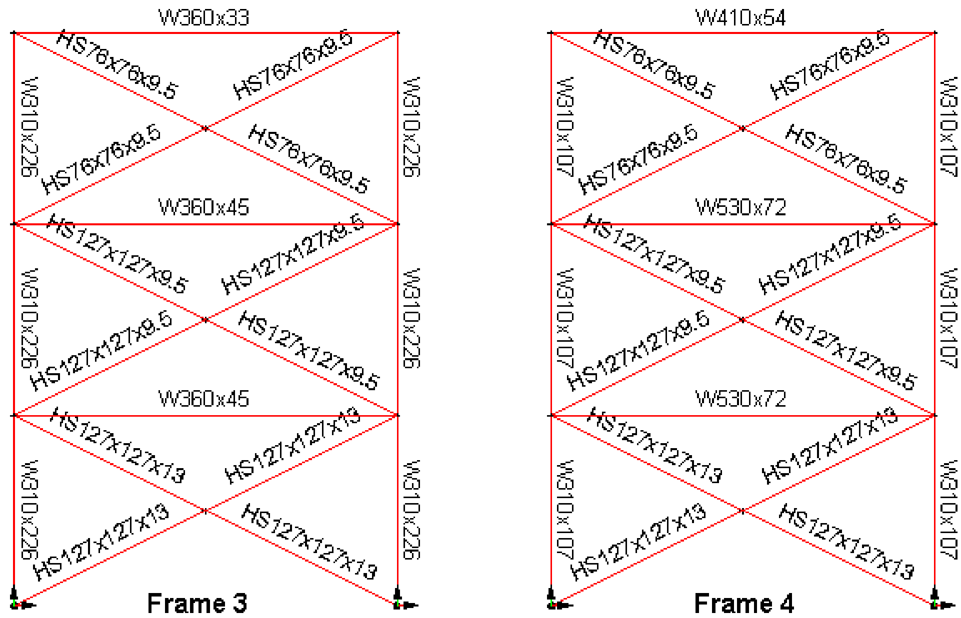
285	Number	Group	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear
227	dC2-Y	C4-X	W310x107	DLE6	22.30	CSA S16-01 13.8.2c	DLE5
228	eC2-Y	C4-X	W310x107	DLE6	21.05	CSA S16-01 13.8.2c	DLE5
229	cD1-Z	P1-RMF	W460x97	DL1	84.05	CSA S16-01 13.9.b	DL1
230	cB1-Z	P1-RMF	W460x97	DL1	84.05	CSA S16-01 13.9.b	DL1
231	cC1-Z	P1-RMF	W460x97	DL1	82.96	CSA S16-01 13.9.b	DL1
232	cD2-Z	P2-RMF	W610x84	DL1	91.37	CSA S16-01 13.6	DL1
233	cB2-Z	P2-RMF	W610x84	DL1	91.37	CSA S16-01 13.6	DL1
234	cC2-Z	P2-RMF	W610x84	DL1	90.94	CSA S16-01 13.6	DL1
235	cB3-Z	P3-RMF	W360x64	DL1	96.77	CSA S16-01 13.6	DL1
236	cD3-Z	P3-RMF	W360x64	DL1	96.77	CSA S16-01 13.6	DL1
237	cC3-Z	P3-RMF	W360x64	DL1	94.99	CSA S16-01 13.6	DL1
238	455	X- Level 1	HS127x127x13	DLE6	99.10	CSA S16-01 13.9.a	DLE5
239	414	X- Level 1	HS127x127x13	DE4	93.04	CSA S16-01 13.9.a	DLE5
240	431	X- Level 1	HS127x127x13	DE4	89.39	CSA S16-01 13.9.a	DLE5

Close

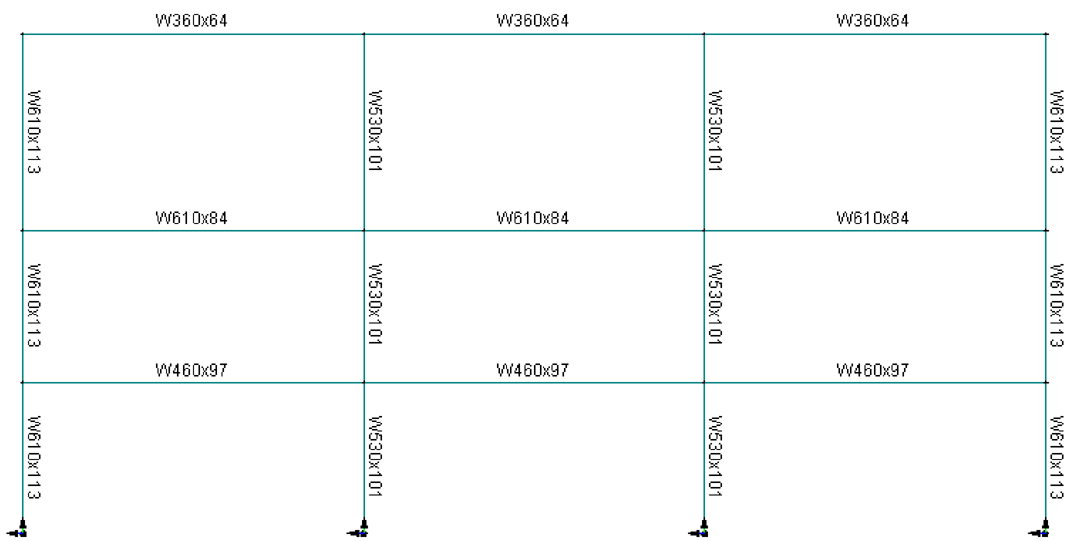
Steel design results are OK.

Chosen Shapes for Concentrically Braced Frames





Chosen Shapes for Moment-Resisting Frame



Seismic Steel Design Results

The seismic design results spreadsheet is available when a load combination is selected on Activation toolbar. This spreadsheet is composed of parameters that are relevant to ductile frames calculated according to section 27 of S16-01 Standard.

Direction x – Envelope E02

- Select the load combination DE4 (=1.0D + 1.0E02) on Activation toolbar. The spectral envelope E02 corresponds to lateral forces towards direction-x where concentrically braced frames are modeled to dissipate this energy.

- Display the concentrically braced frames through the **Selections** dialog box (**Edit / Select / Choose a selection**) and mask the rest of the structure.

- Go to **Results / Structure Design / Steel – Seismic Design** and verify the members that are part of ductile frames, which dissipated energy during the earthquake. Sort data using design groups.

Steel Design Results Spreadsheet - Seismic Design

285	Number	Shape	Group	Usage	Ag (mm²)	Aw (mm²)	Aw/Ag	Zx (10³mm³)	Ry	Fy (MPa)
48	474	HS127x127x13	X- Level 1	[X t] Diagonal	5391.09	0.00	0.00	224.52	1.10	350.00
49	cD3-Z	W360x64	P3-RMF	[ ] Beam	8140.00	2464.00	0.30	1140.00	1.10	300.00
50	cB3-Z	W360x64	P3-RMF	[ ] Beam	8140.00	2464.00	0.30	1140.00	1.10	300.00
51	cC3-Z	W360x64	P3-RMF	[ ] Beam	8140.00	2464.00	0.30	1140.00	1.10	300.00
52	cB2-Z	W610x84	P2-RMF	[ ] Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
53	cC2-Z	W610x84	P2-RMF	[ ] Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
54	cD2-Z	W610x84	P2-RMF	[ ] Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
55	cC1-Z	W460x97	P1-RMF	[ ] Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
56	cD1-Z	W460x97	P1-RMF	[ ] Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
57	cB1-Z	W460x97	P1-RMF	[ ] Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
58	eC0-Y	W310x107	C4-X	[X] Column	13600.00	3019.30	0.22	1770.00	1.10	300.00
59	dC0-Y	W310x107	C4-X	[X] Column	13600.00	3019.30	0.22	1770.00	1.10	300.00

- Press the F1 control key to get the definition of each column composing this spreadsheet, through VisualDesign On-line Help.

Steel Design Results Spreadsheet - Seismic Design

285	Class Mx Bending	Class My Bending	Class Web	Class Compression	e (m)	Pf (kN)	Vf (kN)	Vp (kN)	V'p (kN)	Mp (kN.m)	M'p (kN.m)
48	1	1	1	1	4.47	1398.68	1.93	0.00	0.00	78.58	23.99
49	1	1	1	1	9.00	8.46	44.55	440.86	440.86	342.00	402.16
50	1	1	1	1	9.00	8.46	44.55	440.86	440.86	342.00	402.16
51	1	1	1	1	9.00	5.81	44.47	440.86	440.86	342.00	402.60
52	2	2	2	2	9.00	0.02	98.07	885.06	885.06	708.00	835.44
53	2	2	2	2	9.00	0.02	97.90	885.06	885.06	708.00	835.43
54	2	2	2	2	9.00	0.02	98.07	885.06	885.06	708.00	835.44
55	1	1	1	1	9.00	0.01	98.40	876.55	876.55	654.00	771.72
56	1	1	1	1	9.00	0.01	98.48	876.55	876.55	654.00	771.72
57	1	1	1	1	9.00	0.01	98.48	876.55	876.55	654.00	771.72
58	2	2	1	2	4.00	2097.01	1.10	559.33	479.80	531.00	304.53
59	2	2	1	2	4.00	2466.83	1.71	559.33	445.52	531.00	247.74

Steel Design Results Spreadsheet - Seismic Design

285	ϕV'p or 2ϕM'p/e (kN)	1.1RyMp (kN.m)	AgRyFy (kN)	0.2AgRyFy (kN)	1.2Cpr (kN)	γ (*)	γmax (*)	e min (m)	e max (m)	1.15 Ry Vn (kN)	1.30 Ry Vn (kN)
48	0.00	95.08	2075.57	415.11	1019.21	0.00	0.00	0.13	0.13	0.00	0.00
49	68.40	413.82	2686.20	537.24	2351.29	0.00	1.72	0.35	10.00	96.14	108.68
50	68.40	413.82	2686.20	537.24	2351.29	0.00	1.72	0.35	10.00	96.14	108.68
51	68.40	413.82	2686.20	537.24	2351.29	0.00	1.72	0.35	10.00	96.14	108.68
52	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
53	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
54	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
55	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
56	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
57	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
58	137.04	642.51	4488.00	897.60	4326.57	0.00	1.72	0.35	10.00	192.62	217.74
59	111.48	642.51	4488.00	897.60	4326.57	0.00	1.72	0.35	10.00	156.70	177.13

Direction z – Envelope E01

The load combination DE3 (=1.0D + 1.0E01) is selected. The lateral forces are acting on the moment-resisting frame.

- Display the moment-resisting frame through the **Selections** dialog box.
- Open the seismic steel design spreadsheet. Sort data using design groups.

Steel Design Results Spreadsheet - Seismic Design

21	Number	Shape	Group	Usage	Ag (mm²)	Aw (mm²)	Aw/Ag	Zx (10³mm³)	Ry	Fy (MPa)
9	cC2-Y	W530x101	C3-RMF	Column	12900.00	5473.98	0.42	2620.00	1.10	300.00
10	cB1-Y	W610x113	C4-RMF	Column	14400.00	6422.08	0.45	3290.00	1.10	300.00
11	cB2-Y	W610x113	C4-RMF	Column	14400.00	6422.08	0.45	3290.00	1.10	300.00
12	cB0-Y	W610x113	C4-RMF	Column	14400.00	6422.08	0.45	3290.00	1.10	300.00
13	cB1-Z	W460x97	P1-RMF	Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
14	cC1-Z	W460x97	P1-RMF	Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
15	cD1-Z	W460x97	P1-RMF	Beam	12300.00	4879.20	0.40	2180.00	1.10	300.00
16	cD2-Z	W610x84	P2-RMF	Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
17	cB2-Z	W610x84	P2-RMF	Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
18	cC2-Z	W610x84	P2-RMF	Beam	10700.00	5153.40	0.48	2360.00	1.10	300.00
19	cD3-Z	W360x64	P3-RMF	Beam	8140.00	2464.00	0.30	1140.00	1.10	300.00

Steel Design Results Spreadsheet - Seismic Design

21	Class Mx Bending	Class My Bending	Class Web	Class Compression	e (m)	Pf (kN)	Vf (kN)	Vp (kN)	V'p (kN)	Mp (kN.m)	M'p (kN.m)
9	1	1	1	1	5.21	164.83	40.05	965.79	964.92	786.00	887.98
10	1	1	1	1	4.00	536.34	131.24	1123.58	1114.89	987.00	1020.06
11	1	1	1	1	5.21	169.88	77.88	1123.58	1122.71	987.00	1118.86
12	1	1	1	1	4.00	902.73	107.02	1123.58	1098.78	987.00	921.29
13	1	1	1	1	9.00	0.00	140.32	876.55	876.55	654.00	771.72
14	1	1	1	1	9.00	0.00	138.62	876.55	876.55	654.00	771.72
15	1	1	1	1	9.00	0.00	140.32	876.55	876.55	654.00	771.72
16	2	2	2	2	9.00	0.00	128.66	885.06	885.06	708.00	835.44
17	2	2	2	2	9.00	0.00	128.66	885.06	885.06	708.00	835.44
18	2	2	2	2	9.00	0.00	126.73	885.06	885.06	708.00	835.44
19	1	1	1	1	9.00	10.04	54.90	440.86	440.86	342.00	401.90

Steel Design Results Spreadsheet - Seismic Design

21	ϕV'p or 2ϕM'p/e kN	1.1RyMp kN.m	AgRyFy kN	0.2AgRyFy kN	1.2Cpr kN	γ °	γmax °	e min m	e max m	1.15 Ry Vn kN	1.30 Ry Vn kN
9	271.50	951.06	4257.00	851.40	1873.00	0.00	1.72	0.55	10.00	381.61	431.39
10	444.15	1194.27	4752.00	950.40	3200.70	0.00	1.72	0.65	10.00	624.28	705.71
11	340.93	1194.27	4752.00	950.40	2301.44	0.00	1.72	0.65	10.00	479.20	541.70
12	414.58	1194.27	4752.00	950.40	3200.70	0.00	1.72	0.65	3.04	582.71	658.72
13	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
14	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
15	130.80	791.34	4059.00	811.80	4079.90	0.00	1.72	0.50	10.00	183.85	207.83
16	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
17	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
18	141.60	856.68	3531.00	706.20	3830.59	0.00	1.72	0.60	10.00	199.03	224.99
19	68.40	413.82	2686.20	537.24	2351.29	0.00	1.72	0.35	10.00	96.14	108.68

Checking the Main Vibration Modes

- Open the **Linear Seismic Directions** spreadsheet (**Loads** menu).

Linear Seismic Directions Spreadsheet

2	Number	Dir. x	Dir. y	Dir. z	Envelope	Tdyn (Mode)	Calibration
1	Dir z	0.00	0.00	1.00	Envelope 1	Mode 3	[x]
2	Dir x	1.00	0.00	0.00	Envelope 2	Mode 5	[x]

Modes 1 and 2 have been replaced by modes 3 and 5 during the design process.

Interstory Drifts

- Check the interstory drifts through the **Levels** spreadsheet (**Results / Modal / Spectral**) to make sure that they are below the allowable limit fixed by the NBC code.

Information on Levels (stories) according to Seismic Direction

9	Seismic Direction	Height m	Width m	F kN	V kN	W kN	δave mm	δmax mm	Bx	Δmx mm	hs m	θx
1	Dir x	12.00	45.00	534.21	534.21	4124.41	44.75	226.30	5.06	34.56	4.00	0.06
2	Dir x	8.00	45.00	988.53	1522.74	11297.03	29.69	38.02	1.28	13.79	4.00	0.03
3	Dir x	4.00	45.00	530.31	2053.05	18500.24	15.94	25.68	1.61	14.48	4.00	0.03
4	Dir x	0.00	45.00	0.63	2053.69	18582.37	1.47	8.38	5.71	1.47	0.00	0.00
5												
6	Dir z	12.00	32.00	472.04	472.04	4124.41	119.92	131.20	1.09	21.11	4.00	0.04
7	Dir z	8.00	32.00	404.83	876.87	11297.03	100.48	127.46	1.27	34.99	4.00	0.11
8	Dir z	4.00	32.00	330.05	1206.91	18500.24	66.97	83.97	1.25	60.21	4.00	0.22
9	Dir z	0.00	32.00	0.55	1207.46	18582.37	7.56	41.52	5.49	7.56	0.00	0.00

The allowable interstory drift is 80mm and the maximum reached is 60.21mm.

E X A M P L E 8

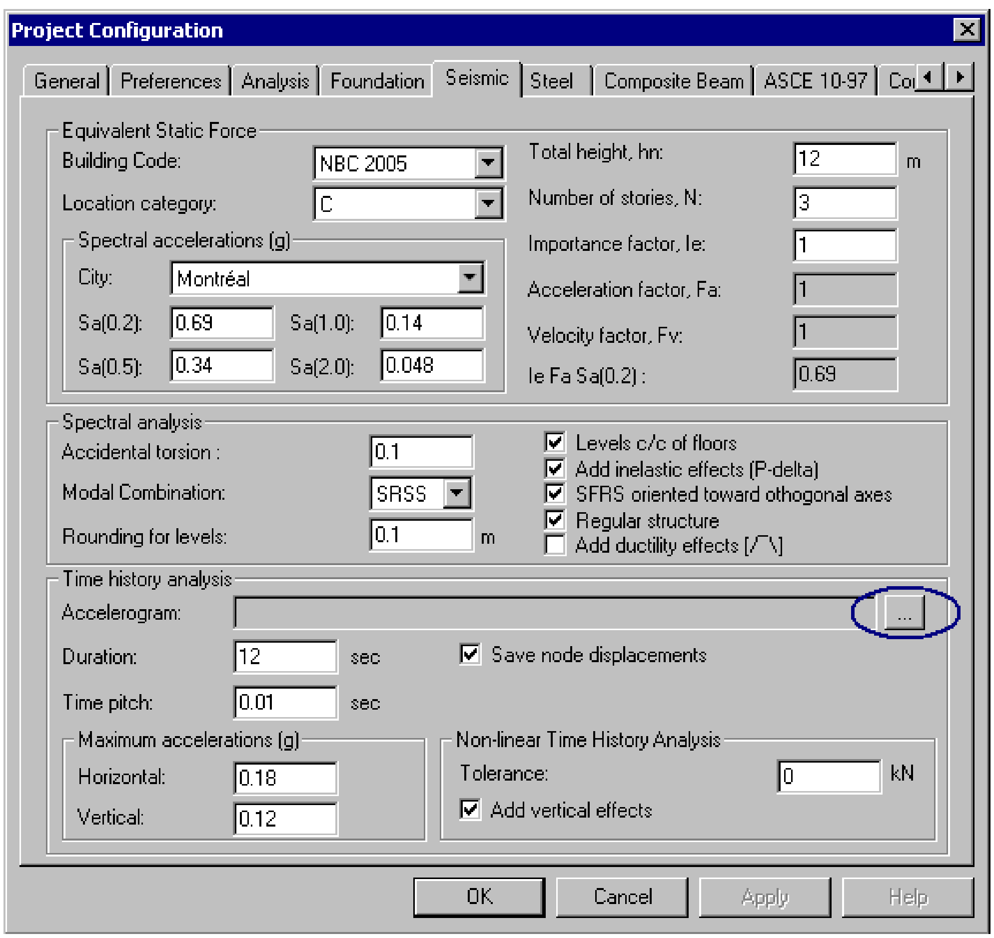
Linear Time History Analysis

Linear Time History Analysis & Steel Design

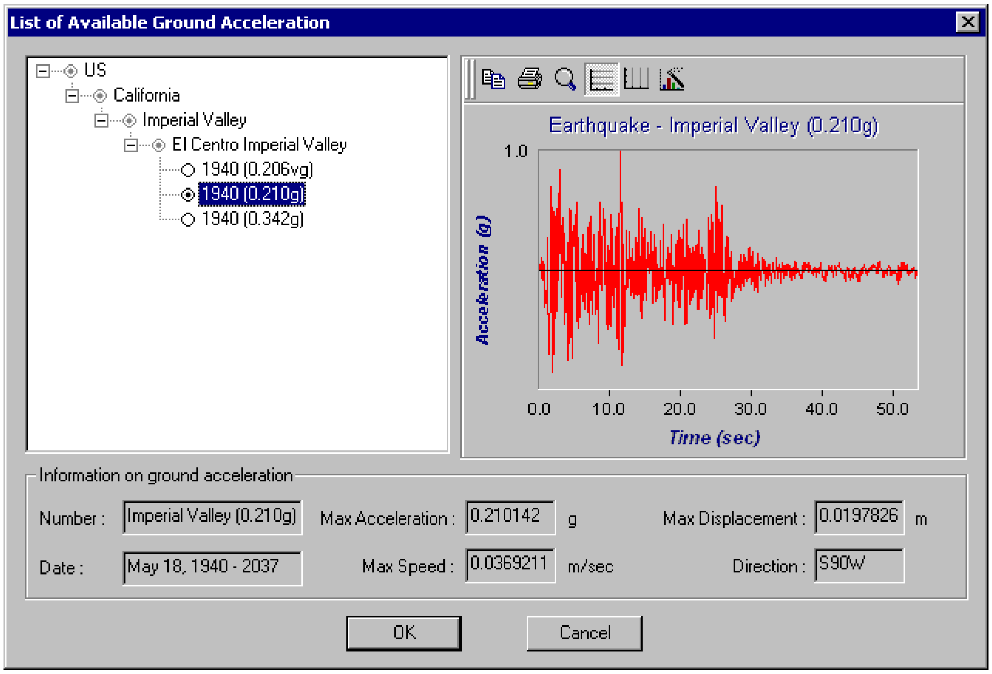
We are going to run a linear time history analysis of the building (before the seismic design) that we studied in the last examples. Then, we will run a steel design.

Project Configuration

- Open the **Project Configuration** dialog box (**File** menu) and select the **Analysis** tab. Activate a linear type of analysis.
- Select the **Seismic** tab. Disable the option "Add ductility Effects [/‾\ ]". Click the button next to the "Accelerogram" list box.

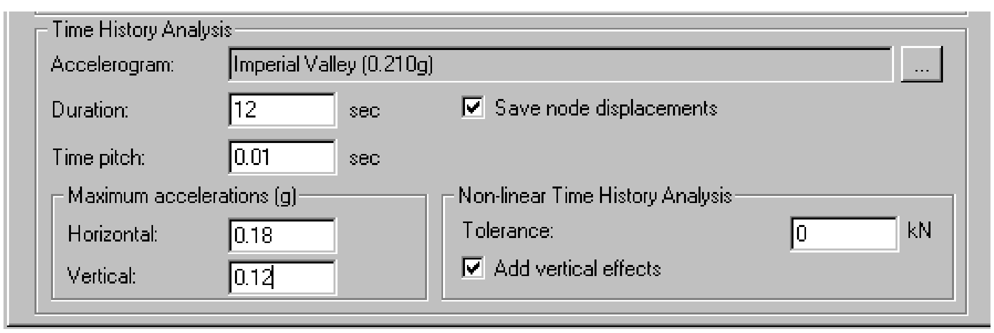


- Activate the radio button to select an accelerogram.



- Click OK. You will go back to the **Seismic** tab.

- The time pitch is associated to the accelerogram. Enter the duration for the application of this accelerogram and specify the maximum horizontal and vertical accelerations. For Montreal area, these accelerations are respectively equal to 0.18g and 0.12g (2/3 x horiz. acc.).



- Activate option *Save node displacements*.

- Press OK to save data and exit the **Project Configuration** dialog box.

Member Usages

- Select all members and choose a *Standard* usage in the **Member** tab.

Loads and Load Combinations

Use the **Combination Generation Wizard** to generate load combinations. Include time history envelopes *Et01* and *Et02* and *Ultimate* envelope.

Load Combinations

Load Combinations | Load Factors

9	Number	Status	Definition
1	DE3	Ultimate	1.00D+1.00Et01
2	DE4	Ultimate	1.00D+1.00Et02
3	DL1	Ultimate	1.25D+1.50S01+1.50Lx
4	DL2	Ultimate	0.85D+1.50S01+1.50Lx
5	DLE5	Ultimate	1.00D+0.50S01+1.00Et01+0.50Lx
6	DLE6	Ultimate	1.00D+0.50S01+1.00Et02+0.50Lx
7	DL8	Service	1.00D+1.00S01+1.00Lx
8	L9	Instant. Deflection	1.00S01+1.00Lx
9	Mass_7	Mass	Mass

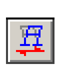
Modal and Spectral Analysis

The modal and spectral analyses are requested before running a linear time history analysis because accidental torsion effects are calculated during the spectral analysis.

- Click on this icon  to launch the modal analysis. Select the *Mass* load combination in the dialog box.
- Open the **Linear Seismic Directions** spreadsheet. Data are the same as obtained in the last example.

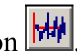
Linear Seismic Directions Spreadsheet

2	Number	Dir. x	Dir. y	Dir. z	Envelope	Tdyn (Mode)	Calibration	Torsion
1	Dir z	0.00	0.00	1.00	Envelope 1	Mode 3	[x]	[x]
2	Dir x	1.00	0.00	0.00	Envelope 2	Mode 5	[x]	[x]

- Close the spreadsheet.
- Click this icon  to launch the spectral analysis.

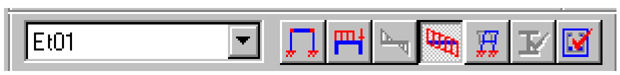
Linear Time History Analysis

- While keeping the [Ctrl] key down, select nodes for which displacements are required. (If you do not select nodes, displacements will be calculated for all nodes and this type of analysis can be very long). In this example, we selected nodes eA2 and eA3, which are located at the top of the structure, near a corner.

- Start the **Time History** analysis by pressing this icon .

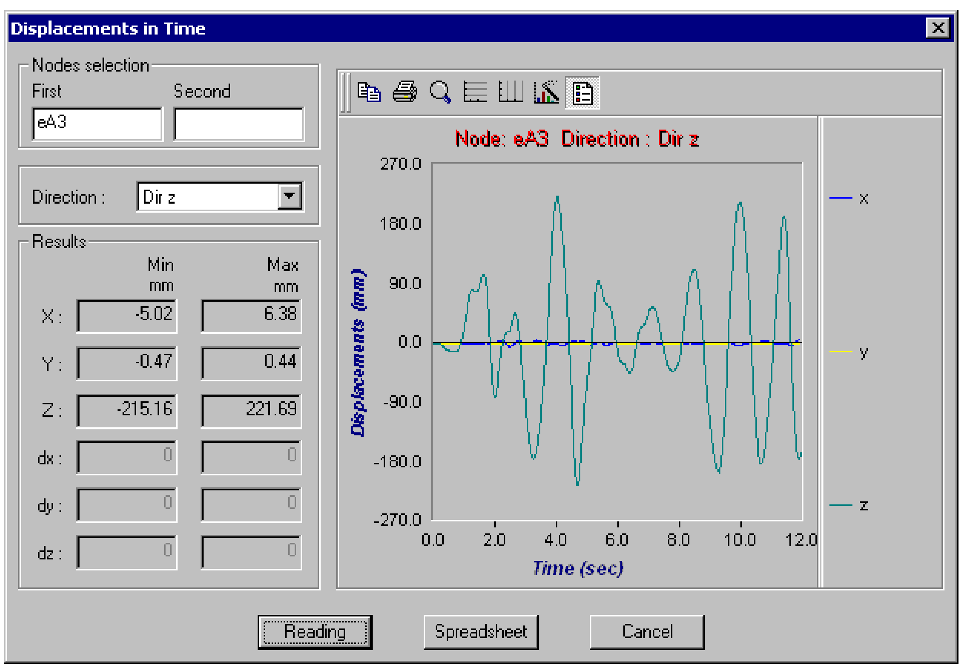
Time History Results

When the analysis is completed, time history envelopes will be available on Activation toolbar.

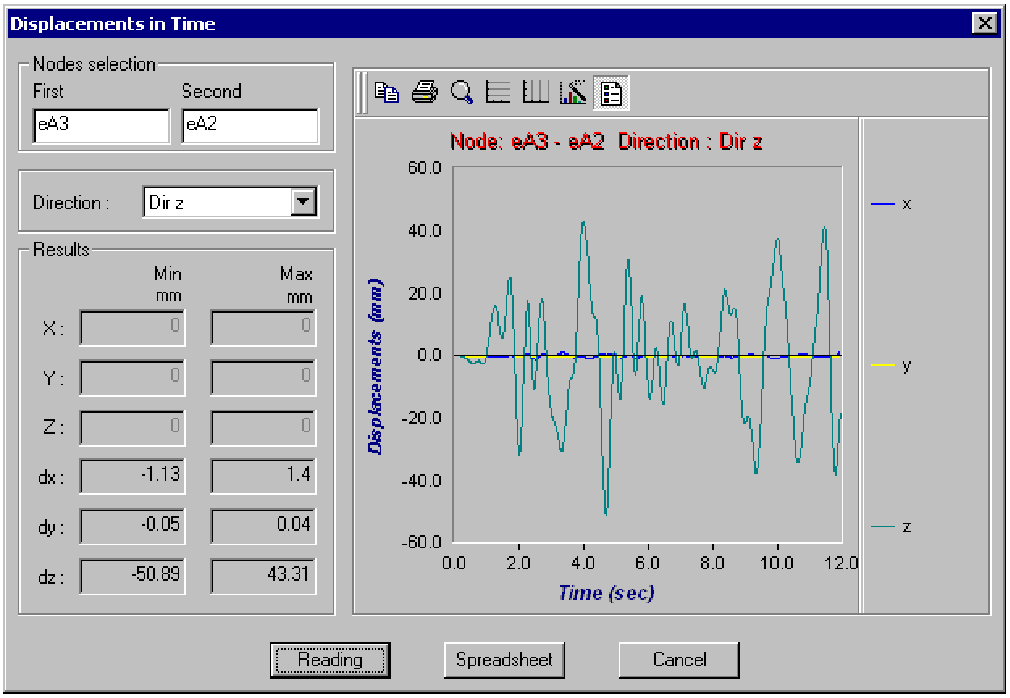


- Go to **Results** menu and select **Time History / Node Displacements in Time**.

- Enter a node number and press the "Reading" button. To look at results in the form of a spreadsheet, press the "Spreadsheet" button.



To know differential displacements between two nodes, enter node numbers in appropriate fields. Select the direction and press the "Reading" button.



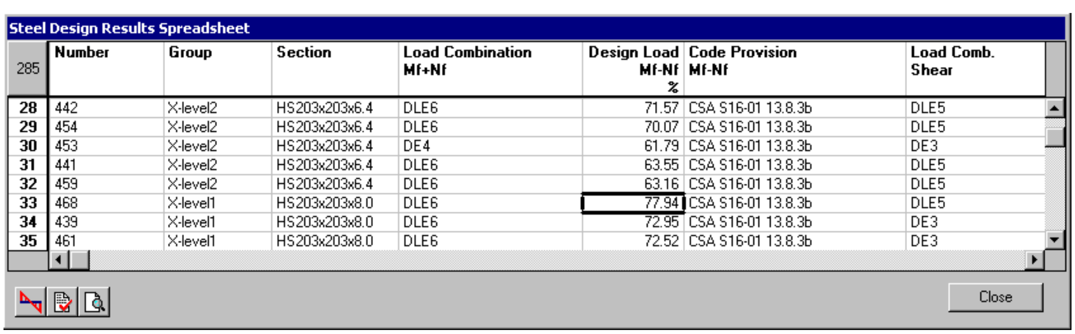
Steel Design

We will run a steel design that includes the time history envelopes Et01 and Et02.

- Select nodes eA2 and eA3.

- Launch the steel design by clicking on this icon . The modal, spectral, and time history analyses will be automatically launched in the design process.

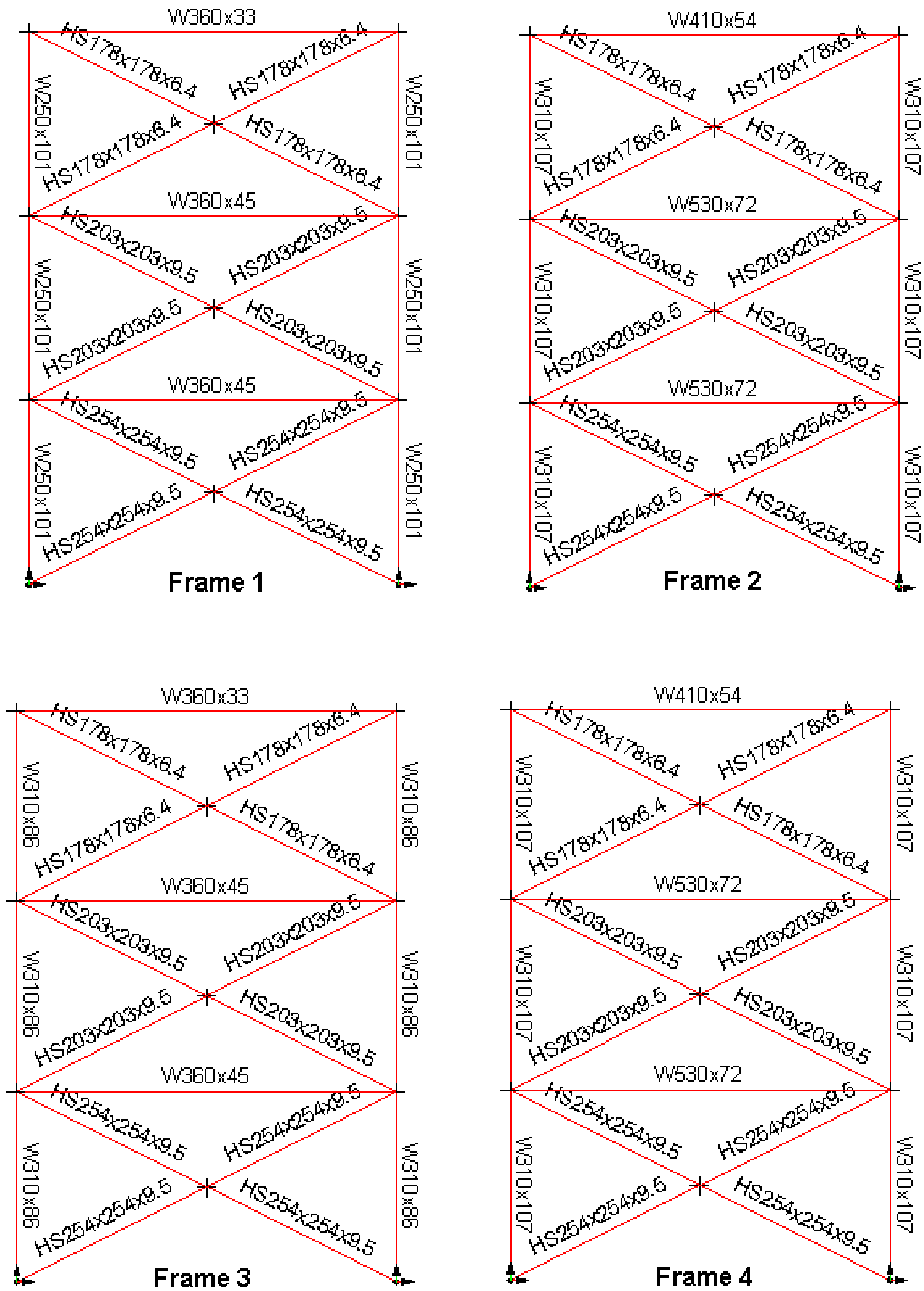
Steel Design Results



Steel Design Results Spreadsheet

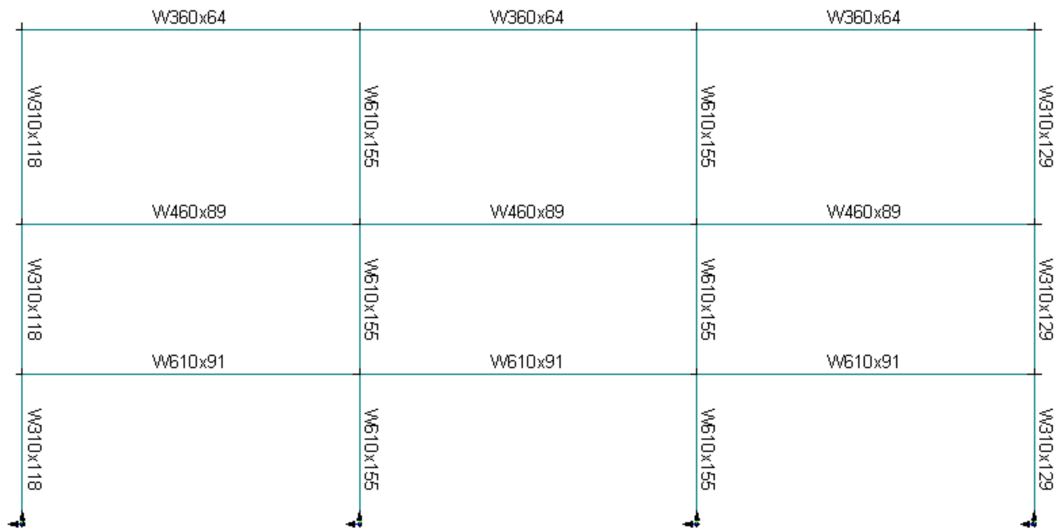
285	Number	Group	Section	Load Combination Mf+Nf	Design Load Mf-Nf %	Code Provision Mf-Nf	Load Comb. Shear
28	442	X-level2	HS203x203x6.4	DLE6	71.57	CSA S16-01 13.8.3b	DLE5
29	454	X-level2	HS203x203x6.4	DLE6	70.07	CSA S16-01 13.8.3b	DLE5
30	453	X-level2	HS203x203x6.4	DE4	61.79	CSA S16-01 13.8.3b	DE3
31	441	X-level2	HS203x203x6.4	DLE6	63.55	CSA S16-01 13.8.3b	DLE5
32	459	X-level2	HS203x203x6.4	DLE6	63.16	CSA S16-01 13.8.3b	DLE5
33	468	X-level1	HS203x203x8.0	DLE6	77.94	CSA S16-01 13.8.3b	DLE5
34	439	X-level1	HS203x203x8.0	DLE6	72.95	CSA S16-01 13.8.3b	DE3
35	461	X-level1	HS203x203x8.0	DLE6	72.52	CSA S16-01 13.8.3b	DE3

Chosen Shapes for Bracings



The bracings are much bigger than those obtained from the seismic design with ductile frames but columns are lighter.

Chosen Shapes for Moment-Resisting Frame



These shapes are almost equivalent to those obtained from the seismic design using ductile frames. Columns are a bit smaller and beams are almost the same.

Checking the Main Vibration Modes

Linear Seismic Directions Spreadsheet

2	Number	Dir. x	Dir. y	Dir. z	Envelope	Tdyn (Mode)	Calibration	Torsion
1	Dir z	0.00	0.00	1.00	Envelope 1	Mode 3	[x]	[x]
2	Dir x	1.00	0.00	0.00	Envelope 2	Mode 8	[x]	[x]

Mode 5 was changed to mode 8 for direction x during the design process. Direction z remained the same (mode 3).

Procedures for Linear Time History Analysis

- Select the **Preferences tab** of **Project Configuration** dialog box and activate a linear type of analysis.

- Select the **Seismic tab** and complete parameters for the spectral analysis. They are required to calibrate the linear time history analysis. Select an accelerogram and enter the duration, the maximum horizontal and vertical accelerations and activate the option *Save node displacements*.

- Generate load combinations with the **Combination Generation Wizard** including the linear time history envelopes *Et01* and *Et02*, if required.

- Run a modal analysis.

- Open the **Linear Seismic Directions** spreadsheet and enter components dir.x, dir.y, and dir.z for two orthogonal directions, as they are supplied in the **Frequencies and Vibration Modes** spreadsheet. VisualDesign will automatically choose the vibration mode numbers corresponding to these components, which contribute the most for each seismic direction.

- Run the spectral analysis. Make sure that at least 90% of participating mass is used for each seismic direction. Select the **Information on Levels** spreadsheet and look at inter story drifts. They must not exceed the limit permitted by the Code.

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

- Launch a linear time history analysis.

- Go to **Results** menu and select **Time History / Nodes Displacements in Time**.

- Run a static analysis or design**.**

- Look at load combinations and envelopes results.

EXAMPLE 9

Non-Linear Time History Analysis

Step-By-Step Procedures

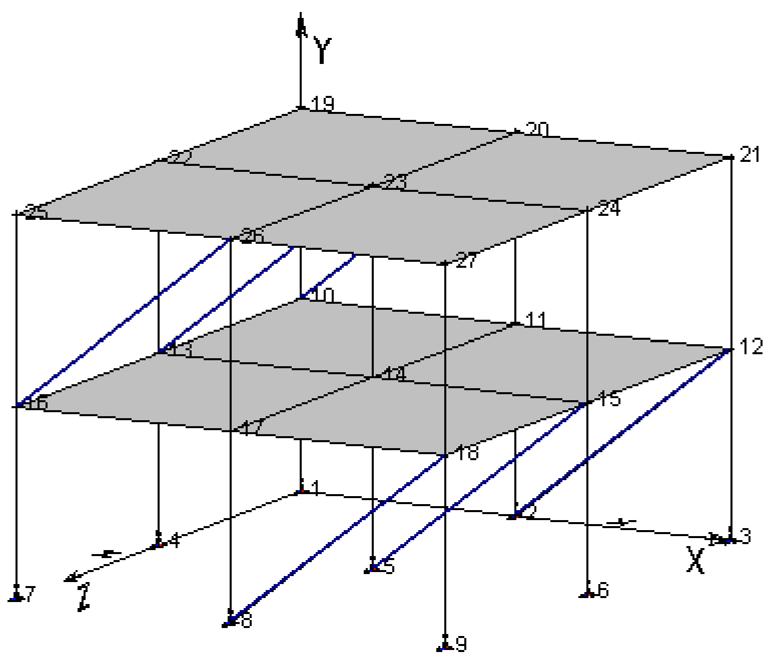
Location of Friction Dampers in the Structure

Procedure:

- Run a modal and spectral analysis of the structure.
- Take note of the maximum forces in bracings (tension and compression).
- Choose the members that will behave as elastoplastic members. They will absorb the amount of tension and compression force that will be specified in the **Behaviour** tab (**Member Characteristics** dialog box).

It is recommended to place a PALL system per story, into different bays.

This small building will be analysed with Pall members.



Project Configuration

Analysis Tab

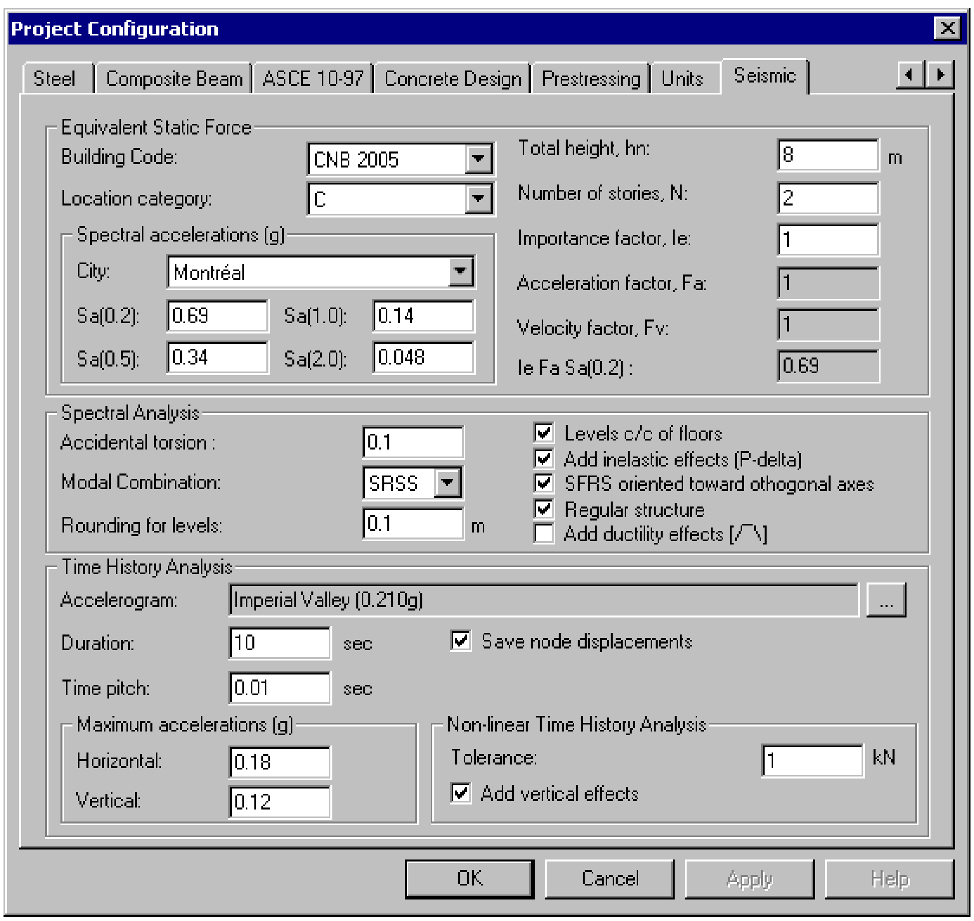
- Activate a non-linear type of analysis. Modify parameters for non-linear analysis, if required.

Seismic Tab

- Select an accelerogram and enter the duration. The time pitch is initialized when an accelerogram is selected. Enter the maximum horizontal and vertical accelerations and activate the option *Add vertical effects* in the non-linear time history section of the dialog box.

- Use a tolerance of 1 kN to facilitate convergence for this type of analysis.

Non-linear analyses are not calibrated. If you want to factor non-linear time history analysis, apply load factors through the **Load Combinations** spreadsheet.



Modelling Members as Friction Dampers

- Activate the Structure mode and select members that you want to model as elastoplastic members (friction dampers). Press the **Properties** icon to open the **Member Characteristics** dialog box.

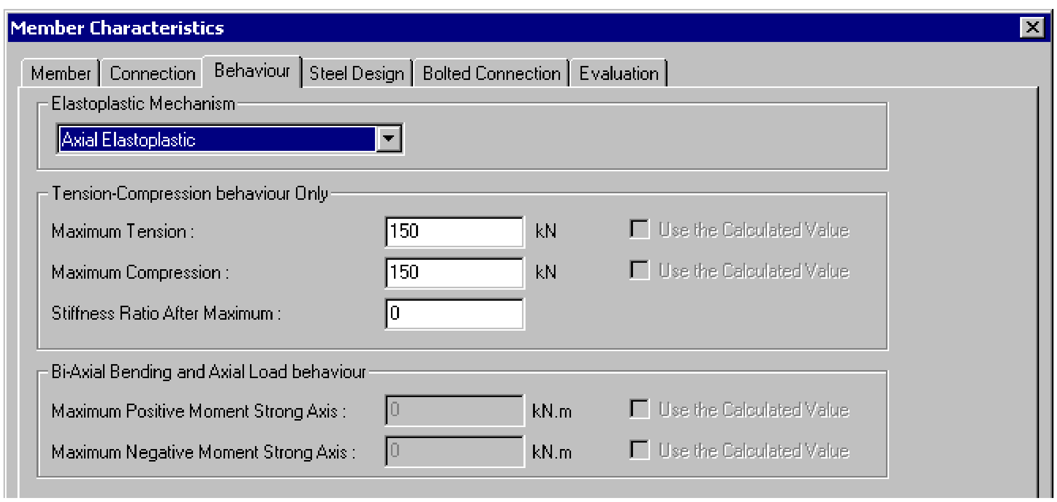
The Member tab

- In the **Member** tab, double-click in the *Behaviour* list box and select the option *Elastoplastic*.

The Behaviour tab

- Go to the **Behaviour** tab, select an *axial elastoplastic* mechanism and enter the maximum tension and compression that they will absorb.

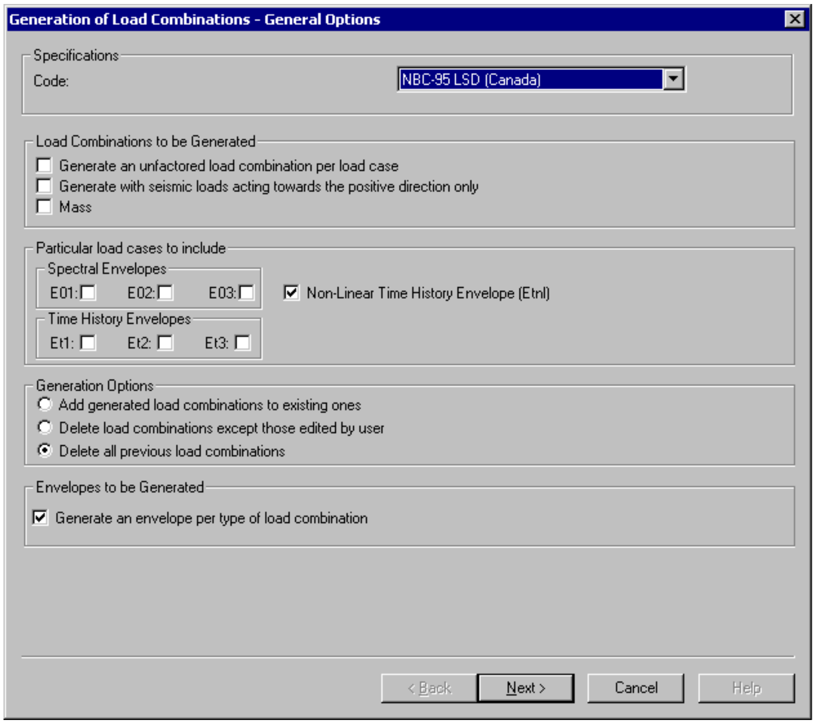
  When elastoplastic members will attain this value, they will enter into plastic phase. You can specify the stiffness ratio for elastoplastic members when they will reach maximum plasticity.

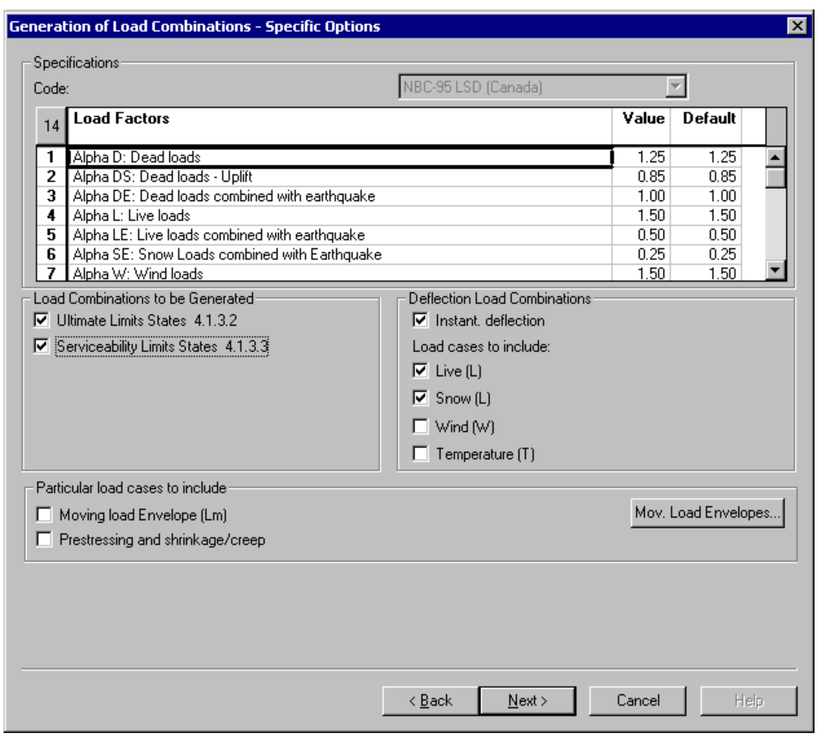


- Close the dialog box.

Load Combinations

Use the **Combination Generation Wizard** to generate load combinations. Include the non-linear time history envelope *Etnl*.





Click the *Next* button and then, the *Finish* button.

Generated Load Combinations

Load Combinations

Load Combinations | Load Factors

7	Number	Status	Definition
1	DE3	Ultimate	1.00D+1.00Etnl
2	DL1	Ultimate	1.25D+1.50S01+1.50Lx
3	DL2	Ultimate	0.85D+1.50S01+1.50Lx
4	DLE4	Ultimate	1.00D+0.50S01+1.00Etnl+0.50Lx
5	DL6	Service	1.00D+1.00S01+1.00Lx
6	L7	Instant. Deflection	1.00S01+1.00Lx
7	Mass_5	Mass	Mass

Modal Analysis

You are ready to run a modal analysis. Press this icon and select the *Mass* load combination in the **Modal Analysis** dialog box.

This modal analysis is a standard one and will consider all members as elastic members.

Non-Linear Seismic Direction

- When the modal analysis is completed, select the **Non-Linear Seismic Directions** spreadsheet (**Loads** menu). For each elastoplastic load combination, enter a value of 1.0 (meaning 100% of modal contribution acting in this direction) for the direction that you want to study.

Non Linear Seismic Directions Spreadsheet

2	Number	Dir. x	Dir. y	Dir. z	Vdyn. kN	Modal M/M %
1	DE3	1.00	0.00	0.00	0.00	0.00
2	DLE4	1.00	0.00	0.00	0.00	0.00

The last columns will be completed when the non-linear time history analysis will be done.

Studying the Second Seismic Direction

If elastoplastic members are modeled in the two directions of a building, you can analyse both directions in one analysis depending on the computer memory and the structure complexity (a lot of nodes).

If your project is simple:

If your project is quite simple, create additional elastoplastic load combinations. (Use the **Duplicate** function in contextual menu, to copy a load combination along with the corresponding load factors). The second seismic direction will be assigned to these load combinations, as shown below.

Non Linear Seismic Directions Spreadsheet

4	Number	Dir. x	Dir. y	Dir. z	Vdyn. kN	Torsion M kN.m	Modal M/M %
1	DE3x	1.00	0.00	0.00	0.00	0.00	0.00
2	DE3z	0.00	0.00	1.00	0.00	0.00	0.00
3	DLE4x	1.00	0.00	0.00	0.00	0.00	0.00
4	DLE4z	0.00	0.00	1.00	0.00	0.00	0.00

If your project is big and complex:

To save time and computer memory, do as follows:

- Run the non-linear time history analysis for one seismic direction only. Consult the results and rename the file.
- Go to the **Non-linear Seismic Directions** spreadsheet and change the direction of analysis.
- Select some nodes and launch the non-linear time history analysis right away. Do not launch the modal analysis or the non-linear static analysis because results will be lost!

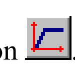
Non-Linear Static Analysis

- Run the non-linear static analysis by clicking on this icon .

Selection of Nodes

- Select a few nodes for which you want to look at results. This type of analysis can be very long.

Non-Linear Time History Analysis

- Launch the non-linear time history analysis by clicking this icon .

Note: The modal analysis that will be automatically launch is not the same as thr first one. This modal analysis includes members that behave linearly only. During analysis, the calculated forces in members that have a non-linear behaviour will be added to time history forces.

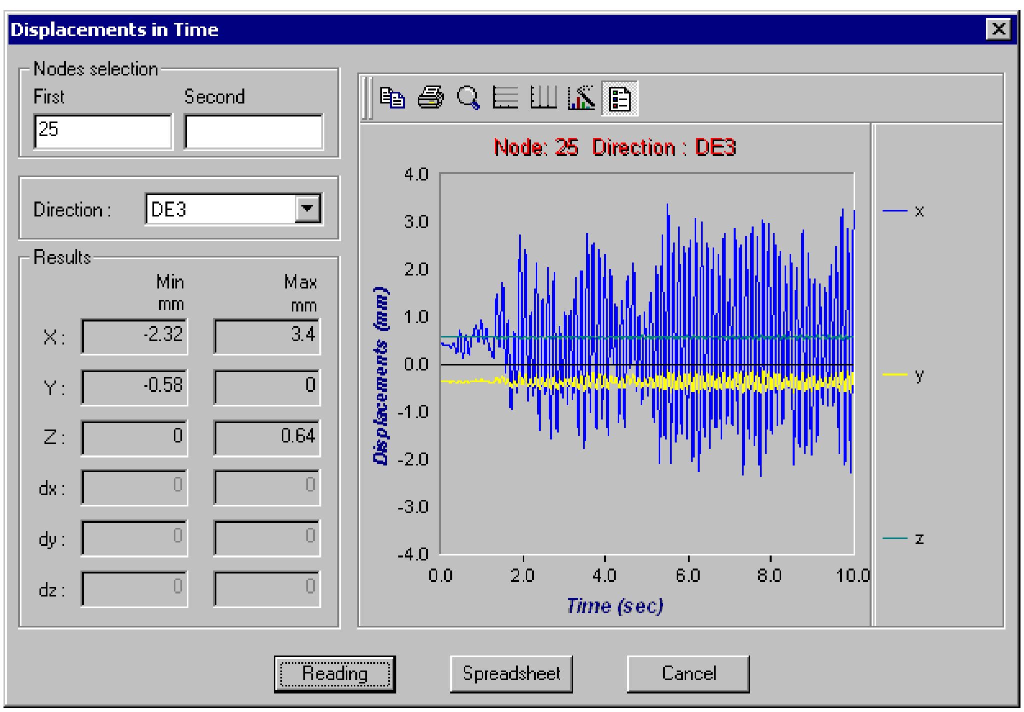
Results

The Load Combination mode is activated

- Select an elastoplastic load combination on Activation toolbar.
- Display node numbers through the **View Options** dialog box.
- Go to **Results / Time History.** Select **Node Displacements in Time**, **Reactions in Time**, **Forces in Time**, or **Forces and Deflections.**

Node Displacements in Time

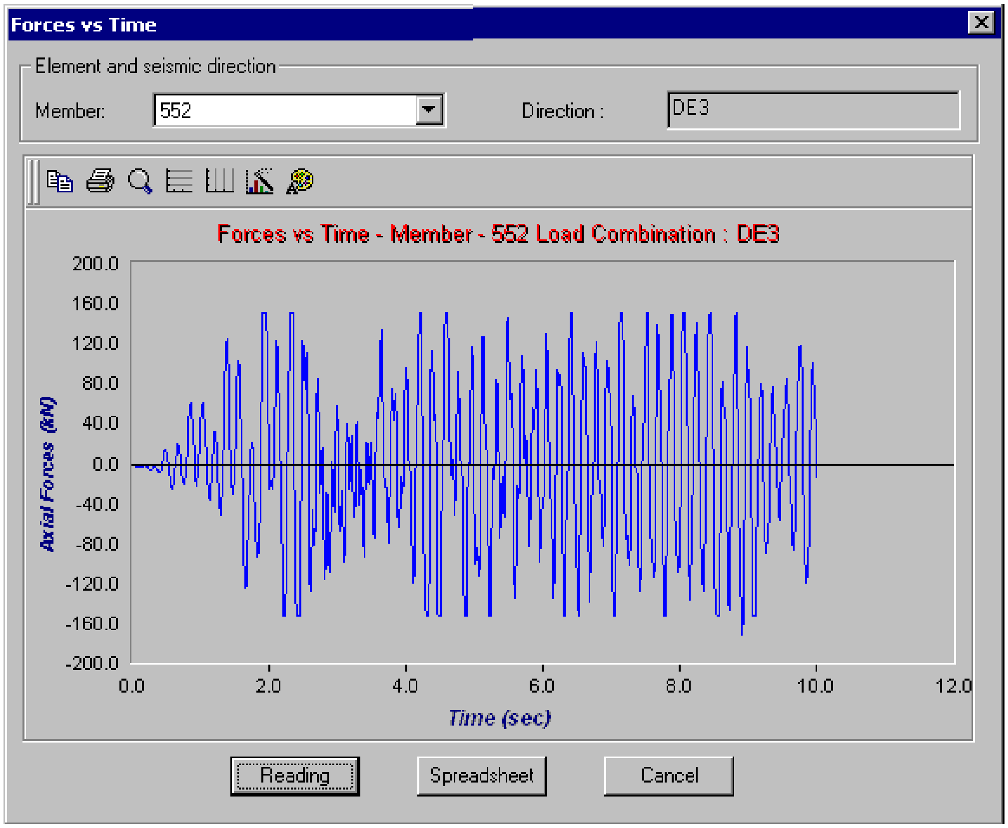
This function allows looking at node displacements in time or differential displacements between two nodes. Enter a node number or two and select a seismic direction in the *Direction* list box. Press the "Reading" button.



Forces in Time

Use this function to visualize forces vs time for elastoplastic members.

- Activate an elastoplastic load combination on Activation toolbar.
- Select a member number in the drop-down list box. Click the "Reading" button.

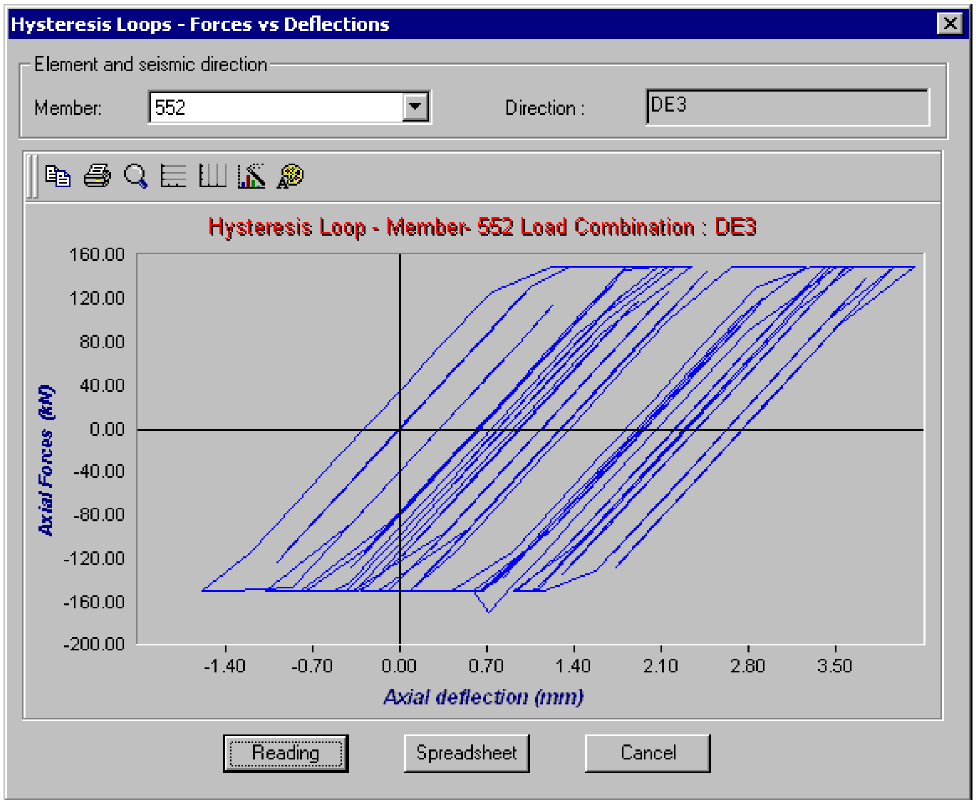


Forces and Deflections - Hysteresis Loops

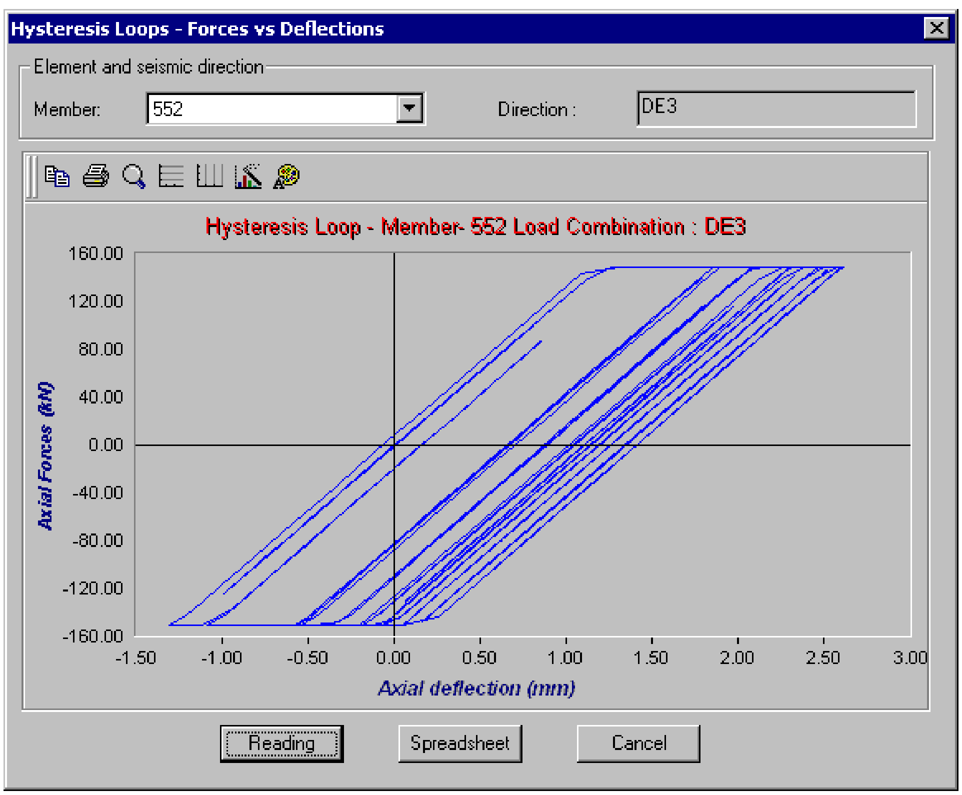
A hysteresis loop is a cyclic curve that represents force-deflection or moment-rotation and defines the elastic behaviour for an element or structural system. (Ref. *Éléments de génie parasismique et de calcul dynamique des structures*, André Filiatrault, 1996)

Use this function to visualize member forces and deflections (hysteresis loop) in time.

- Select an elastoplastic load combination on Activation toolbar.
- Choose an elastoplastic member in the drop-down list box. Click the "Reading" button.



If discontinuities appear in the diagram, reduce the time pitch in the **Seismic** tab and launch the analyses again. We reduced it to 0.005sec:



Summary of Procedure

- Run a modal and spectral analysis of the structure and study the behaviour of bracings (maximum tension and compression). Choose the location of PALL systems (friction dampers) in the structure.
- **Project Configuration** dialog box:
  - Select the **Analysis** tab and specify a non-linear analysis.
  - Select the **Seismic** tab and choose an accelerogram. Complete the required parameters.
- Select the members that you want to model as elastoplastic members. Open the **Member Characteristics** dialog box.
  - In the **Member** tab, select an elastoplastic behaviour.
  - Go to the **Behaviour** tab and specify the maximum tension and compression forces that elastoplastic members will absorb.
- Use the **Combination Generation Wizard** to create load combinations. Include envelope *Etnl* in the generation.
- Run a modal analysis.
- Select the **Non-Linear Seismic Directions** spreadsheet and specify the seismic direction that you want to study (for each elastoplastic load combination).
- Run a non-linear static analysis**.**
- **Select nodes** before launching the non-linear time history analysis. If you don't select nodes, all nodes will be analysed and it will take too much time and computer memory.
- Run a non-linear time history analysis**.**
- Select an elastoplastic load combination on Activation toolbar.
- Go to **Results / Time History** menu, and select one of available graphs.

EXAMPLE 10

General Dynamic Analysis

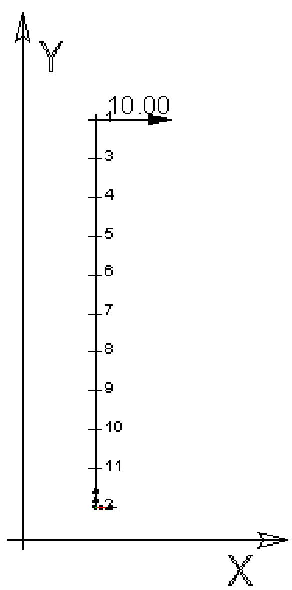
Transient Analysis

Two simple examples will show all the required steps to model and execute a general dynamic analysis. Different impact loads will be applied at the top of a 6.5 metre height steel column.

To get a sufficient number of vibration modes and frequencies, we split this column into 10 pieces. The more vibration modes you get, the more accurate the analysis and results will be.

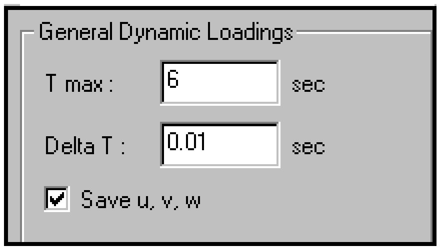
Example 1 – 1 Impact Load

A 10 kN dynamic load will be applied at the top of this column, towards the positive global x-axis.



Project Configuration

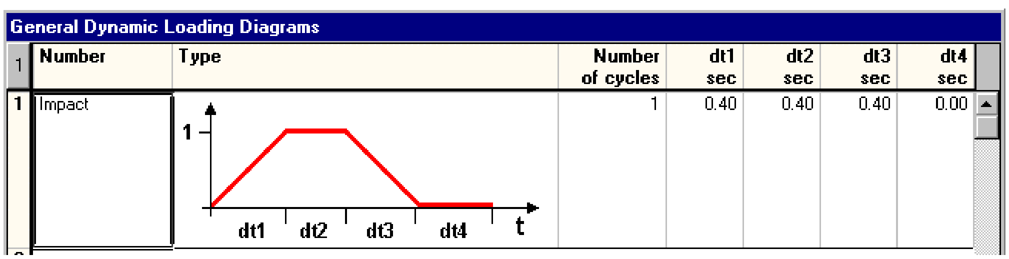
- Enter the following parameters in the **Analysis** tab of **Project Configuration** dialog box:



General Dynamic Load Diagram

VisualDesign has unitary dynamic load diagrams that are available in the **Loads** menu.

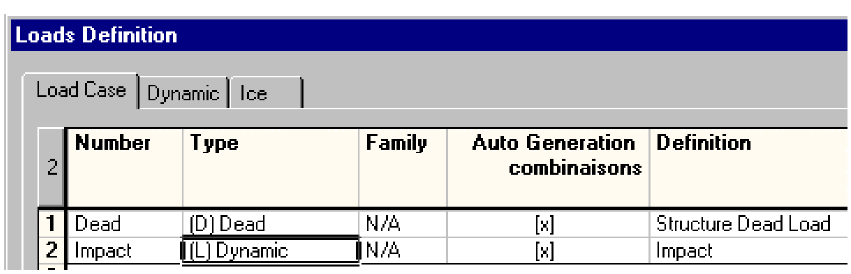
- Open the **General Dynamic Loads** spreadsheet and insert a line. Give a name to this load diagram. Double click in the "Type" column and select a loading diagram among the list box. Enter the number of cycles for this dynamic loading and specify *dti* intervals.



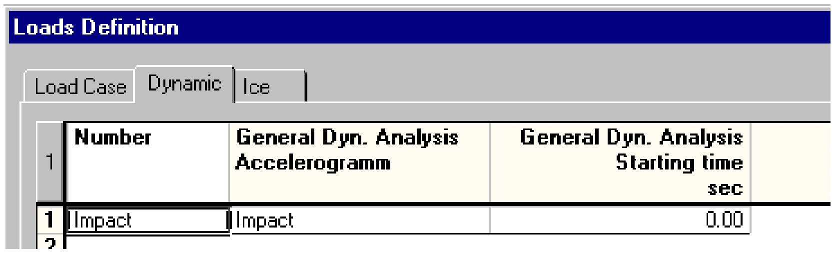
- Press OK.

Load Definitions Spreadsheet

- Select the **Loads Definition** spreadsheet in **Loads** menu and insert a line. Define the dynamic load case by selecting a *Dynamic* type of load in the "Type" column.



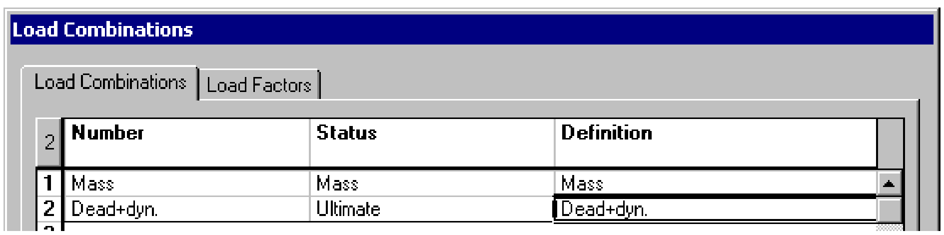
- Go to the **Dynamic** tab. Double-click in the *Accelerogram* cell and select the name of the dynamic load diagram.



There is only one dynamic load diagram; therefore the starting time is not important. This parameter is useful when dynamic loads are applied at different time or in different directions, or both.

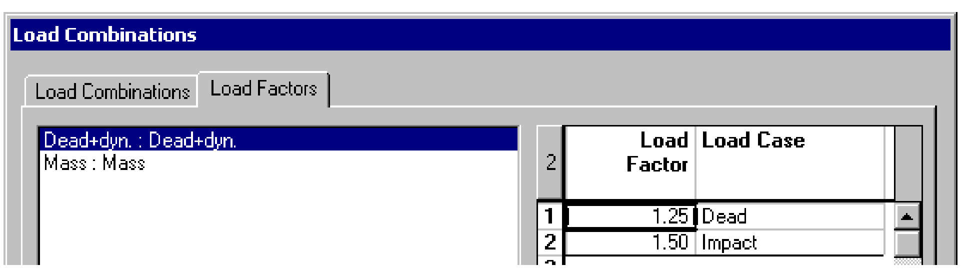
Load Combinations

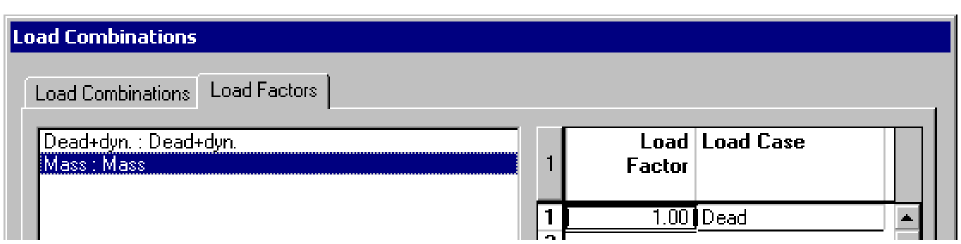
- Go to **Loads** menu and select **Load Combinations / Definition.** Insert two lines.



- The first load combination includes the structure dead load and the impact load. The second one is the *Mass* load combination, composed of the dead load only, which is required for the modal analysis.

- Go to the **Load Factors** tab. Enter load factors and select the appropriate load case in the drop-down list box.





- Press OK.

Applying the Impact Load

The dynamic load will be graphically applied at the top of the column.

- Activate the *Load Case* mode select the *Impact* load case title on Activation toolbar.

- Activate the Node element and double-click on the top node to open the **Forces on Nodes** spreadsheet.

- Enter a load of 10 kN (global x-axis) and click OK.

Forces on Nodes Spreadsheet

1	Fx kN	Fy kN	Fz kN	Mx kN.m	My kN.m	Mz kN.m
1	10.00	0.00	0.00	0.00	0.00	0.00
2						

Modal Analysis

- Launch the modal analysis by clicking on this icon .
  - In the **Modal Analysis** dialog box, select the "Mass" load combination. Ask for 15 vibration modes.
  - Launch the analysis by clicking the "Analyse" button.

Vibration Modes, Frequencies and Damping

- Go to **Results** / **Modal/Spectral/ Vibration Modes and Frequencies**. Look at the γx, γy and γz columns. They represent the modal contribution for each main direction. The modes that contribute the most are mode 1, mode 2, and mode 10.

Frequencies and Vibration Modes Spreadsheet

15	Mode	w rad/sec	f Hz	T sec	ξ %	Shape	Dir.x	Dir.y	Dir.z	γ.x	γ.y	γ.z
1	Mode 1	15.98	2.54	0.39	5.00	Other	1.00	-0.00	-0.00	12.42	-0.00	-0.00
2	Mode 2	54.30	8.64	0.12	5.00	Other	0.00	0.00	1.00	0.00	0.00	12.45
3	Mode 3	98.78	15.72	0.06	5.00	Other	1.00	-0.00	-0.00	6.91	-0.00	-0.00
4	Mode 4	272.81	43.42	0.02	5.00	Other	1.00	-0.00	0.00	4.05	-0.00	0.00
5	Mode 5	323.47	51.48	0.02	5.00	Other	0.00	-0.00	-1.00	0.00	-0.00	-7.01
6	Mode 6	526.05	83.72	0.01	5.00	Other	1.00	-0.00	0.00	2.90	-0.00	0.00
7	Mode 7	576.61	91.77	0.01	5.00	Other	-1.00	0.00	-0.00	-0.00	0.00	-0.00
8	Mode 8	845.70	134.60	0.01	5.00	Other	0.00	0.00	1.00	0.00	0.00	4.14
9	Mode 9	853.12	135.78	0.01	5.00	Other	1.00	0.00	0.00	2.25	0.00	0.00
10	Mode 10	1216.28	193.58	0.01	5.00	Gravitational	0.00	-1.00	0.00	0.00	-14.28	0.00
11	Mode 11	1243.77	197.95	0.01	5.00	Other	-1.00	-0.00	-0.00	-1.82	-0.00	-0.00
12	Mode 12	1515.89	241.26	0.00	5.00	Other	-0.00	0.00	1.00	-0.00	0.00	2.93
13	Mode 13	1679.16	267.25	0.00	5.00	Other	1.00	0.00	-0.00	1.50	0.00	-0.00
14	Mode 14	1716.95	273.26	0.00	5.00	Other	0.20	-0.98	-0.00	0.00	-0.00	-0.00
15	Mode 15	2123.77	338.01	0.00	5.00	Other	-1.00	-0.00	-0.00	-1.22	-0.00	-0.00

Damping

The default damping is set to 5%. To modify the damping percentage, select the whole column and enter another value. (Use the **Replace** function of contextual menu.) Launch the modal analysis again.

Static Analysis (Time History)

- Select the loaded node (or select none) and launch a static analysis.

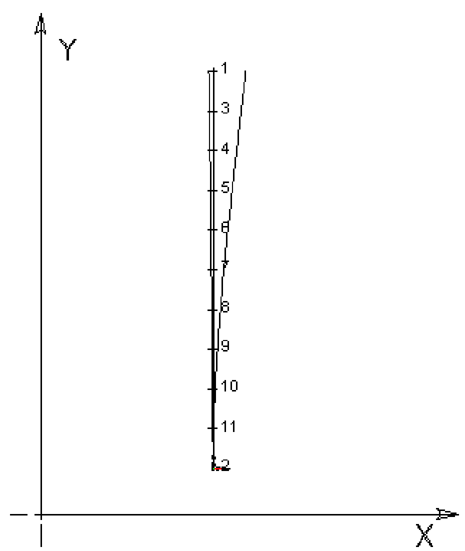
If you do not select any node(s) before launching the static analysis, VisualDesign™ will calculate displacements for all the nodes. In our example, we did not select any node.

VisualDesign™ will run a time history analysis according to the direction of impact load.

N.B. Time history analysis takes more time than other types of analysis. The longer the duration of load application is, the longer the analysis will be.

The *Load Combination* mode is automatically activated when the time history analysis is completed.

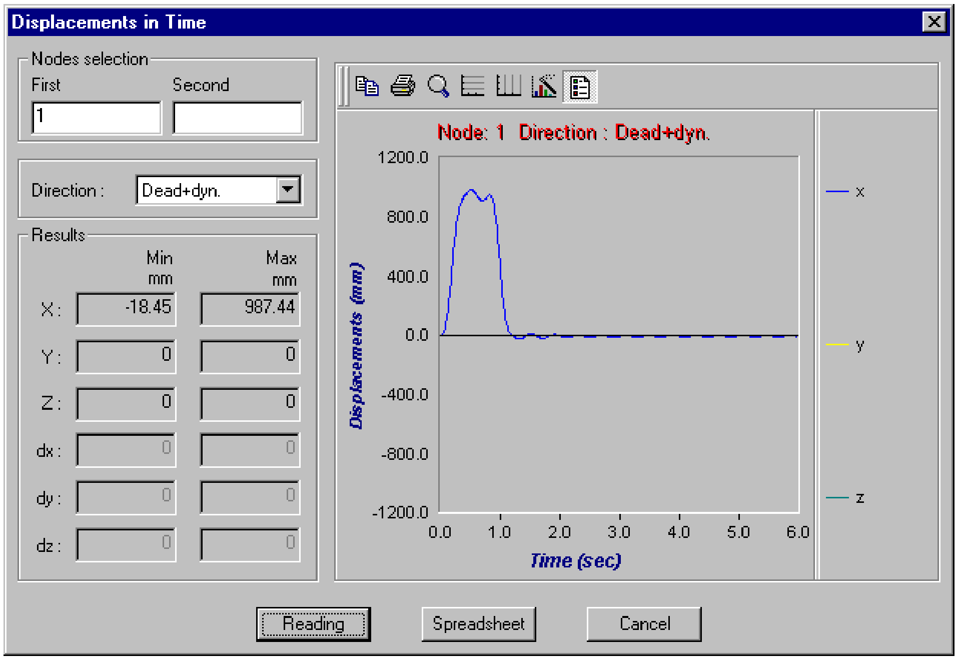
- Select the *Dyn+dead* load combination and look at the deflection.



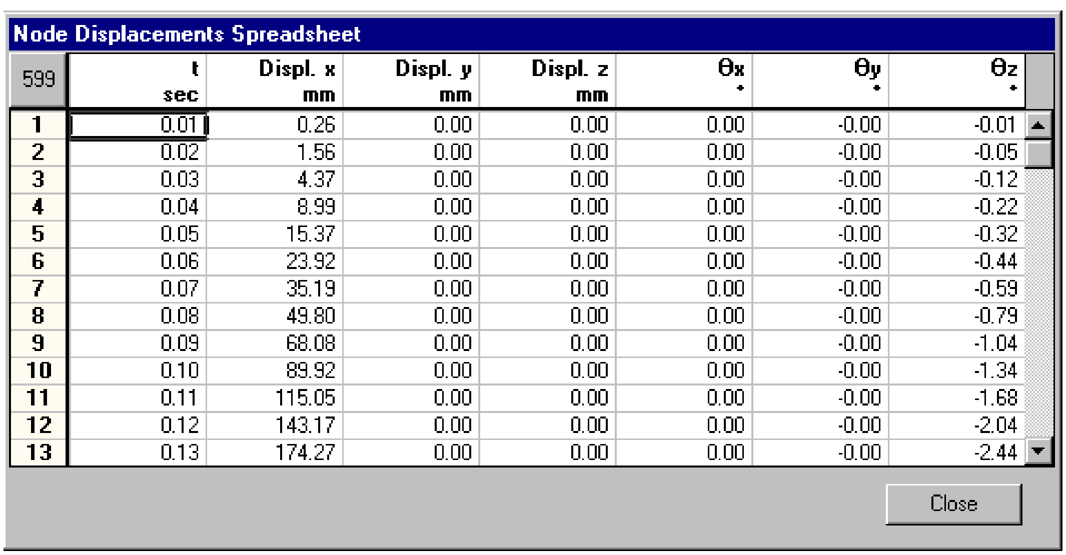
- Load Combination : Dead+dyn.

Time History Results - Node Displacements in Time

- Select the *Dyn+dead* load combination on Activation toolbar.
- Go to **Results** / **Time History/ Node Displacements in Time**. Enter node number 1 and press the "Reading" button.



Press the "Spreadsheet" button to look at results in the form of a spreadsheet.



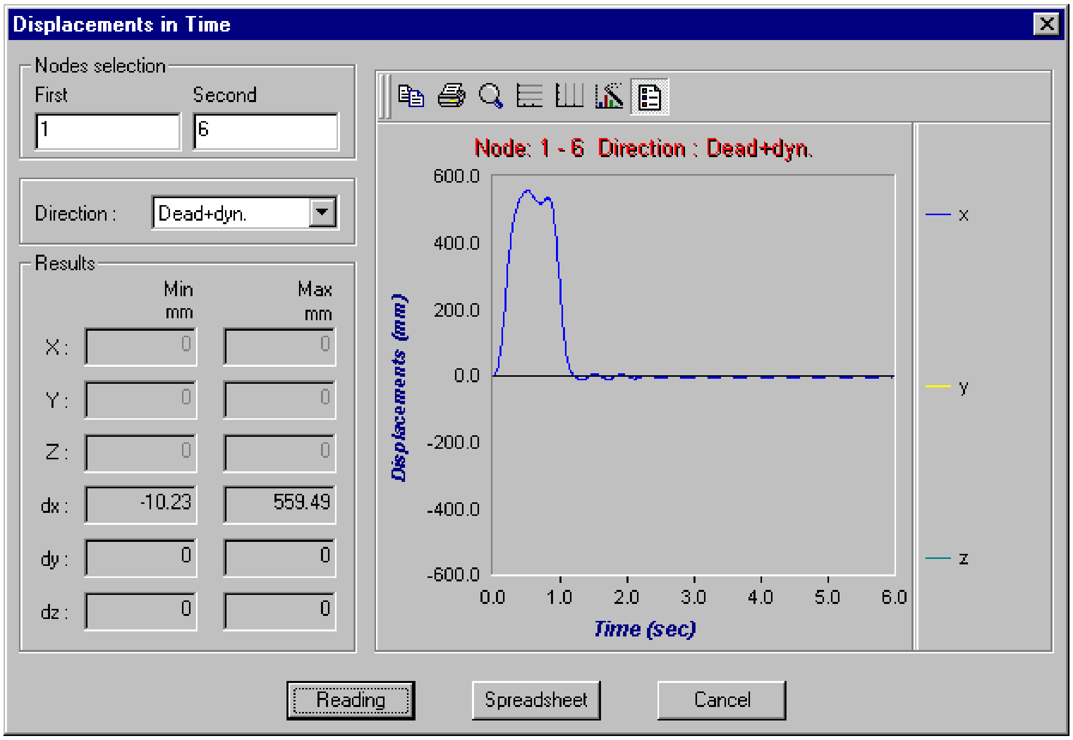
Node Displacements Spreadsheet

599	t sec	Displ. x mm	Displ. y mm	Displ. z mm	θx °	θy °	θz °
1	0.01	0.26	0.00	0.00	0.00	-0.00	-0.01
2	0.02	1.56	0.00	0.00	0.00	-0.00	-0.05
3	0.03	4.37	0.00	0.00	0.00	-0.00	-0.12
4	0.04	8.99	0.00	0.00	0.00	-0.00	-0.22
5	0.05	15.37	0.00	0.00	0.00	-0.00	-0.32
6	0.06	23.92	0.00	0.00	0.00	-0.00	-0.44
7	0.07	35.19	0.00	0.00	0.00	-0.00	-0.59
8	0.08	49.80	0.00	0.00	0.00	-0.00	-0.79
9	0.09	68.08	0.00	0.00	0.00	-0.00	-1.04
10	0.10	89.92	0.00	0.00	0.00	-0.00	-1.34
11	0.11	115.05	0.00	0.00	0.00	-0.00	-1.68
12	0.12	143.17	0.00	0.00	0.00	-0.00	-2.04
13	0.13	174.27	0.00	0.00	0.00	-0.00	-2.44

Close

Differential Displacements between Nodes

If you want to look at differential displacements between two nodes, they must have been selected before running the time history analysis. If this is the case, enter nodes number in appropriate fields and press the “Reading” button.



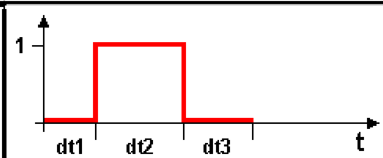
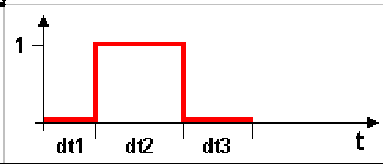
Example 2 – 2 Impact Loads

We are going to add a second impact load of 10 kN towards direction z. It will hit the top of the column 2 seconds after the first impact.

General Dynamic Loads spreadsheet

We will use another type of accelerogram. Enter the following parameters:

General Dynamic Loading Diagrams

2	Number	Type	Number of cycles	dt1 sec	dt2 sec	dt3 sec	dt4 sec
1	Impact_X		1	0.40	0.40	0.40	0.00
2	Impact_Z		1	0.40	0.40	0.40	0.00

Load Cases

- Add the *Impact_Z* load case and select a dynamic type of load.

Loads Definition

Load Case | Dynamic | Ice

3	Number	Type	Family	Stage	Tribu Red
1	Dead	(D) Dead	N/A	0	None
2	Impact_X	(L) Dynamic	N/A	0	None
3	Impact_Z	(L) Dynamic	N/A	0	None

- Specify the starting time for each impact load.

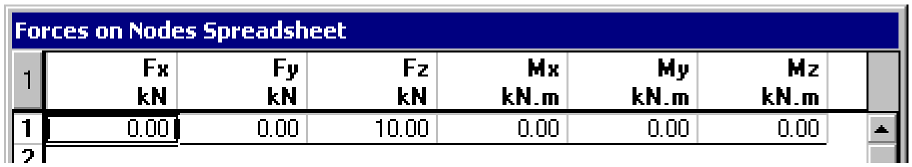
Loads Definition

Load Case | Dynamic | Ice

2	Number	General Dyn. Analysis Accelerogramm	General Dyn. Analysis Starting time sec
1	Impact_X	Impact_X	1.00
2	Impact_Z	Impact_Z	3.00

Apply Impact Loads

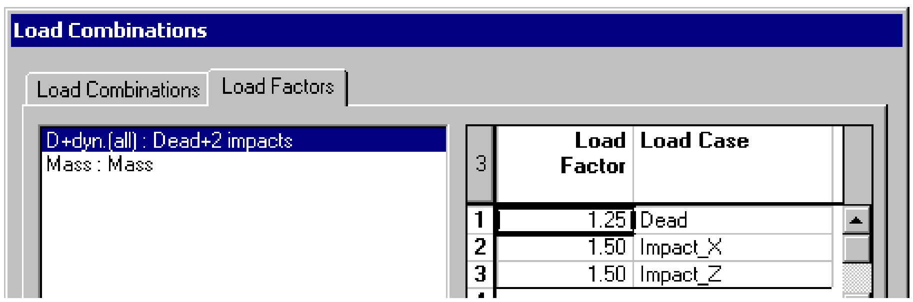
The dynamic load *Impact_X* is already entered. Activate the *Impact_Z* load on activation toolbar and double-click on node 1, at the top of column. Enter a force of 10 kN acting towards the positive global z-axis.



1	Fx kN	Fy kN	Fz kN	Mx kN.m	My kN.m	Mz kN.m
1	0.00	0.00	10.00	0.00	0.00	0.00
2						

Load Combinations

Include all impact loads in one load combination. Enter load factors and select load case titles in the drop-down list box.



3	Load Factor	Load Case
1	1.25	Dead
2	1.50	Impact_X
3	1.50	Impact_Z

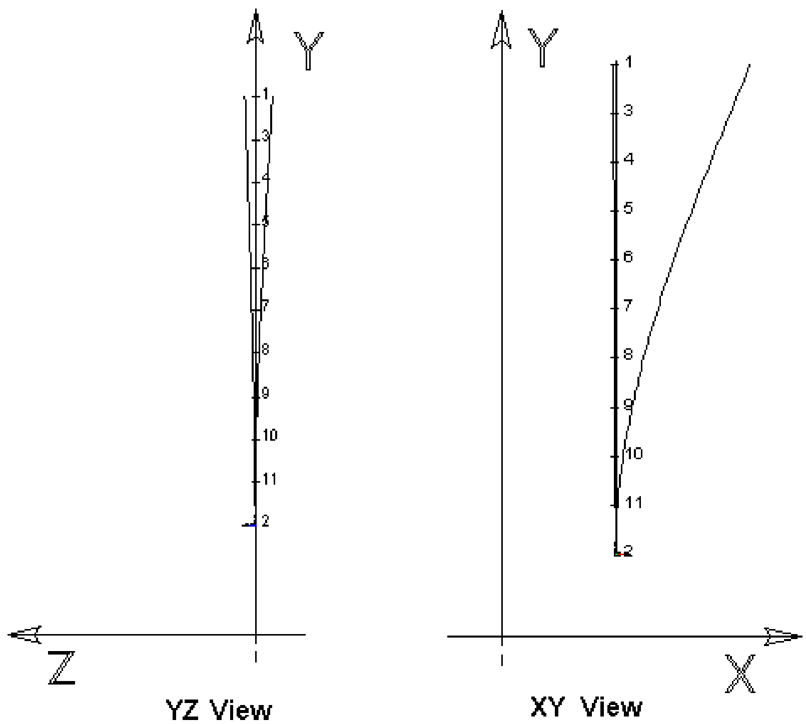
Modal Analysis

- Launch the modal analysis and specify 15 vibration modes in the **Modal Analysis** dialog box. Then, look at results in the **Frequencies and Vibration Modes** spreadsheet.

Static Analysis (Time History)

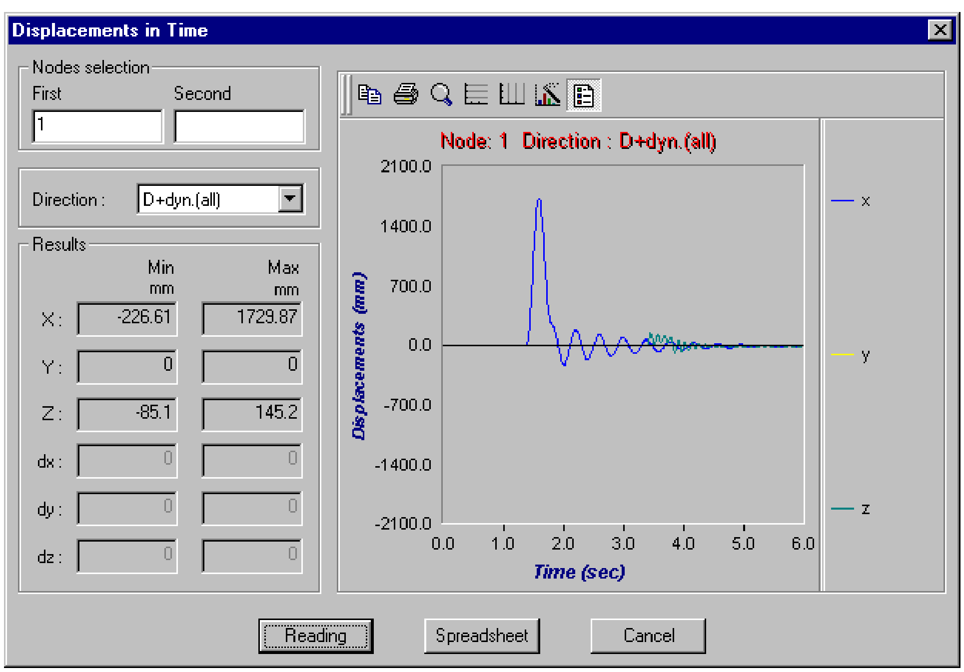
- Launch the static analysis.

- Activate the *Dead + 2 impacts* load combination.



Time History Results

- Go to **Results / Time History / Node Displacements in time**.
- Look at node 1 displacement in time (6 seconds).



EXAMPLE 11

2D Concrete Frame

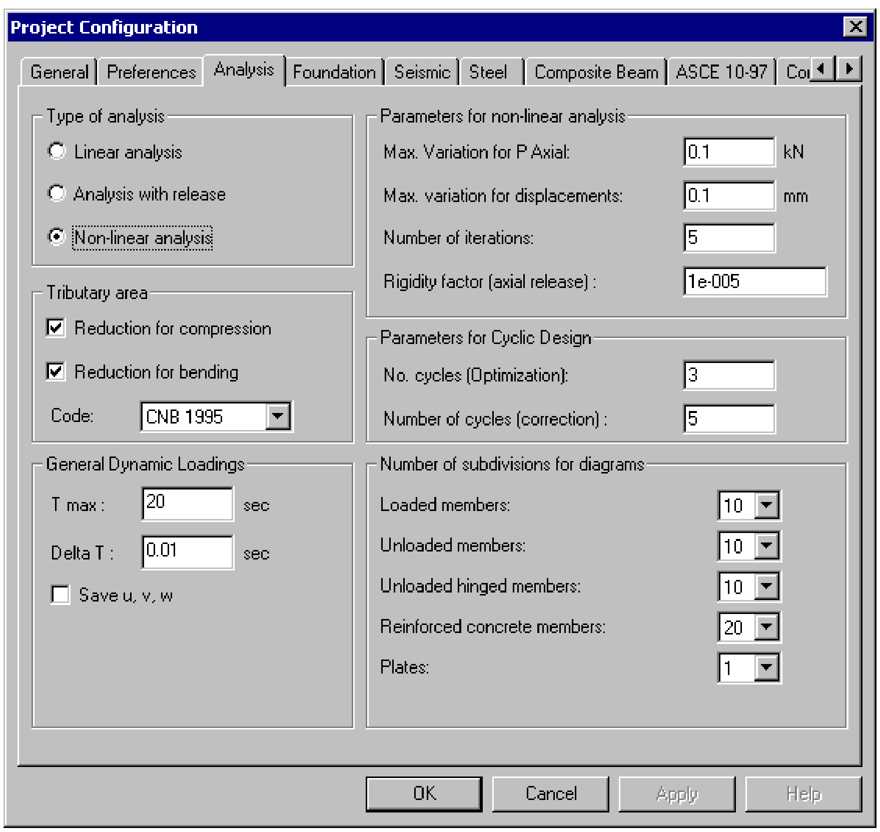
Design of a 2D-Concrete Frame

- Start VisualDesign. Click on the **New Project** icon. The Structure mode is automatically activated.

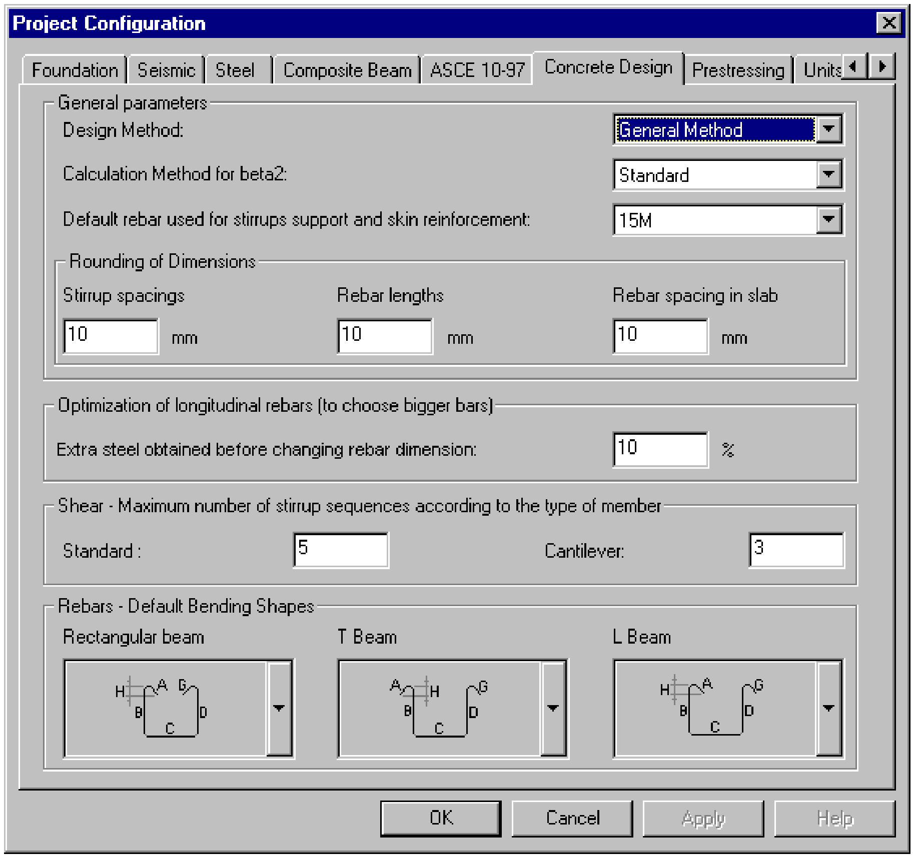
Project Configuration

- Open the **Project Configuration** dialog box (**File** menu) and go to the **Preferences** tab. Disable the *Dialog Box Display* option for nodes and members.

- Select the **Analysis** tab. Specify the number of subdivisions for concrete members. Forces, resistances, and deflections are calculated at each subdivision and are used when displaying diagrams.



- Go to the **Concrete** tab and select the *General Method* for concrete design.



- Close the dialog box.

Modelling

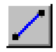
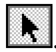
Nodes

- Select the **Nodes** spreadsheet in **Structure** menu and insert four lines. Enter node coordinates. Nodes 3 and 4 are support nodes (double-click in the "Type" cell and select "Support"). By default, the support degrees of freedom are all fixed.

Nodes Spreadsheet

4	Number	Type	Coord. X m	Coord. Y m	Coord. Z m	ID Master No.	Linke
1	1	Normal	10.00	3.00	0.00	0	n/a
2	2	Normal	0.00	3.00	0.00	0	n/a
3	3	Support	0.00	0.00	0.00	0	n/a
4	4	Support	10.00	0.00	0.00	0	n/a

Add Members

- Activate the **Member** icon  on Elements toolbar and the **Add** mode  on Cursor toolbar. To create a member, click on node i and j. Do the same to create other members. Then, exit the **Add** mode by clicking on the **Expanded Window** icon .

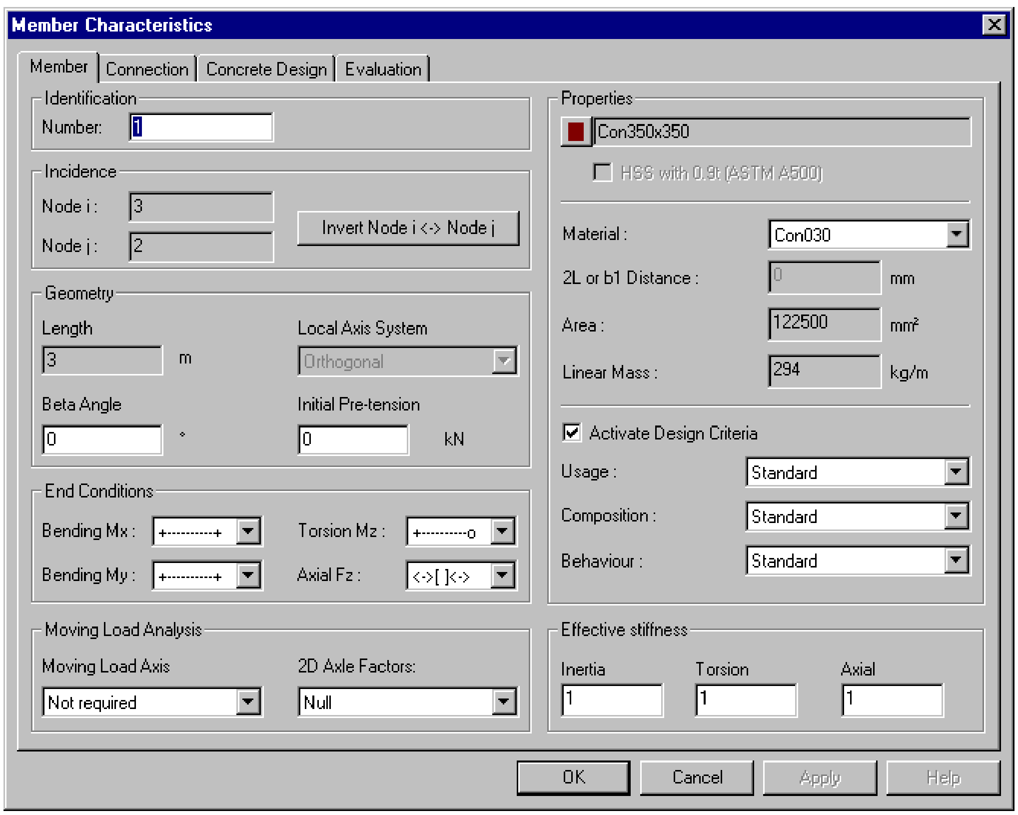
Column Properties

To define columns properties, keep the [Ctrl] key down while you select them.

Click the **Properties**  icon to open the **Member Characteristics** dialog box.

- Click the I beam icon to open the shape selection tree. Open the Concrete *root* and the Rectangular shapes *branch*. Choose a 350mm x 350mm section.

- Select a 30MPa concrete material. Activate design criteria.



- Click OK to close the dialog box.

Beam Properties

- Double click on the beam and choose a 350mm width x 650mm height section. Concrete material is 30 MPa. End conditions are fixed-fixed (symbol +-----+). Activate design criteria.

View Options

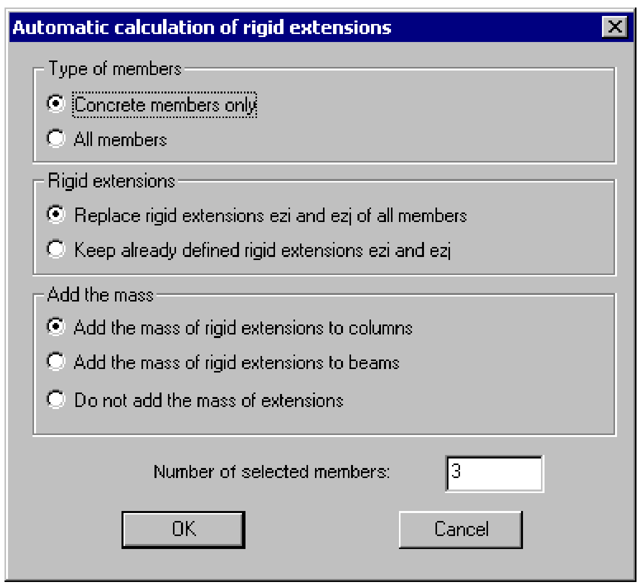
- Display shape outlines: Open the **View Options** dialog box and select the **Attributes** tab. Activate option *Shape outline* or *3D Display*. Press down the [Pg Up] key on your numerical keyboard to get an isometric view of the frame.

Rigid Extensions (Members)

Rigid extensions are required for concrete design. They must be modeled at the junction of transverse elements and both sides of a support for continuous beams.

We are going to use the tool that automatically generates rigid extensions.

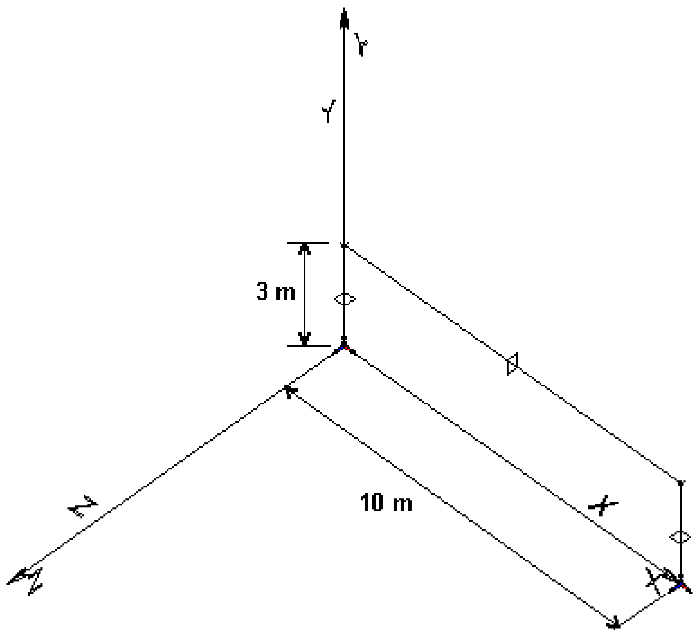
- Select all members and got to **Structure / Tool / Calculate Rigid Extensions**. Click OK.



Rigid extensions are located at the face of columns. Consult data in the **Connection** tab.

- Select all three members again and select the **Properties** function. Go to the **Concrete Design** tab. In the section "Design near maximum forces", select the option *At d or dv from face* for Vy. Press OK.

- At this stage, save your project by clicking on the save icon . Give it a name and choose a directory.



Concrete Specifications

- Go to **Structure** menu and select **Specifications/ Concrete**. Design will be done according to A23.3-95 Standard. Default values are OK.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Number	Code	Type of analysis	Maximum Capacity Factor %	Calcul. Method Mr/Vr Positive	Calcul. Method Mr/Vr Negative
1	A23.3-Design	CAN/CSA-A23.3-95	Design	100.00	Maximize Mr	Maximize Mr
2	A23.3-Verif.	CAN/CSA-A23.3-95	Verification	100.00	Maximize Mr	Maximize Mr
3	S6-00-Design	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
4	S6-00-Verif	CAN/CSA-S6-00	Verification	100.00	Maximize Mr	Maximize Mr
5	AASHTO-Design	AASHTO-LRFD-98 Beta	Design	100.00	Maximize Mr	Maximize Mr
6	AASHTO-Verif.	AASHTO-LRFD-98 Beta	Verification	100.00	Maximize Mr	Maximize Mr

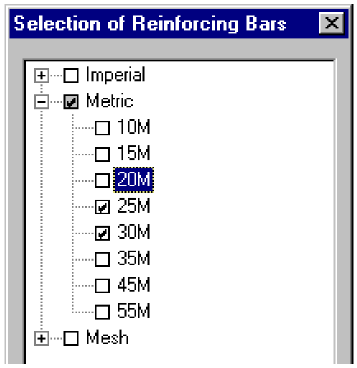
Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Longitudinal Optimization	Longitudinal Reinforcement Material	Selection of Longitidunal Rebar
1	Weight	G30.18-400R	20M 25M 30M
2	Weight	G30.18-400R	20M 25M 30M
3	Weight	G30.18-400R	20M 25M 30M
4	Weight	G30.18-400R	20M 25M 30M
5	Weight	G30.18-400R	# 7 # 8 # 9
6	Weight	G30.18-400R	# 7 # 8 # 9

Selection of Longitudinal Rebars

- Double click in this cell to open the *Rebar Selection* tree. Uncheck the *20M* box to withdraw this rebar from the list. Press OK and exit the spreadsheet.



- Go to the **Beam/Column/Joist** tab. Default values are OK. Close the spreadsheet.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Number	Maximum No. of Layers in Tension	Maximum No. of Layers in Compression	Transverse Optimization	Transverse Reinforcement Material	Selection of Transverse Rebar
1	A23.3-Design	4	4	Weight	G30.18-400R	10M 15M
2	A23.3-Verif.	4	4	Weight	G30.18-400R	10M 15M
3	S6-00-Design	4	4	Weight	G30.18-400R	10M 15M
4	S6-00-Verif	4	4	Weight	G30.18-400R	10M 15M
5	AASHTO-Design	4	4	Weight	G30.18-400R	# 4 # 5
6	AASHTO-Verif.	4	4	Weight	G30.18-400R	# 4 # 5

Continuous Systems

Three continuous systems have been automatically created by VisualDesign. Specification A23.3 must be assigned to them through the **Continuous Systems** spreadsheet, available in the **Structure** menu.

- Select the *A23.3-Design* specification for each continuous system and choose the type of exposure in the drop-down list box. Concrete covers will be considered as per code A23.3-95.

Continuous Systems Spreadsheet

3	Number	Specification	Type	Interaction	Description	Exposure Top	Top Cover mm
1	S_1	A23.3-Design	Beam/Column	Bending / Compression	S_1	Int. Exposure	40.00
2	S_2	A23.3-Design	Beam/Column	Bending	S_2	Int. Exposure	30.00
3	S_3	A23.3-Design	Beam/Column	Bending / Compression	S_3	Int. Exposure	40.00

N. B. When code S6-00 is assigned to continuous systems, concrete covers have to be specified.

Continuous Systems Spreadsheet

3	Exposure Bottom	Bottom Cover mm	Exposure Left	Left Cover mm	Exposure Right	Right Cover mm	Crack Control Top kN/m	Crack Control Bottom kN/m
1	Int. Exposure	40.00	Int. Exposure	40.00	Int. Exposure	40.00	30000.00	30000.00
2	Int. Exposure	30.00	Int. Exposure	30.00	Int. Exposure	30.00	30000.00	30000.00
3	Int. Exposure	40.00	Int. Exposure	40.00	Int. Exposure	40.00	30000.00	30000.00

- Specify the crack control parameters.
- Press OK.

Load Cases Definition

A live load and a wind load will be applied to the frame. VisualDesign automatically creates the dead load according to the density of chosen materials.

- Define load case titles and types in the **Load Definition** spreadsheet (**Loads / Load Cases / Definition**).
- Insert two lines and define load case titles and types according to the building code. Close the spreadsheet.

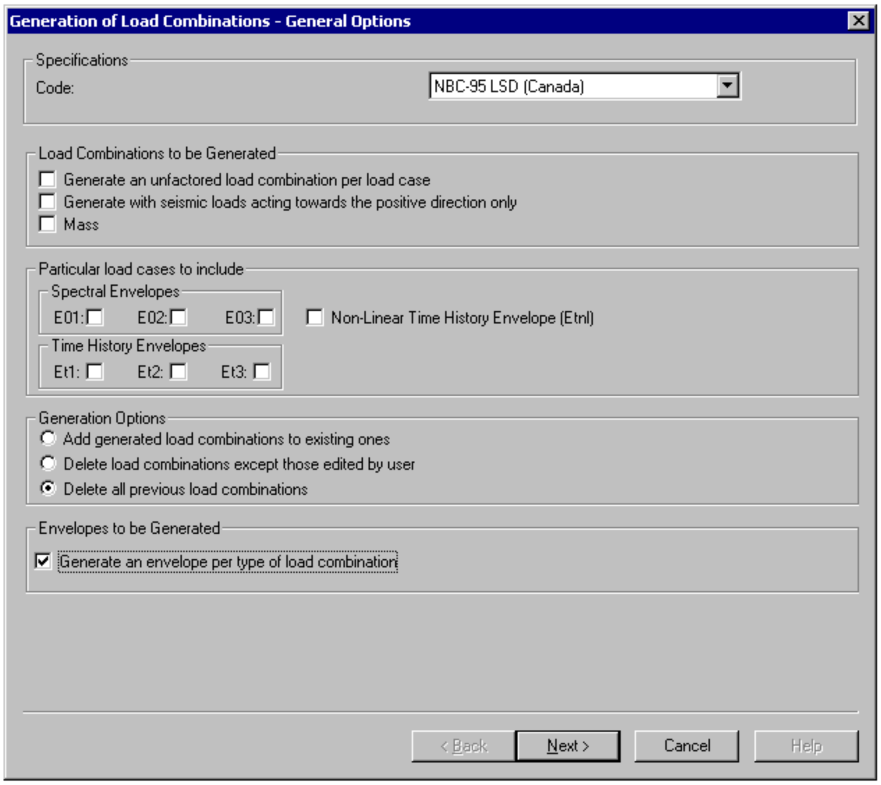
Loads Definition

Load Case | Dynamic | Ice

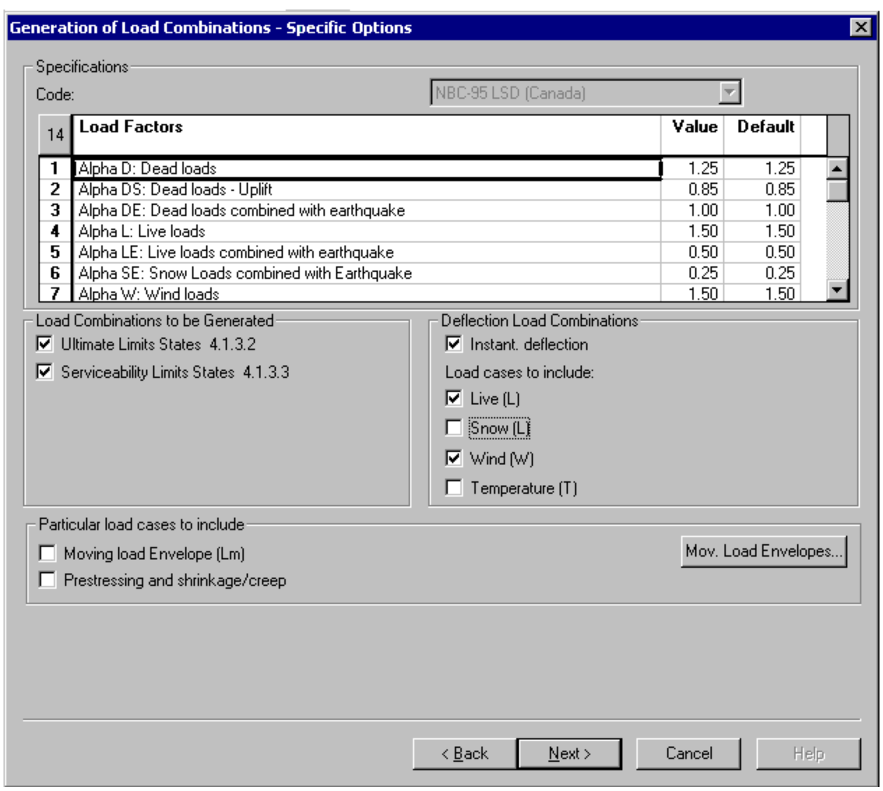
3	Number	Type	Family	Tributary Area Reduction	Tributary Area Overload kPa	Auto Gene combin
1	Dead	(D) Dead	N/A	None	0.00	[x]
2	Wind load	(W) Wind	N/A	None	0.00	[x]
3	Live load	(L) Live	N/A	None	0.00	[x]

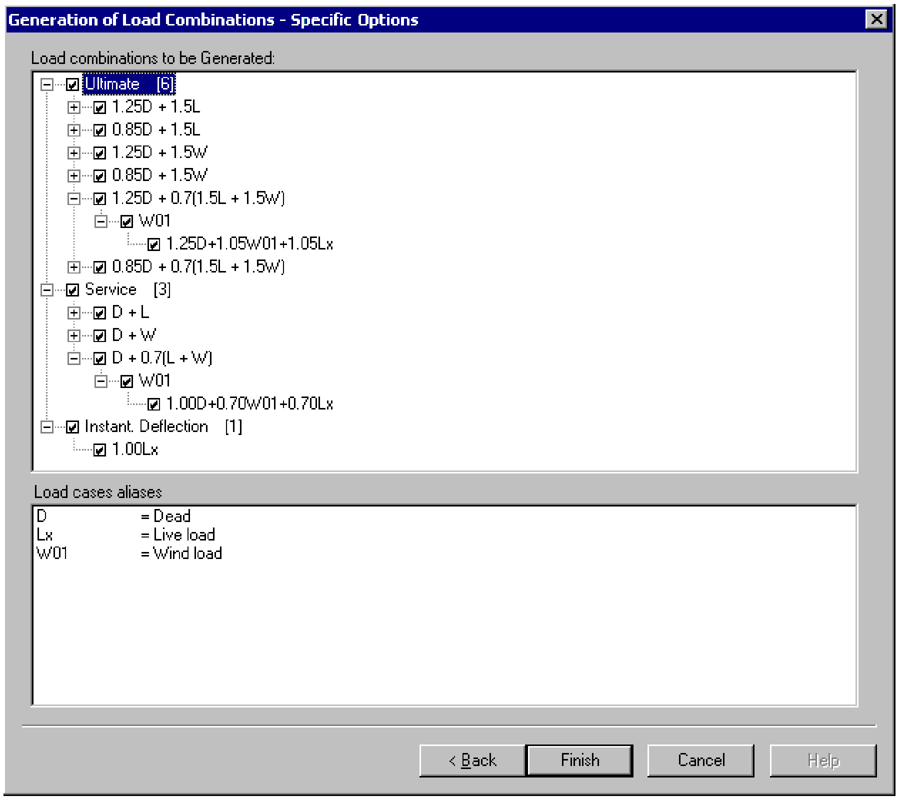
Generation of Load Combinations

- Go to **Loads** / **Load Combinations/ Generation Wizard**. The **General Options** page will appear on your screen. Select the CNBC code in the upper part of the dialog box. Click the "Next" button.

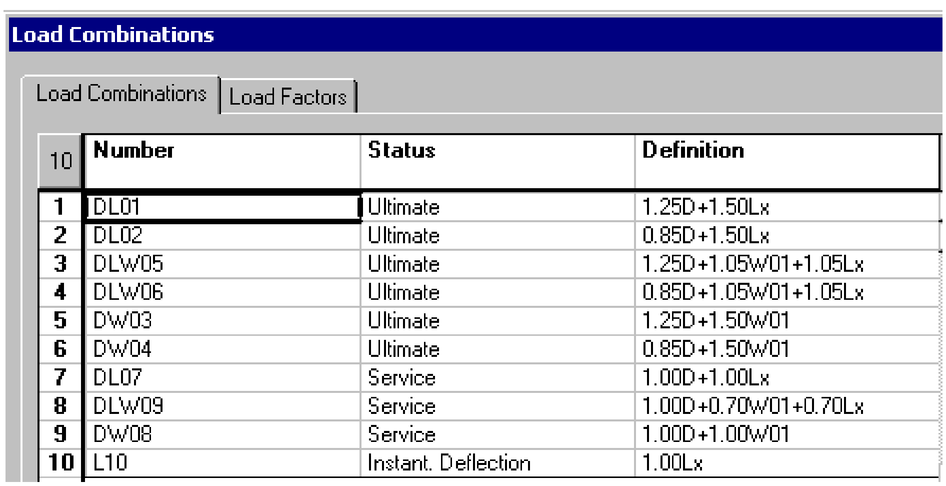


- In the **Specific Options** page, select load combinations to be generated. Click the "Next" button.





- Click the *Finish* button. The **Load Combinations** dialog box will automatically appear on screen.



10	Number	Status	Definition
1	DL01	Ultimate	1.25D+1.50Lx
2	DL02	Ultimate	0.85D+1.50Lx
3	DLW05	Ultimate	1.25D+1.05W01+1.05Lx
4	DLW06	Ultimate	0.85D+1.05W01+1.05Lx
5	DW03	Ultimate	1.25D+1.50W01
6	DW04	Ultimate	0.85D+1.50W01
7	DL07	Service	1.00D+1.00Lx
8	DLW09	Service	1.00D+0.70W01+0.70Lx
9	DW08	Service	1.00D+1.00W01
10	L10	Instant. Deflection	1.00Lx

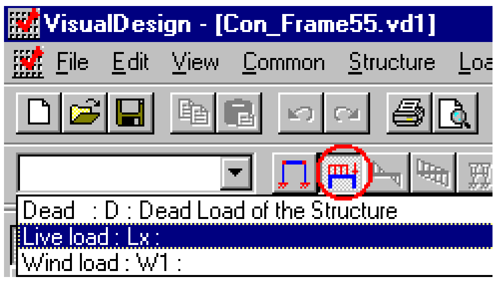
- If you prefer not to consider a load combination for a particular analysis, double click in the "Status" cell and change the status to *Not required.*

Application of Loads

A uniformly distributed live load will be applied on the beam. A triangular wind load will be applied to the first column.

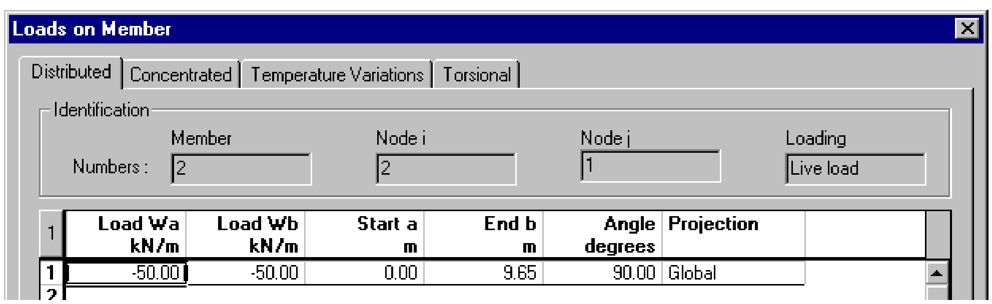
Live Load

- Activate the Loading mode on Activation toolbar and choose "Live load" in the drop-down list box.



- At the message "Save project", answer "Yes". Give a name to your file. The name of the current load (live load) will be posted in the bottom part of the screen.

- Double click on the beam. Insert a line in the **Distributed** tab. Double click in the *Load Wa* cell and enter -50. Do the same for *Load Wb*. Press OK. You will see the load diagram on your screen.



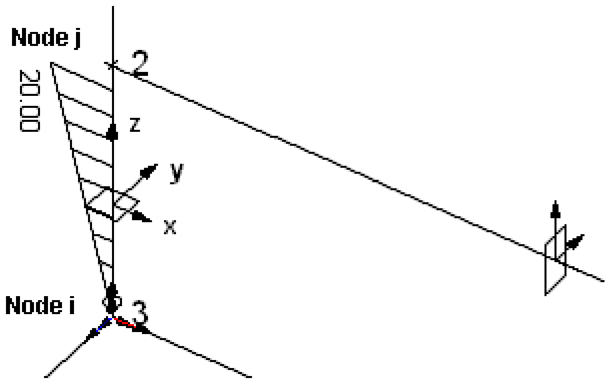
Remark. The calculated value for *End b* is equal to 9.65 m, which corresponds to the length of beam minus rigid extensions.

Wind Load

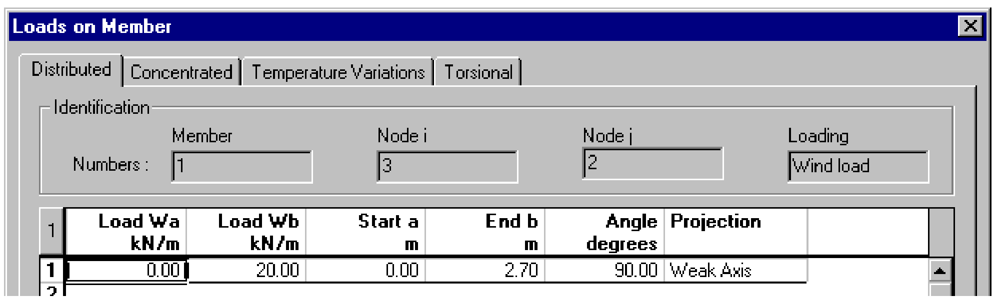
- Select "Wind load" on Activation toolbar.

- Open the **View Options** dialog box and display node numbers through the **View** tab. Go to **Attributes** tab and display member local axis system. We have to look at the column strong and weak axes to make sure that wind is properly applied.

Wind load will be null at the support, and equal to 20 kN/m at node #2, towards the positive local x-axis of the column. The local y-axis corresponds to the column weak axis. The local z-axis is always pointing towards node j.



- Double click on the column and insert a line in the **Distributed** tab. *Load Wa* is applied to node i and *Load Wb*, to node j. Wind is projected on the weak axis at an angle of 90 degrees. Click OK.

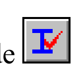


You are now ready to launch the concrete design.

Reinforced Concrete Design

- Press down the **Analysis and Design**  icon on Tools toolbar. The **Design** dialog box will appear on the screen. Three continuous systems will be optimized. Press the "Analyse" button. When the design is done, close the **Design** dialog box.

Design Results for Continuous Systems

- Stay in VisualDesign main window. Activate the *Design Results* mode  and go to **Results** / **Structure Design / Concrete.** Consult calculated design loads for all continuous systems.

Design Results for Continuous Systems

Beam / Joist | Column | Shear Wall | Slab

1	Number	Design load Positive moment %	Design load Negative moment %	Design load Shear force %	Cracking Positive moment %	Cracking Negative moment %
1	S_2	98.91	98.14	96.61	59.73	59.73

- Select the **Column** tab.

Design Results for Continuous Systems

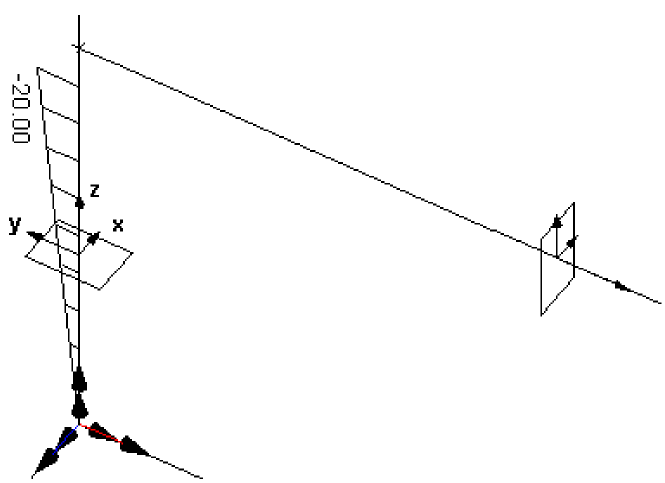
Beam / Joist | Column | Shear Wall | Slab

2	ID	Number	Design load Shear force Strong axis %	Design load Shear force Weak axis %	Design Load Interaction %	As/Ag max %
1	1	S_1	0.00	65.98	124.80	6.86
2	3	S_3	0.00	65.98	124.80	6.86

Columns are underdesigned (124.8%). We are going to modify the columns dimensions to 350mm x 500mm and launch the design again.

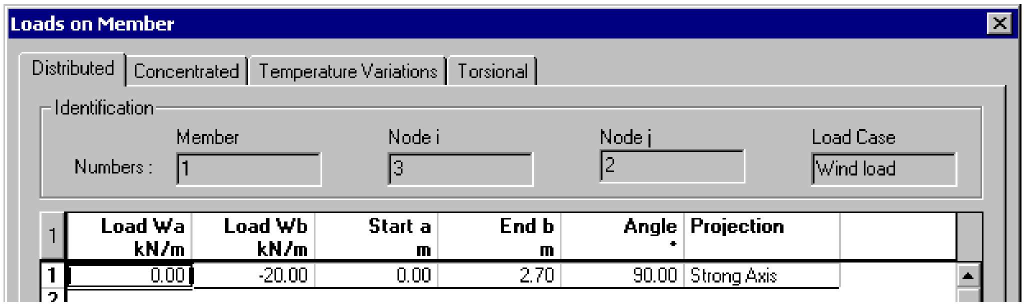
Modification of Columns Dimensions

- Activate the Structure mode, the "Member" icon, and select columns. Click the **Properties** icon. Select a 350mm x 500mm concrete section. Press OK.
- Display shape outlines and member local axis system (**Attributes** tab of **View Options**). We can see that columns are not well oriented.
- Select the columns again and open the **Member Characteristics** dialog box. Enter a beta angle of 90 degrees in the **Member** tab. Click OK.



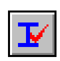
The wind load is now applied towards the negative direction of local y-axis and is projected on the column strong axis at an angle of 90 degrees.

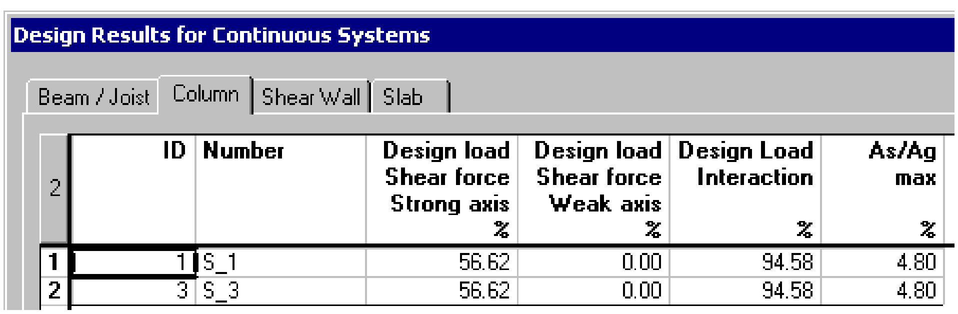
- Activate the Load Case Activation mode, and select the wind load. Double click on the first column. In the **Distributed** tab, enter a negative sign and change the load projection to *Strong Axis*.



- Close the dialog box.
- Launch a new concrete design.

Design Results for Columns

- Activate the Design Results mode  and go to **Results** / **Structure Design / Concrete.** Look at columns design loads.



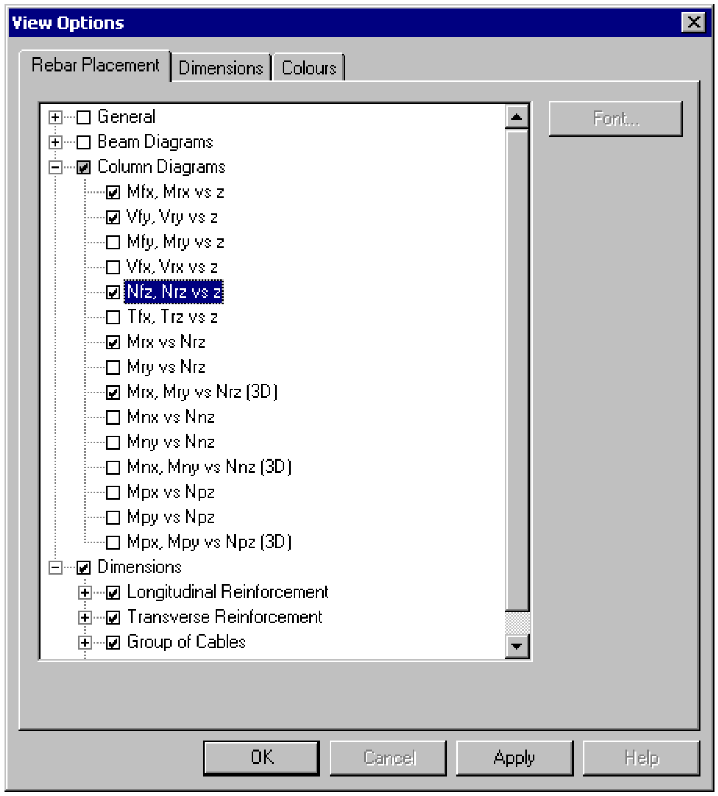
Columns are OK.

Rebar Placement for Columns

- Activate the *Rebar Placement* mode  and double click on the first column to open the *Rebar Placement* window. An elevation view of the column will appear on screen.

Display Rebars, Resistance & Force Diagrams

- Open the **View Options** dialog box and check the *Dimensions* root to display rebars. Expand the *Column diagrams* root. Check the boxes corresponding to the diagrams that you want to display on screen.



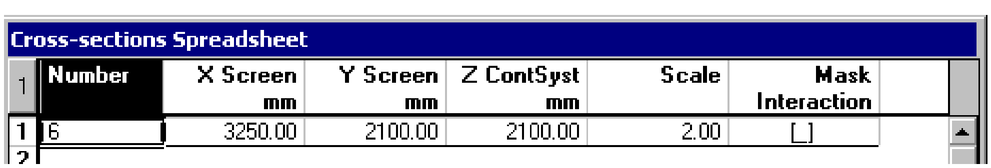
- Click OK and look at rebar placement and diagrams.

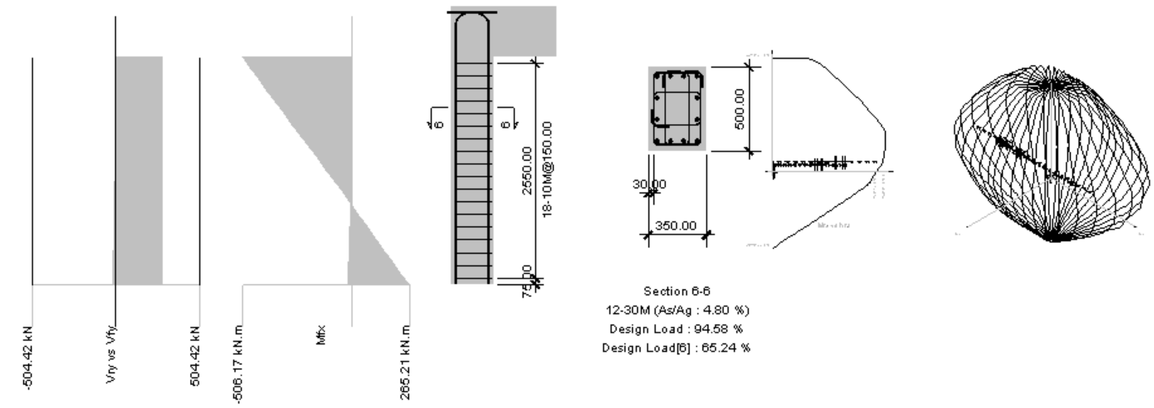
- Select the **Colours** tab to know the default colour for displayed forces and resistances.

Cross-sections

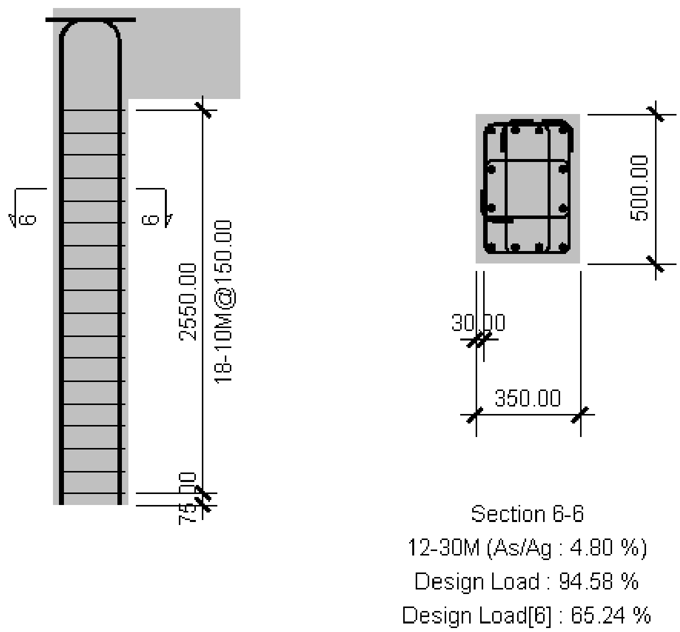
At least one cross-section must be created in order to display a 3D interaction curve and calculated design loads.

- Go to **Rebar Placement** menu and select **Automatic Generation of Cross-sections**. The Cross-sections spreadsheet will be appearing on screen. Click OK.





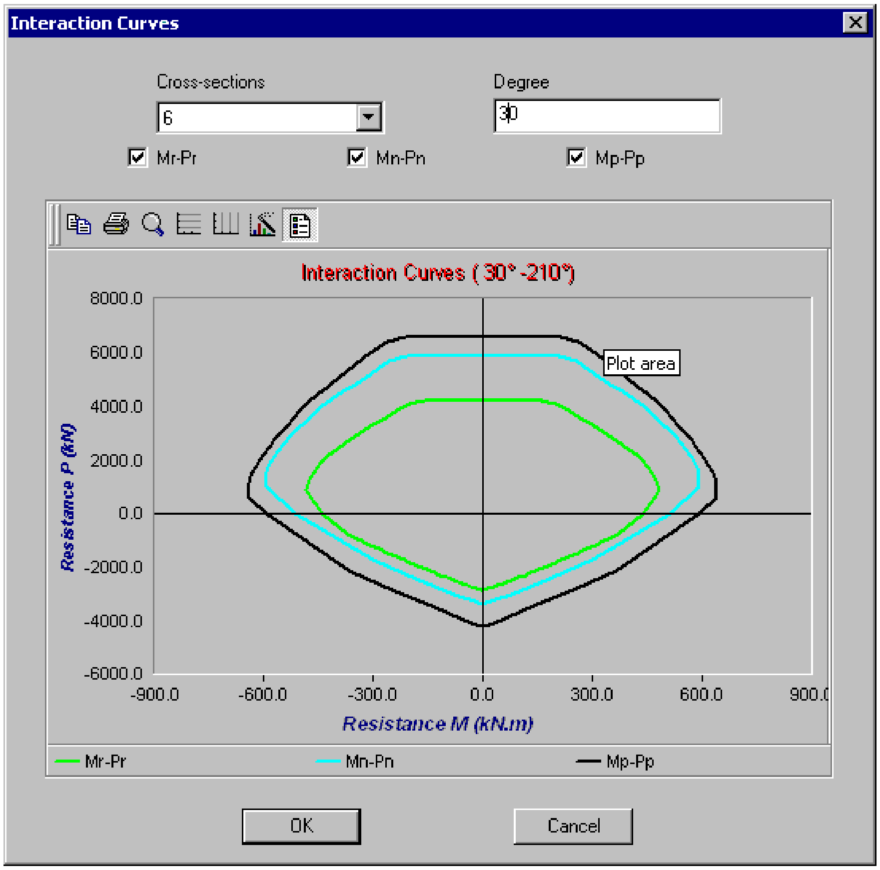
- Use the **Zoom+** and **Zoom Window** functions to have a better look at rebar placement. Click on the 3D interaction curve and use the keyboard arrows to rotate it.



We can see that the column has 12 main rebars of dimension 30M, from the bottom to the top. 18 stirrups of dimension 10M are 150mm spaced apart. The second design load [6] represents the design load at the location of cross-section no.6.

2D Interaction Curves

- Go to **Results / Interaction curves.** Look at a slice of 3D interaction curve. The values for resistance, nominal and probable forces can be displayed. An angle of cut can also be specified



General Results Spreadsheet

This spreadsheet includes all numerical values from calculated forces and resistances, for each subdivision along members that are part of the displayed continuous system. These values are also used when displaying diagrams. Yellow lines mean that some parameters do not respect the code requirements.

- Go to **Results / General Results**.

When designing a column, VisualDesign considers forces acting on the strong axis and weak axis.

General Results Spreadsheet - S_1

21	Member Number	Z mm	Vfy Max kN	Vfy Min kN	Design Load %	Vcy kN	Vsy kN	Vry kN	dy mm	dvy mm
1	1	0.00	285.58	-15.64	56.62	102.08	402.33	504.42	443.75	399.37
2	1	135.00	285.58	-15.54	56.62	102.08	402.33	504.42	443.75	399.37
3	1	270.00	285.58	-15.24	56.62	102.08	402.33	504.42	443.75	399.37
4	1	405.00	285.58	-14.73	56.62	102.08	402.33	504.42	443.75	399.37
5	1	540.00	285.58	-14.02	56.62	102.08	402.33	504.42	443.75	399.37
6	1	675.00	285.58	-13.11	56.62	102.08	402.33	504.42	443.75	399.37

General Results Spreadsheet - S_1

21	Vfx Max kN	Vfx Min kN	Design Load %	Vcx kN	Vsx kN	Vrx kN	dx mm	dvx mm
1	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
2	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
3	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
4	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
5	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
6	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37
7	0.00	0.00	0.00	102.08	260.79	362.87	293.75	264.37

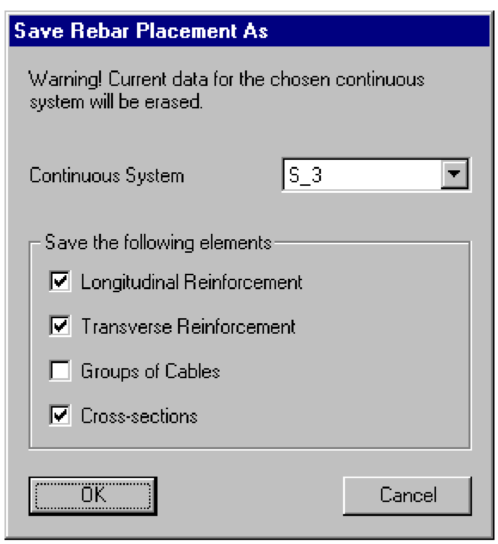
- Close the spreadsheet.

Copy Rebar Placement to the Other Column

Continuous systems must be identical to be allowed to copy reinforcement from one continuous system to another. Identical continuous systems must have the same geometry, the same rigid extension lengths, and local axis systems must point in the same direction.

The continuous system #S_1 corresponds to the first column. We want to copy the rebar placement to the second column, which is continuous system #S_3.

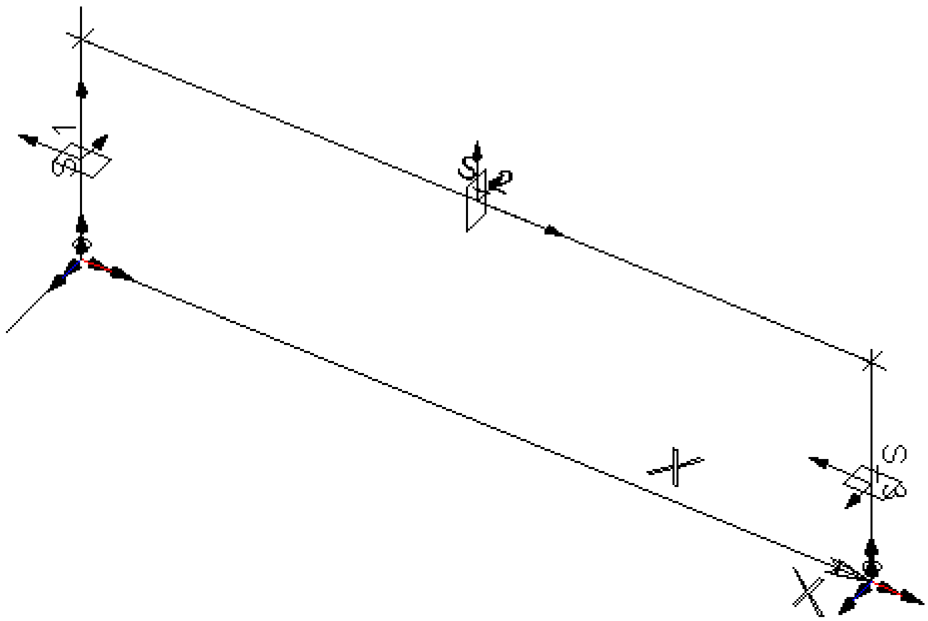
- In the *Rebar Placement* window, go to **File** / **Save Rebar Placement As**. Select the continuous system number that will be modified. Activate the boxes corresponding to elements you want to copy.



- Click OK.

The following message appears on screen:

Go back to VisualDesign main window and look carefully at the structural model. Display members' local axes system and check the length of rigid extensions.



The problem is the local axis system of the second column. To correct this, we will invert the node incidence for the second column.

- Activate the *Structure* mode, and double-click on member #3.
- In the **Member** tab, click the button ***Switch Node i ↔ Node j***. Close the dialog box.
- Activate the *Rebar Placement* mode, and double-click on continuous system #S_1 again.
- Go to **File** / **Save Rebar Placement As**. Select Continuous system #S_3 and activate appropriate options. Click OK. Close the *Rebar Placement* window.

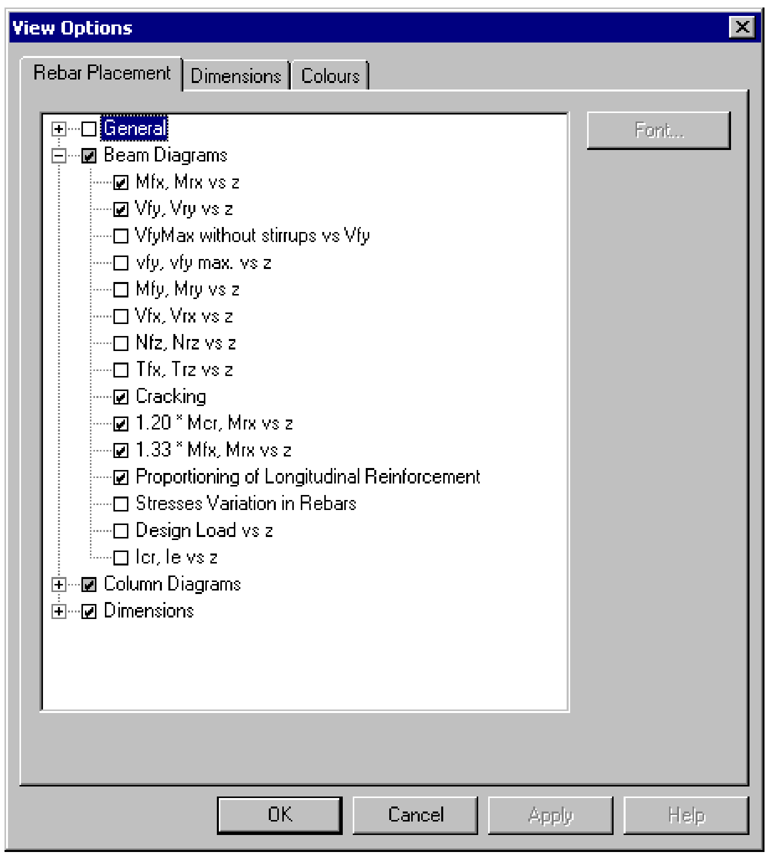
Rebar Placement for the Beam

Double click on the beam. The *Rebar Placement* window will open and display an elevation view of the beam.

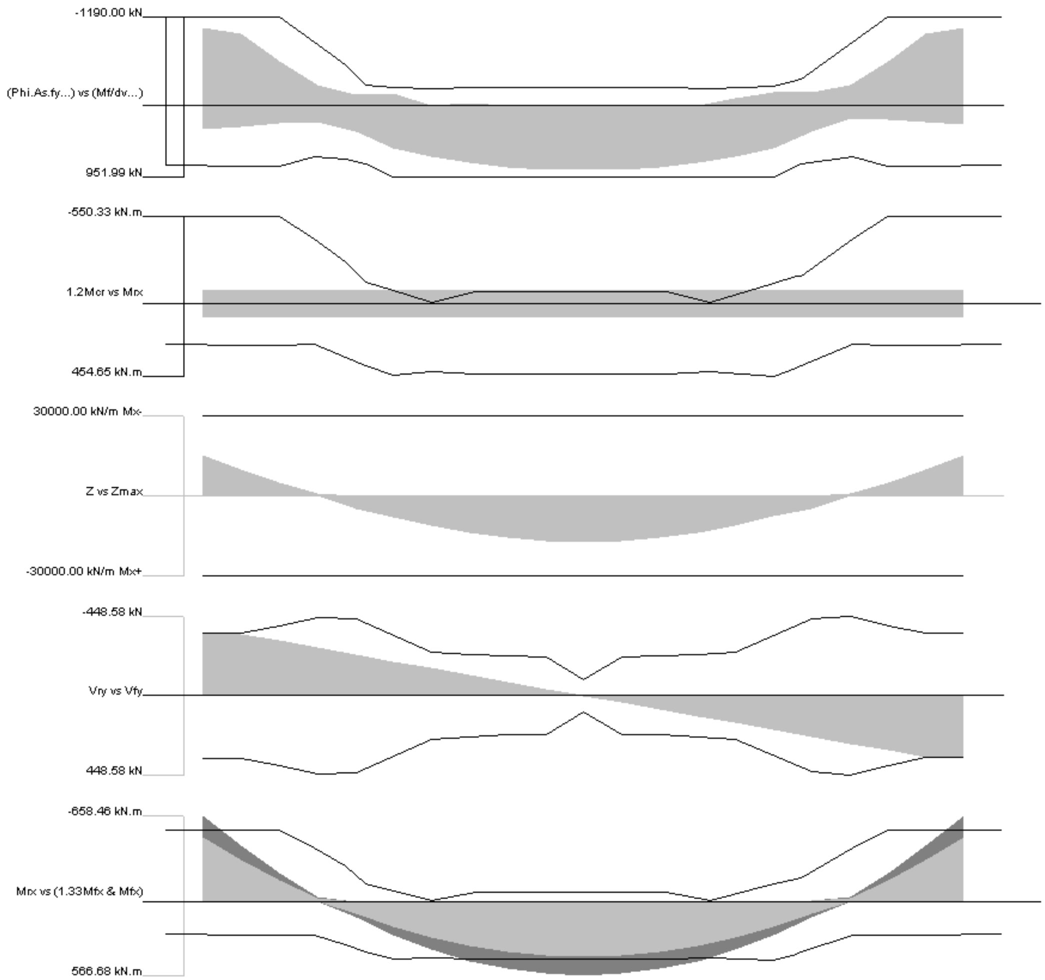
Display Reinforcement, Dimensions & Diagrams

- Open the **View Options** dialog box and expand the *Beam Diagram* root. Activate force and resistance diagrams. Go to the **Dimensions** tab and enter 0 in the *Decimal* field.

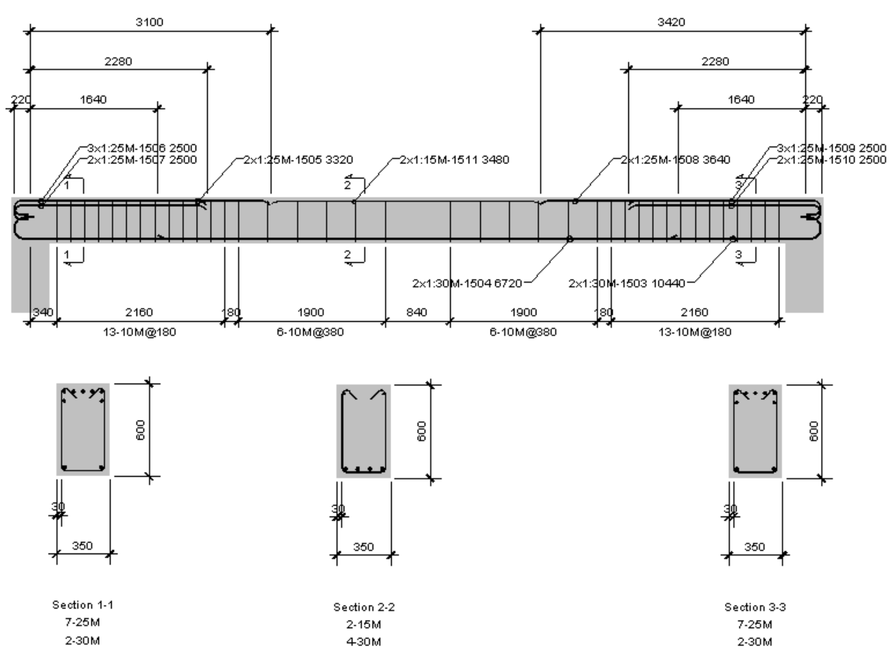
Cracking results will be available if serviceability load combinations were analysed.



- Generate cross-sections with the function **Automatic Generation of Cross-sections**. VisualDesign creates cross-sections at mid-span and at supports (or columns in our case).



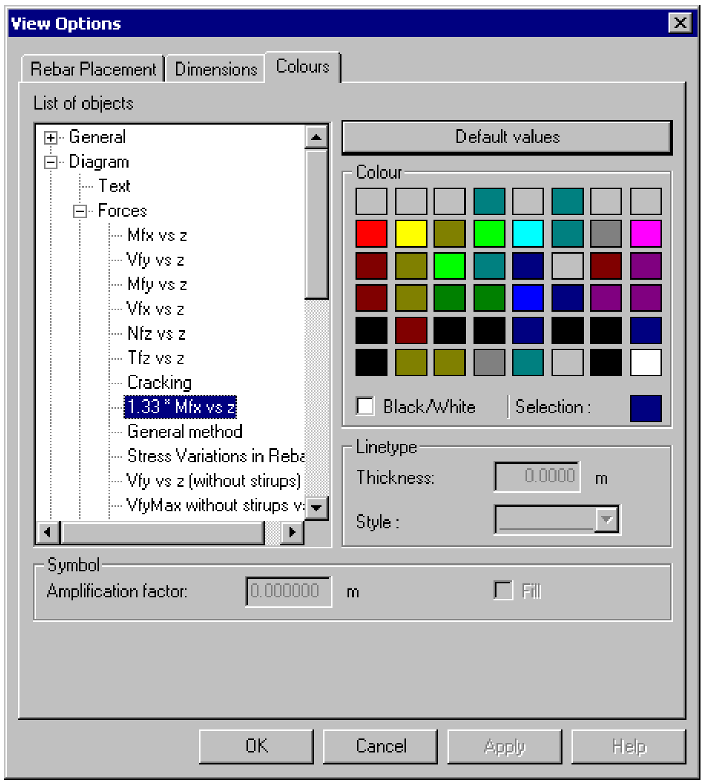
-1190.00 kN
(Phi.As.fy...) vs (Mf/dv...)
951.99 kN
-550.33 kN.m
1.2Mcr vs Mrx
454.65 kN.m
30000.00 kN/m Mx-
Z vs Zmax
-30000.00 kN/m Mx+
-448.58 kN
Vry vs Vfy
448.58 kN
-658.46 kN.m
Mrx vs (1.33Mfx & Mfx)
566.68 kN.m



3100
2280
220
1640
3420
2280
1640
220
3x1:25M-1506 2500
2x1:25M-1507 2500
2x1:25M-1505 3320
2x1:15M-1511 3480
2x1:25M-1508 3640
3x1:25M-1509 2500
2x1:25M-1510 2500
2x1:30M-1504 6720
2x1:30M-1503 10440
340
2160
180
1900
840
1900
180
2160
13-10M@180
6-10M@380
6-10M@380
13-10M@180
600
30
350
Section 1-1
7-25M
2-30M
Section 2-2
2-15M
4-30M
Section 3-3
7-25M
2-30M

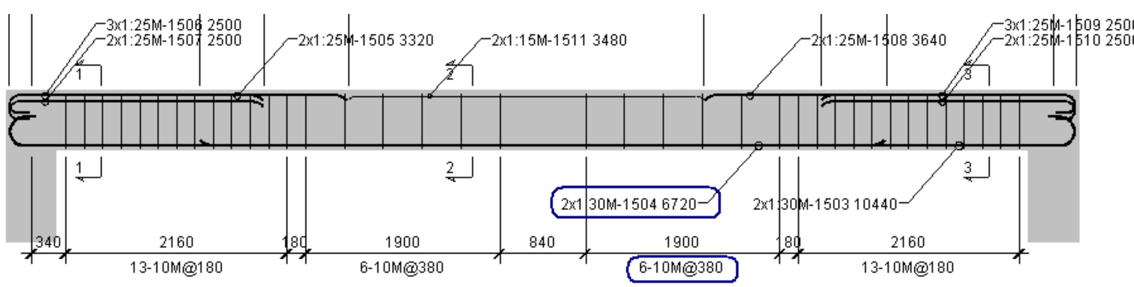
- Use the zoom and dynamic pan functions to look at diagrams and reinforcing bars details.

- Select the **Colours** tab (**View Options**) to know the colour for each result.



Longitudinal Reinforcement

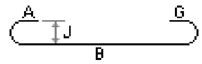
- Double click on any longitudinal rebar to open the **Longitudinal Reinforcement** spreadsheet. It contains details about this rebar and its placement in the beam.



We double clicked on longitudinal rebars #1504:

N. B. Nomenclature 2x1-30M-1504 6720: 2 rebars of dimension 30M, VisualDesign number 1504, length of 6720mm.

Longitudinal Reinforcement Spreadsheet

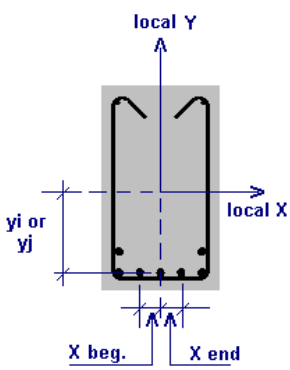
1	Number	Reinforcement	Bending Shape	Horiz.Flip	Verti.Flip	No. of Bars
1	1:30M-1504	30M		[ ]	[ ]	2

Longitudinal Reinforcement Spreadsheet

1	X beg. mm	X end mm	zi mm	zj mm	yi mm	yj mm	Left End	Right End
1	-55	55	1640	8360	-244	-244	No Hook	No Hook

zi and zj indicate the positions of the rebars along the z-axis of continuous system. Therefore, rebars begin at zi = 1640mm and end at zj=8360mm.

X beg. and X end represent the far left and far right positions of rebars in the beam, according to the local x-axis of continuous system. This x-axis is shown in the image below, on the beam cross-section.



Stirrups

- Double click on the stirrup sequence shown above to open the **Transverse Reinforcement** spreadsheet.

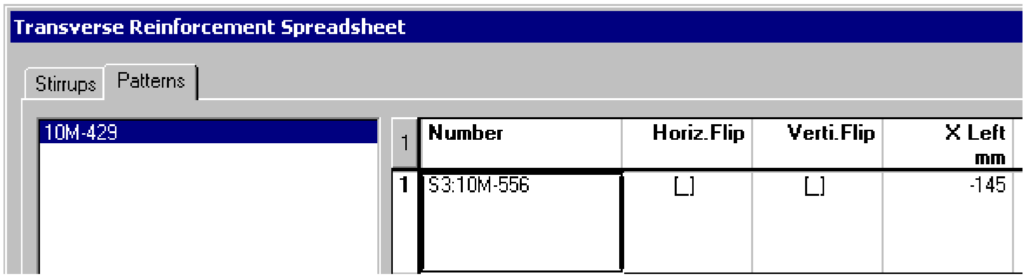
Transverse Reinforcement Spreadsheet

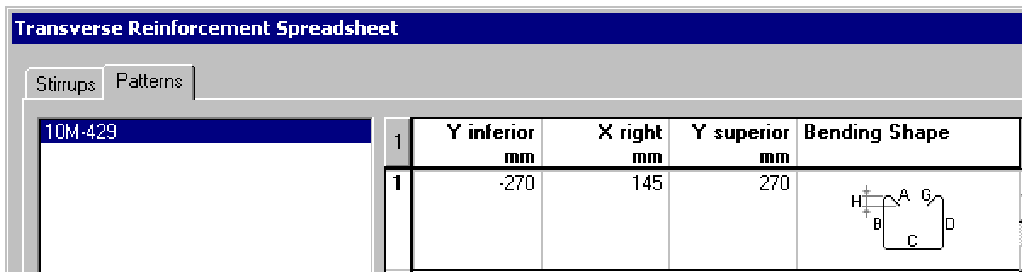
Stirrups | Patterns

1	Number	Reinforcement	Number of patterns	Spacing s mm	zi mm	zj mm
1	10M-429	10M	6	380	5420	7320

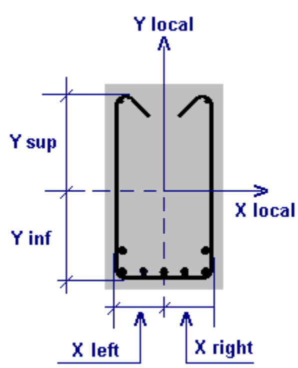
The VisualDesign number for this sequence is 429. There is six stirrups spaced apart at 380mm, and the reinforcing bar is 10M. The sequence begins at zi = 5420mm along the z-axis and ends at zj=7320mm.

Go to the **Patterns** tab.





Position of stirrups is measured from the axis of continuous system to the corners defining the stirrup outer outline. Y-axis points up.



General Results Spreadsheet

- Go to **Results** / **General Results**.

Note:

In general, data include in this spreadsheet are those displayed on screen. For example, if the ***Cracking*** diagram is not displayed on screen, values ***Z*** and ***Zmax*** will not appear in the spreadsheet. (VisualDesign builds the spreadsheet from the displayed and calculated values, each time that this spreadsheet is called up.)

The Positive Bending Moment tab

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	Member Number	Z mm	Mfx Max kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %	fcr MPa	Mcr kN.m	d mm
1	2	250	-16.98	793.41	314.68	249.56	293.41	355.44	39.66	3.29	71.43	360
2	2	725	-8.96	816.00	282.15	260.55	305.77	370.88	34.58	3.29	71.43	365
3	2	1200	-1.90	816.00	245.29	260.55	305.77	370.88	30.06	3.29	71.43	365
4	2	1675	4.23	684.01	219.21	259.88	306.56	374.60	32.05	3.29	71.43	424
5	2	2150	93.78	740.11	353.61	360.02	426.17	525.33	47.78	3.29	71.43	534
6	2	2625	195.23	951.99	571.58	445.26	537.64	652.85	60.04	3.29	71.43	544

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	dv mm	bw mm	As mm²	ρ %	εx	θ °	β	c/d	c/d Max	Z kN/m	Z Max kN/m	Ieff	ICr
1	315	350	2333.59	1.85	0.000601	33.16	0.17	0.21	0.64	0.00	30000.00	1.00	1.00
2	319	350	2400.00	1.88	0.000549	32.84	0.18	0.21	0.64	0.00	30000.00	1.00	1.00
3	319	350	2400.00	1.88	0.000497	31.95	0.19	0.21	0.64	0.00	30000.00	1.00	1.00
4	380	350	2011.82	1.36	0.000488	31.46	0.20	0.18	0.64	0.00	30000.00	1.00	1.00
5	486	350	2247.38	1.20	0.000741	33.73	0.19	0.18	0.64	4868.21	30000.00	1.00	0.46
6	468	350	2800.00	1.47	0.000927	35.27	0.19	0.33	0.64	7780.84	30000.00	0.62	0.55

The Negative Bending Moment tab

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	Member Number	Z mm	Mfx Min kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %	fcr MPa	Mcr kN.m	d mm
1	2	250	-495.08	1190.00	1040.22	548.36	656.78	803.32	87.41	3.29	71.43	529
2	2	725	-320.51	1190.00	964.49	550.33	658.41	805.69	81.05	3.29	71.43	529
3	2	1200	-164.14	1190.00	591.29	550.33	658.41	805.69	49.69	3.29	71.43	529
4	2	1675	-26.03	817.66	265.94	394.90	467.30	577.57	32.52	3.29	71.43	532
5	2	2150	9.40	402.62	143.15	202.74	240.53	297.93	35.55	3.29	71.43	540
6	2	2625	13.62	244.00	155.19	81.32	96.80	118.49	63.61	3.29	71.43	357

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	dv mm	bw mm	As mm²	ρ %	εx	θ °	β	c/d	c/d Max	Z kN/m	Z Max kN/m	Ieff	ICr
1	461	350	3500.00	1.89	0.002000	36.60	0.09	0.31	0.64	15500.49	30000.00	0.63	0.63
2	462	350	3500.00	1.89	0.001863	36.46	0.10	0.31	0.64	9943.11	30000.00	0.65	0.63
3	462	350	3500.00	1.89	0.001127	36.14	0.15	0.31	0.64	4976.34	30000.00	0.73	0.63
4	483	350	2404.88	1.29	0.000600	32.54	0.19	0.18	0.64	810.17	30000.00	1.00	0.49
5	504	350	1184.28	0.63	0.000459	29.66	0.21	0.11	0.64	0.00	30000.00	1.00	1.00
6	333	350	3241.20	2.59	0.000332	28.56	0.25	0.15	0.64	0.00	30000.00	1.00	1.00

The Shear Force tab

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	Member Number	Z mm	Vfy Max kN	Vfy Min kN	Design Load %	εx	θ °	β	Vcy kN	Vsy kN	Vry kN	dy mm	dvy mm
1	2	250	-15.87	-347.04	99.38	0.001781	37.94	0.13	89.14	260.05	349.20	529	476
2	2	725	-15.87	-347.04	99.38	0.001269	36.76	0.15	108.48	240.72	349.20	529	476
3	2	1200	-13.88	-308.48	78.16	0.000778	34.38	0.19	131.86	262.82	394.68	529	476
4	2	1675	-11.88	-269.92	61.34	0.000488	31.46	0.20	142.57	297.46	440.03	532	483
5	2	2150	-9.89	-231.36	53.82	0.000741	33.73	0.19	145.00	284.91	429.90	540	504
6	2	2625	-7.89	-192.80	57.25	0.000927	35.27	0.19	135.80	200.95	336.74	544	489

The Axial Force tab

General Results Spreadsheet - S_2

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

21	Member Number	Z mm	Nz Max kN	Mx Max kN.m	Nz Min kN	Mx Min kN.m
1	2	250	-24.86	-16.98	-285.65	-495.08
2	2	725	-24.86	-8.96	-285.65	-320.51
3	2	1200	-24.86	-1.90	-285.65	-164.14
4	2	1675	-24.86	4.23	-285.65	-26.03
5	2	2150	-24.86	93.78	-285.65	9.40
6	2	2625	-24.86	195.23	-285.65	13.62

Bar List (partial)

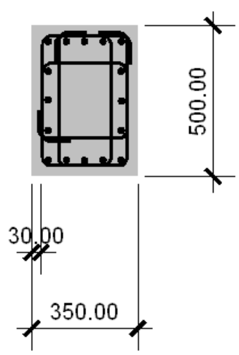
- To consult or print the bar list for this beam, go to **Rebar Placement** / **Bar List**. Select spreadsheet content, right click, and select the "Print" function in the contextual menu.

Bar List (Complete)

- To consult or print the bar list for the whole project, go back to VisualDesign main window and go to **Results / Bill of Materials / Bar List**.

Editing Rebars

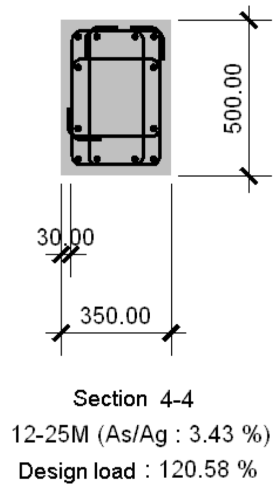
Here is an example that will show you how to edit main rebars in a column. We are going to reduce the number of main rebars to 12 and change the dimension to 30M. To help you, zoom in the cross-section.



Section 4-4
16-25M (As/Ag : 4.57 %)
Design load : 98.54 %

Deleting 4 rebars:

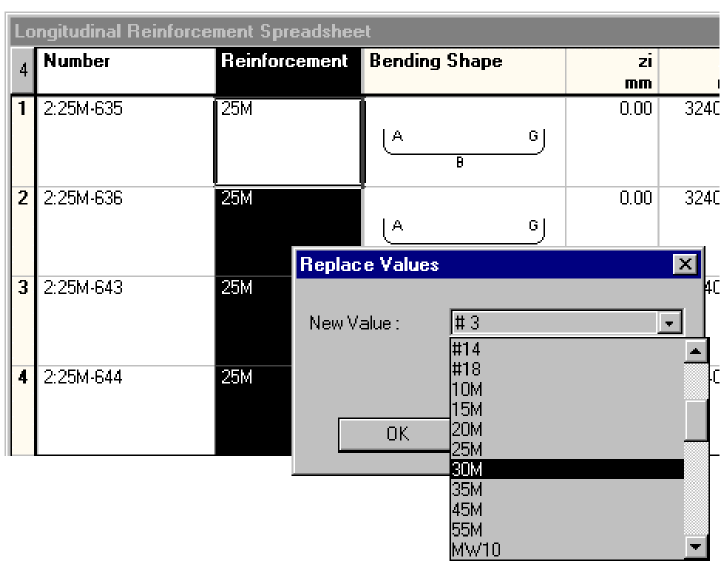
- Press down the [Ctrl] key while you click on each 25M rebar located at the middle on each face. Press the [Delete] key.



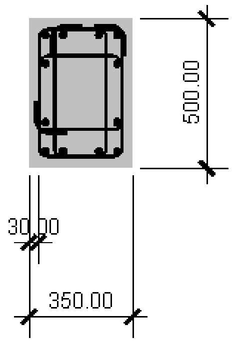
The column resistances are automatically recalculated and displayed on diagrams.

Modifying the dimension of rebars:

- Open the **Longitudinal Reinforcement** spreadsheet in **Rebar Placement** menu. Click the *Reinforcement* column title and right click. Choose the **Replace** function in contextual menu, and select 30M in the list box.



- Click OK to close the spreadsheet.



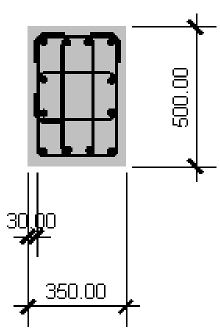
Section 4-4
12-30M (As/Ag : 4.80 %)
Design load : 89.93 %

Moving main rebars:

- Go to **Help** menu and select **Editing keys**. These tables present a summary of short cut keys that are helpful to edit main rebars, stirrups, and cross-sections. To keep this window open while editing, press the button "Options" and select the command **Keep Help on Top / On top**.

- Activate the **Move**  icon on Edit toolbar and select one rebar. Use the displayed target to guide you. Release the mouse button to fix the rebar position. Use the **Undo** command if needed. Move other rebars. When the **Move** command is no longer needed, disable the **Cursor** mode . If you do not, everything selected by your cursor will be moved!

Main rebars have been moved towards the centre and stirrup legs also. The final design is as follows:



Coupe 4-4
12-30M (As/Ag : 4.80 %)
Sollicitation : 94.83 %

- To save this design, go to **File / Save Rebar Placement**.

EXAMPLES 12 & 13

Prestressed Concrete Design

Pre-Tensioning

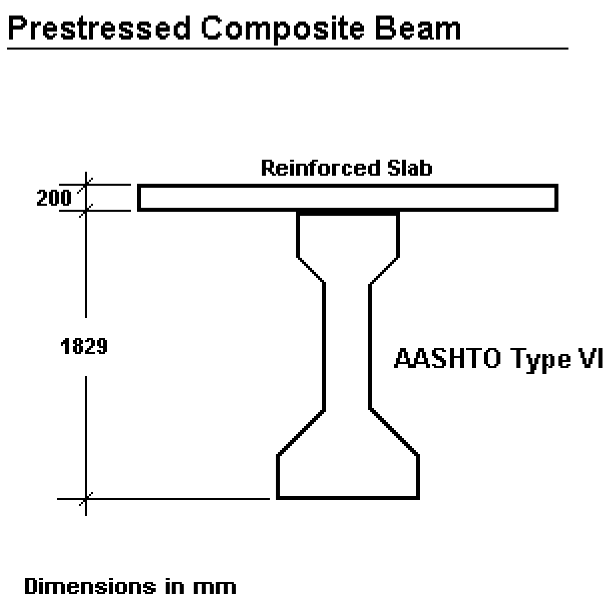
Post-Tensioning

Continuous Beam with Pre-Tension

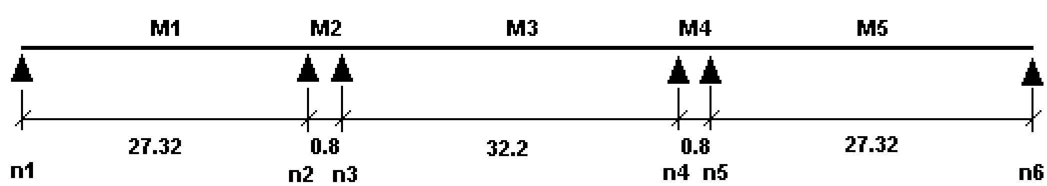
Description of the project

A continuous prestressed concrete beam is composed of an AASHTO (type VI) section and is composite with a slab of 200mm thick. The specified compressive strength of concrete is 50MPa for the beam and 30MPa for the slab. 20M rebars of grade 400W are supplied in two directions and act perpendicular to the beam section.

½" prestressing strands will be used. The grade is 270 ksi (1860 MPa). Jacking will be done at 75% of maximum stress fpu. Pre-tension will be applied at initial pre-tensioning stage.



The beam spans are as follows:



Project Configuration

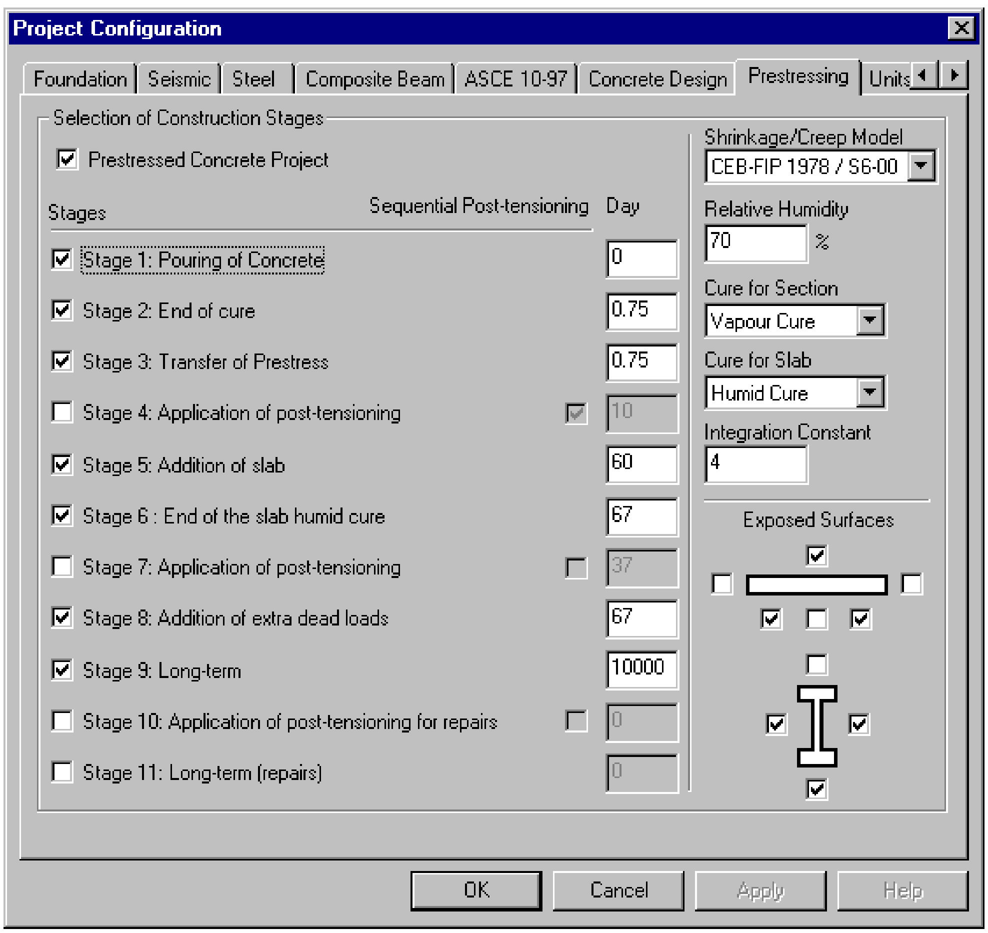
- Go to the **Project Configuration** dialog box (**File** menu) and select the **Preferences** tab. In section *Dialog Box Display*, disable the display of dialog boxes for nodes and members to get a quicker editing. Press OK.

- Now, select the **Analysis** tab. Choose a non-linear analysis. We will set the number of subdivisions to 40 for concrete beams to get accurate diagrams.

- Select the **Concrete Design** tab. Choose the *General Method* for the design of prestressed concrete elements. The number of stirrup sequences is fixed to 7.

- Finally, select the **Prestressing** tab. Check the *Prestressed Concrete Project* box to activate construction stages located below.

  - Check the boxes corresponding to construction stages 1, 2, 3, 5, 6, 8 and 9. Days are cumulative. Choose a shrinkage and creep model and enter the relative humidity of area. Complete the *Exposed Surfaces* section of the dialog box. These data are required to compute shrinkage and creep effects.



- Press OK to save data and exit the dialog box.

Concrete Materials, Slab and Strands

Concrete Materials

The section and slab materials are already included in database.

Definition of the Slab

The slab must be defined in the **Slabs** spreadsheet. Thickness is 200mm and reinforcement is composed of 20M rebars with a spacing of 300mm in both directions. The steel material is 400W.

- Go to **Structure** menu and select **Slabs**. Insert a line in the **Slabs** spreadsheet and enter the following data:

Slabs Spreadsheet

1	Number	Steel deck	Direction	tc mm	hd mm	to mm	Rebar, top	s, top mm
1	Slab	Null	Parallel	200	0	200	20M	300

hd represents the deck thickness and *to* is the total thickness, which is automatically calculated.

Slabs Spreadsheet

1	d, top mm	Rebar, bot.	s, bot. mm	d, bottom mm	Material Rebar	Material Concrete
1	150	20M	300	150	G30.18-400W	Con030

Definition of Strands

Prestressing cables are composed of ½" strands (grade 1860 MPa (270ksi)).

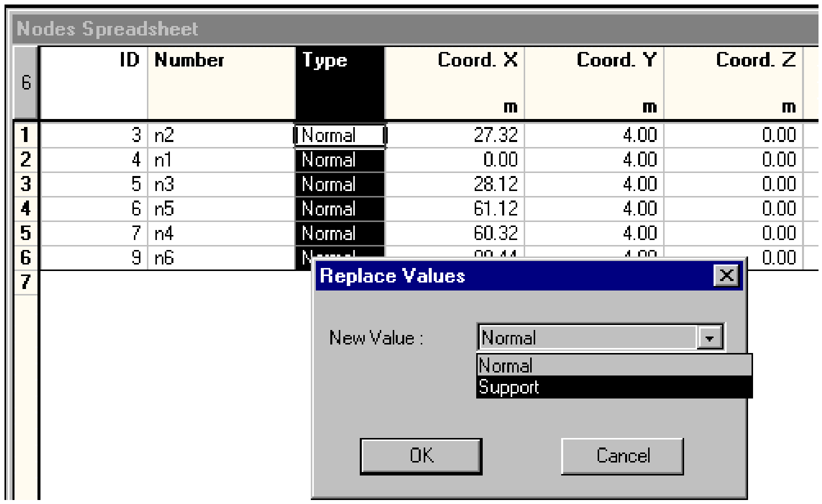
- Go to **Common** menu and select **Cables / Steel Grades**. Grade 270 ksi (1860 MPa) is already included in the **Cable Steel Grades** spreadsheet. Press OK.

- Select **Cables / Strands**. ½ in strands are included. Make sure that the steel grade is 1860MPa. If this is not the case, double click in the cell and select the right steel grade.

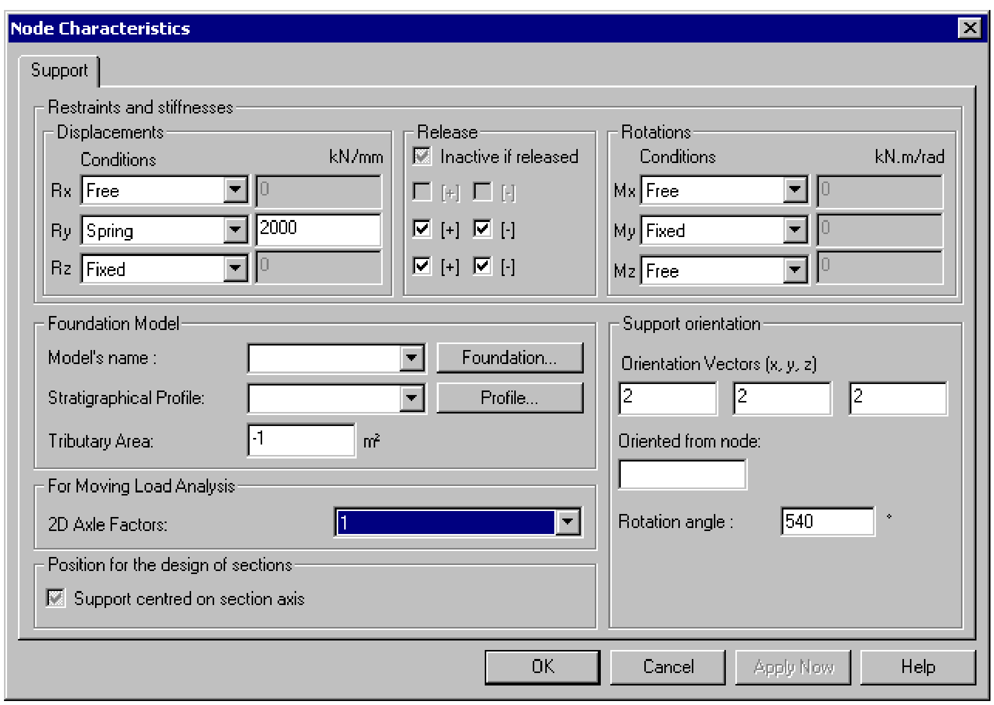
Structural Model

Nodes

- Go to **Structure** menu and select **Nodes**. Insert six lines and enter node coordinates. All of our nodes are supports. Select the "Type" column title, right click, and select the function **Replace** in the contextual menu. In the **Replace Values** dialog box, select *Support,* and press OK. Close the **Nodes** spreadsheet.



- Activate the Support icon, select them all, and press the **Properties** icon to open the **Node Characteristics** dialog box. Go to the **Support** tab.



- ♦ Free support displacements towards the x direction (Rx) to allow displacement due to prestress.
- ♦ Model an elastic support for Ry. Then, enter a value of 2000 kN/mm as the spring stiffness.
- ♦ Restrain rotations My and free Mz. For the moment, free all rotations Mx.

- Select 2D axle factor and press OK to exit the dialog box.

- Double click on the first support and restrain displacement Rx and rotation Mx. Press OK.

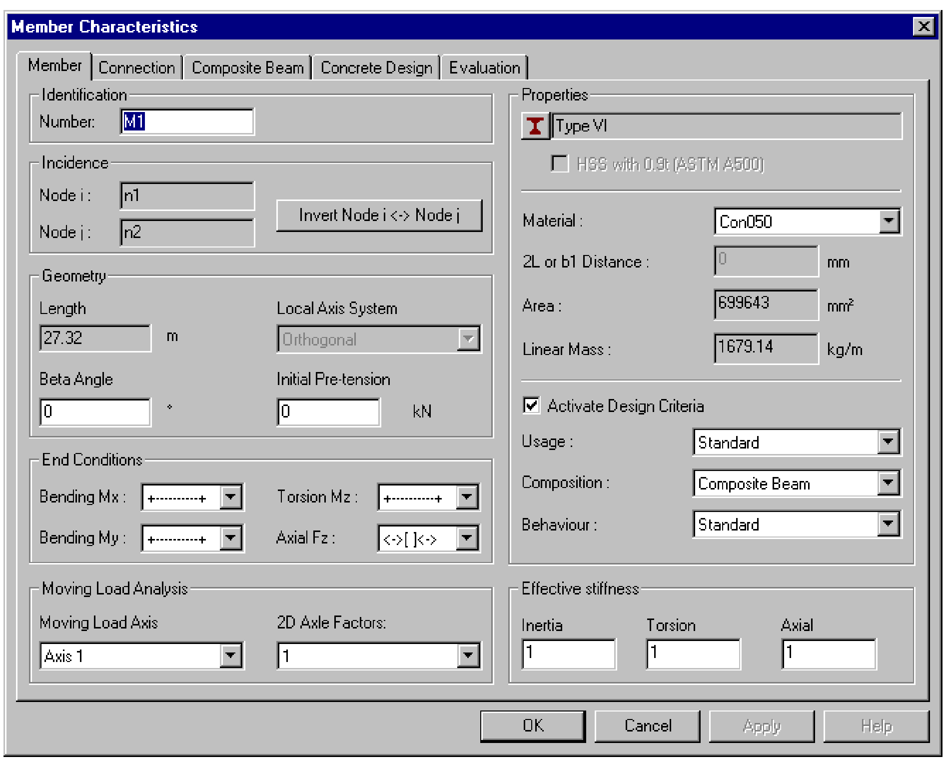
Add Members

- Add members between supports: Activate the "Member" icon and the "Add" mode . Click once the first node and then, the second node. Do the same to model other members.

- Exit the **Add** mode by selecting either the **Restricted window** icon or the **Extended window** icon . You can also right click on screen and select the **Cancel** function in contextual menu.

Members Properties

- Select all members and press the **Properties** icon to open the **Member Characteristics** dialog box.

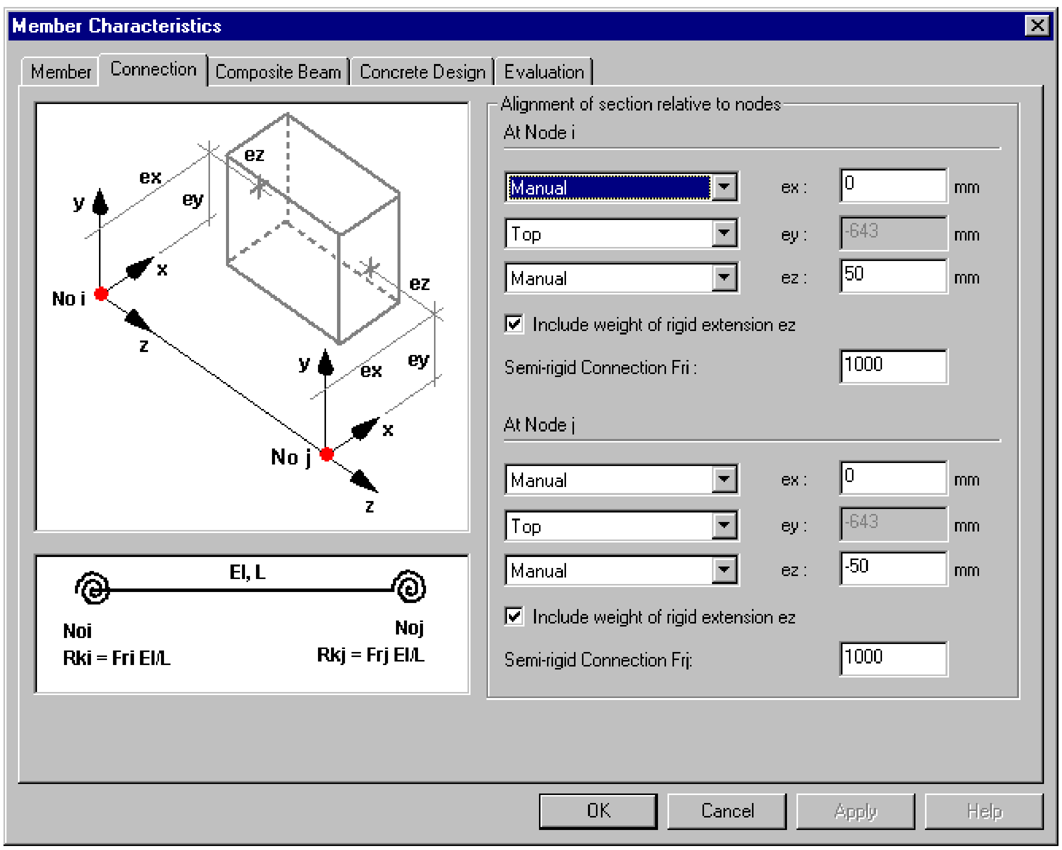


- Click on the *I Beam* icon and select section *AASHTO Type VI* in the *Concrete* root. Choose a 50MPa concrete material. Activate design criteria and select option *Composite beam* in the "Composition" field. Define the beam end conditions (+----+ for continuous members). Select moving load axis and 2D axles factors.

Alignments and Rigid Extensions

- Go to the **Connection** tab. Rigid extensions (ez) must be modeled at the left and right side of each support. To create rigid extensions "ez", choose option *Manual* and enter 50mm for node i and –50mm for node j. ***Look carefully at member local axis system***.

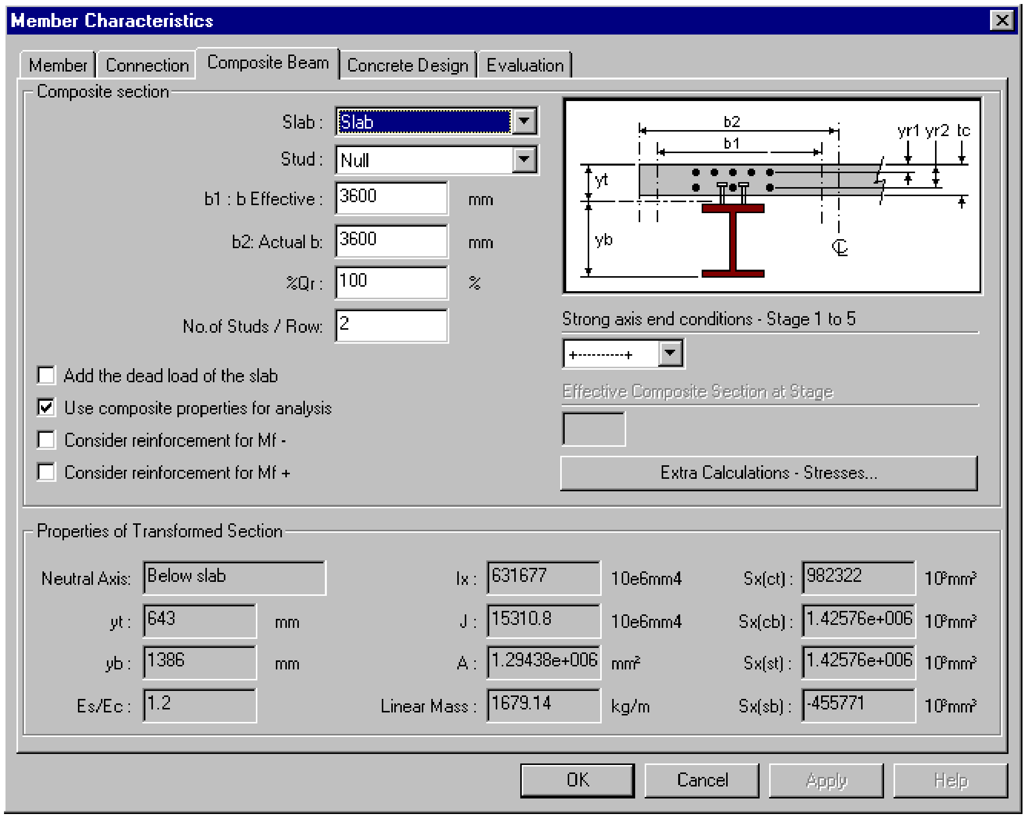
- Then, we will align the members' longitudinal axis (ey) at the top of the sections. Select *Top* for alignment "ey", for node i and j. Include the weight of rigid extensions.



The centre of gravity is located 643mm below the top of composite member. This dimension will be useful when placing prestressing cables within the beams.

Composite Beams

- Go to the **Composite Beam** tab.



- Select the slab that you defined beforehand. Enter *Effective b* and *Actual b*. Never check “Add the dead load of the slab” if your project includes construction stages. The slab dead load will be added at the right construction stage, as we will see further on.

- Activate option "Use composite properties for analysis".

- **Member End Conditions**: Specify the beams' end conditions on strong axis (+------+) for stages 1 to 5 (before it changes to a statically determinate structure).

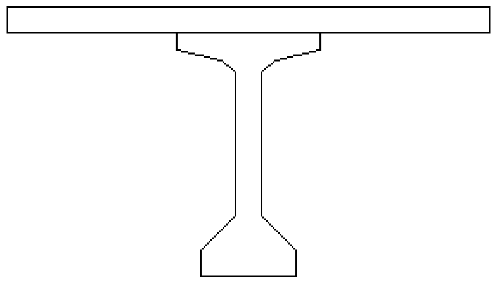
Beams M1, M3, and M5 must have continuous end conditions (+---------+) but beams M2 and M4 will be hinged (o------o). (To edit, click OK to exit this dialog box, select beams M2 and M4, and choose end conditions o------o.)

- Select the **Concrete Design** tab. For shear design, select option *At face of support*. Press OK.

View Options

- Display the shape outline through the **Attributes** tab of **View Options** dialog box. Press the keyboard control key [Pg Up] to get an isometric view of continuous beams. Press the [Home] key to get a front view of the section and zoom in.

Here is a view of the composite section, as it is displayed on screen without global axis system, nor supports.



Concrete Specification

- Go to **Structure** / **Specifications/ Concrete**. The design will be done using standard CAN/CSA-S6-00.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

4	Number	Code	Type of analysis	Maximum Capacity Factor %	Calcul. Method Mr/Vr Positive	Calcul. Method Mr/Vr Negative
1	A23.3-Design	CAN/CSA-A23.3-95	Design	100.00	Maximize Mr	Maximize Mr
2	A23.3-Vérif.	CAN/CSA-A23.3-95	Verification	100.00	Maximize Mr	Maximize Mr
3	S6-00-Design	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
4	S6-00-Vérif	CAN/CSA-S6-00	Verification	100.00	Maximize Mr	Maximize Mr

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

4	Epoxy Coated	Longitudinal Optimization	Longitudinal Reinforcement Material	Selection of Longitidunal Rebar
1	[ ]	Weight	G30.18-400R	20M 25M 30M
2	[ ]	Weight	G30.18-400R	20M 25M 30M
3	[ ]	Weight	G30.18-400R	20M 25M
4	[ ]	Weight	G30.18-400R	20M 25M 30M

Withdraw rebar 30M from the selection by double-clicking and disabling it.

- Select the **Beam / Column / Joist** tab. Modify the number of layers and withdraw rebar 10M from the list of transverse rebars.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

4	Number	Maximum No. of Layers in Tension	Maximum No. of Layers in Compression	Transverse Optimization	Transverse Reinforcement Material	Selection of Transverse Rebar
1	A23.3-Design	4	4	Weight	G30.18-400R	10M 15M
2	A23.3-Vérif.	4	4	Weight	G30.18-400R	10M 15M
3	S6-00-Design	3	2	Weight	G30.18-400R	15M
4	S6-00-Vérif	4	4	Weight	G30.18-400R	10M 15M

- Press OK to exit the spreadsheet.

Continuous Systems

- In the Structure activation mode, go to **Structure** / **Continuous Systems**. A continuous system has been automatically created.

- Select specification *S6-00-Design*. When this specification is selected, the "Exposure" fields are automatically set to *Manual*. Concrete covers must be specified by users and they are measured from the stirrup outside diameter.

Continuous Systems Spreadsheet

1	Number	Specification	Type	Interaction	Description	Exposure Top	Top Cover mm
1	S_4	S6-00-Design	Beam/Column	Bending	S_4	Manual	50

Continuous Systems Spreadsheet

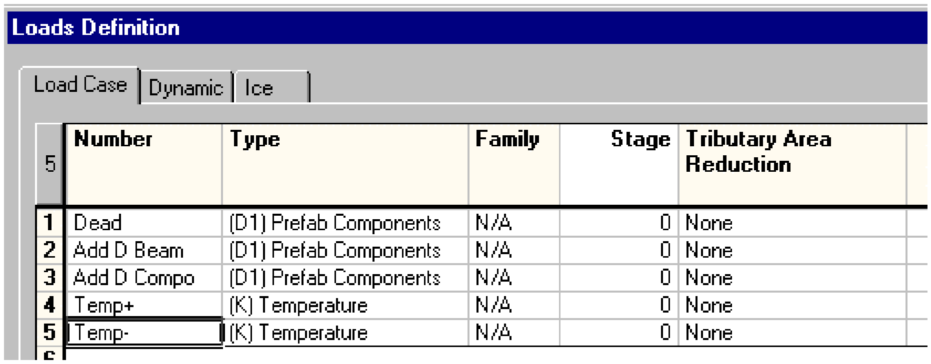
1	Exposure Bottom	Bottom Cover mm	Exposure Left	Left Cover mm	Exposure Right	Right Cover mm	Crack Control Top kN/m	Crack Control Bottom kN/m
1	Manual	50	Manual	50	Manual	50	30000.00	30000.00

- Enter the crack control parameters and click OK.

Load Cases

Define all load cases (types and titles) that will be used for this project, including construction stage loads.

- Go to **Loads** / **Load Cases** / **Definition**. Here are the load cases that we will need:


Loads Definition

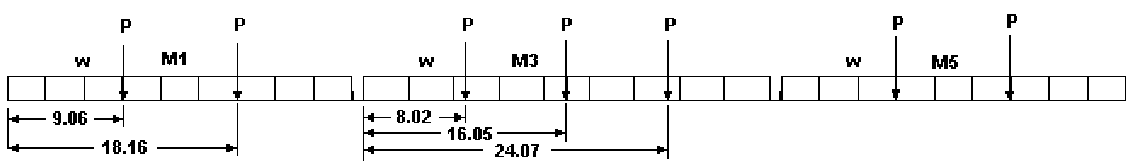
Load Case | Dynamic | Ice

5	Number	Type	Family	Stage	Tributary Area Reduction
1	Dead	(D1) Prefab Components	N/A	0	None
2	Add D Beam	(D1) Prefab Components	N/A	0	None
3	Add D Compo	(D1) Prefab Components	N/A	0	None
4	Temp+	(K) Temperature	N/A	0	None
5	Temp-	(K) Temperature	N/A	0	None

It is very important to select the right type of load (S6-00 standard) in this spreadsheet if you plan to use the generation wizard for load combinations, as you will see further on.

Applying Loads on the Structure

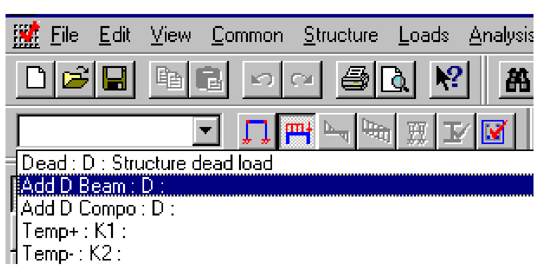
Permanent loads applied at stage 5



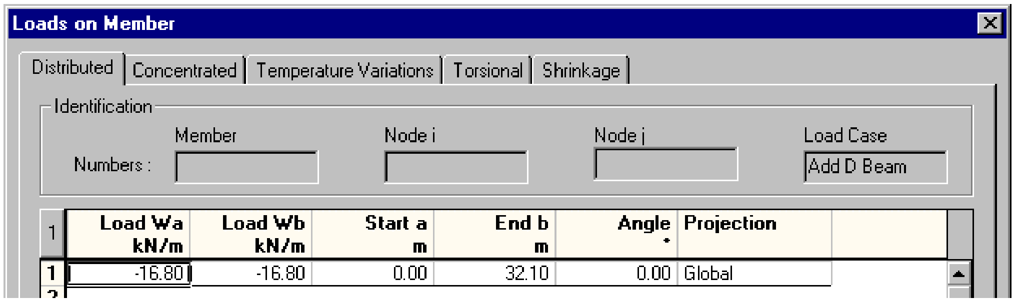
Load Case: Add D Beam

P = 27.60 kN w = 16.80 kN/m

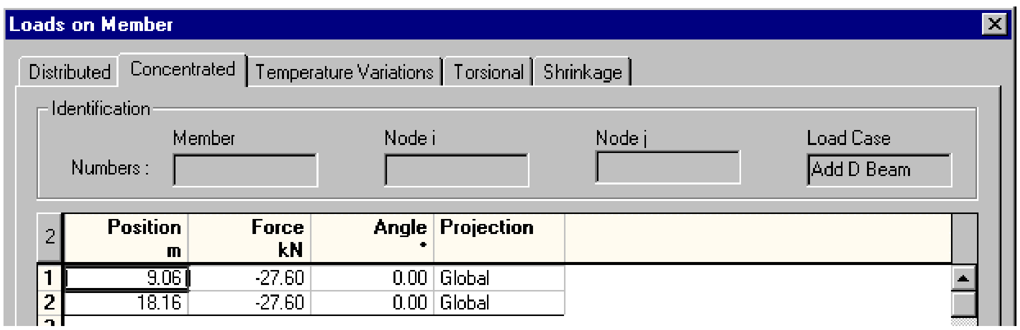
- Activate the "Load Case" mode and select the *Add D Beam* load case. It represents permanent loads that will be applied at construction stage 5.



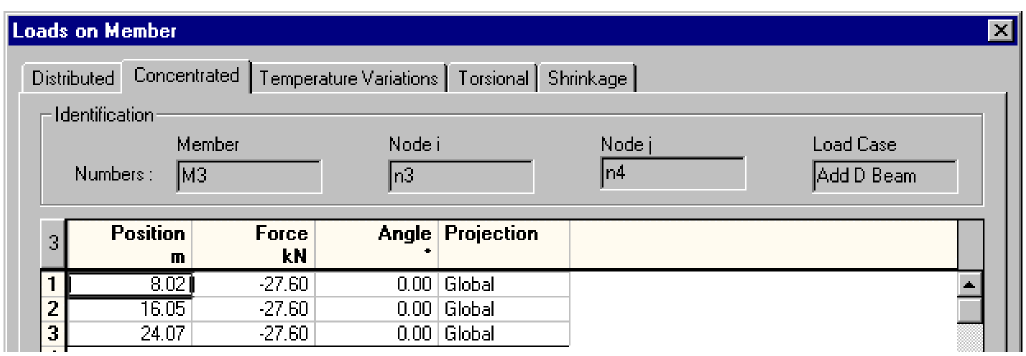
- Select all members and click the **Properties** icon to open the Loads on Member dialog box. Insert a line in the **Distributed** tab. Double click in *Wa* and *Wb* cells and enter –16.80. Press OK.



- Now, select members M1 and M5 and click the **Properties** icon. Insert two lines in the **Concentrated** tab. Enter the position, magnitude, and projection of each punctual load relatively to node i. Close the spreadsheet.



- Double-click on beam M3. Enter other concentrated loads. Close the spreadsheet.



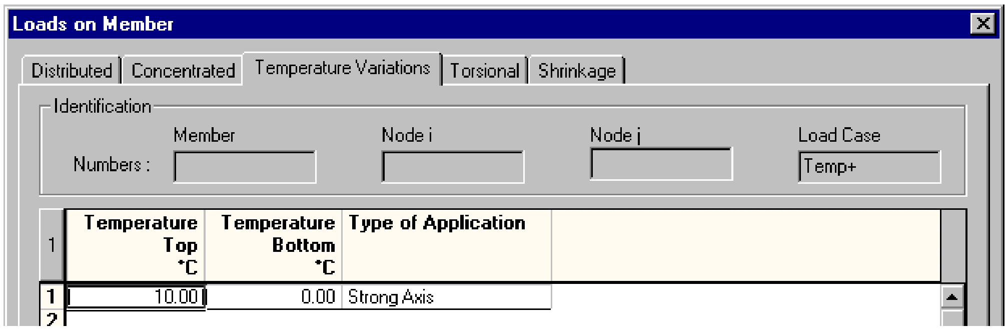
Permanent Loads Applied at Stage 8

Select the *Add D Compo* load case. This uniform load is applied on members M1, M3 & M5, and its magnitude is equal to –8.58 kN/m.

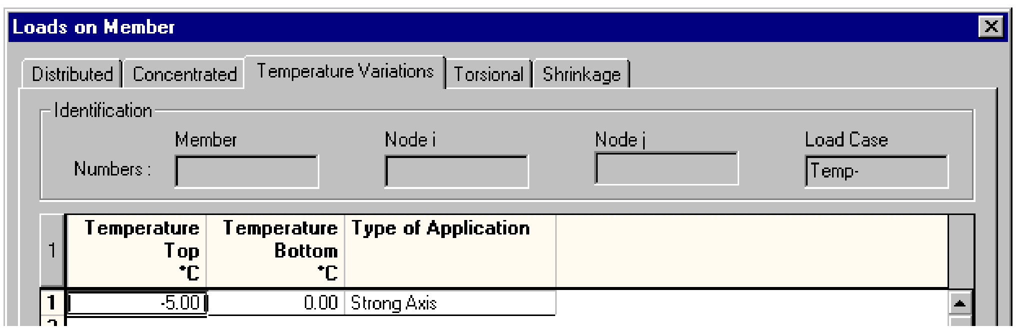
Loads Due to Temperature Variations

These loads are applied to all beams.

- Select the *Temp+* load case, select all beams, and press the **Properties** icon.
- Insert a line in the **Temperature Variations** tab. Values do not represent the absolute temperature but the temperature variation at the top and bottom of the section.



- Press OK.
- Follow the same procedure for load case *Temp-*.



Load Combinations (Stages)

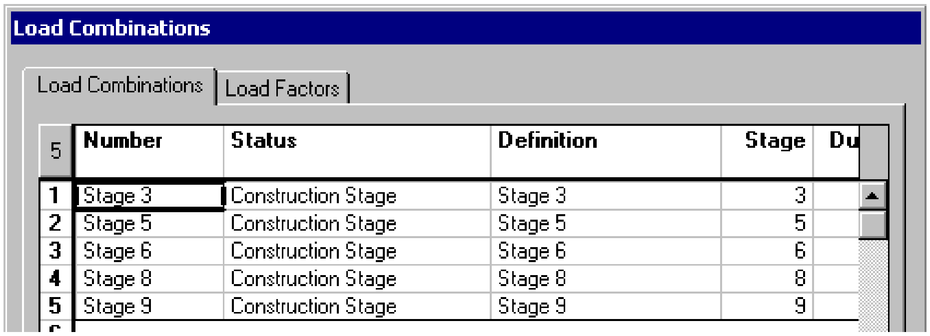
Before calling up the **Load Combination Generation Wizard**, construction stage load combinations must be defined "by hand" in the **Load Combinations Definition** spreadsheet and construction stage numbers must be specified.

Construction Stage Load Combinations

You must define one load combination per construction stage. These load combinations must have a *Construction Stage* status. Construction stage loads are cumulated by VisualDesign, meaning that they are automatically added to the next construction stage.

- Go to **Loads** / **Load Combinations / Definition.**

- Insert five lines in the **Load Combinations** spreadsheet and enter a name for each construction stage. Double-click in the "status" cell and select *Construction Stage*. Double click in the "Stage" cell and type in the construction stage number.



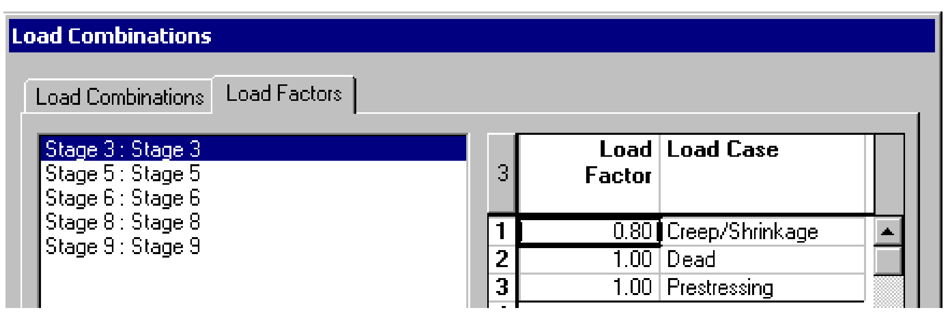
Load Factors

- Select the **Load Factors** tab.

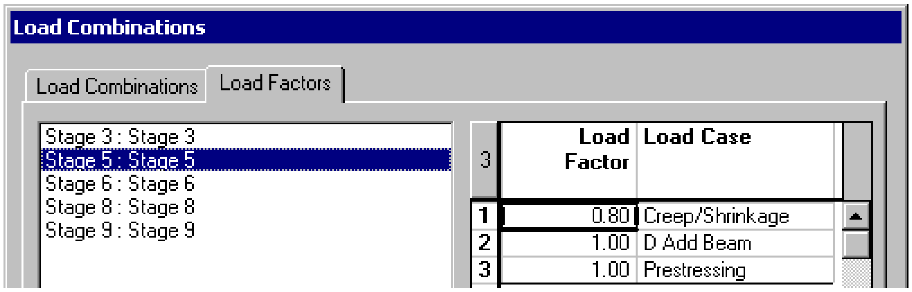
You must select the load case type in the "Load Case" column and enter load factors for each load case that is part of a load combination.
The "Prestressing" and "Shrinkage/Creep" loads are virtual loads. VisualDesign recognizes these loads and recovers secondary prestress results at each construction stage. Do not forget to include these virtual loads in each construction stage load combination.

Stage 3 – Transfer of Prestress. Include the AASHTO dead load only.

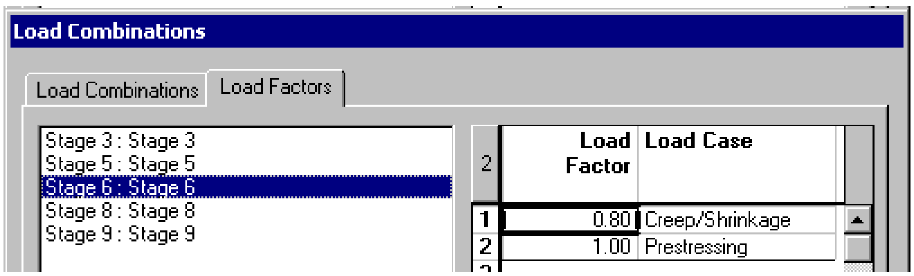
- Highlight *Stage3* load combination in the left part of the dialog box. Place your cursor at line 1 in the right part and insert three lines. Double click in the *Load Case* cell and choose the right type of load case in the drop-down list box. Enter load factors according to CAN/CSA-S6-00.



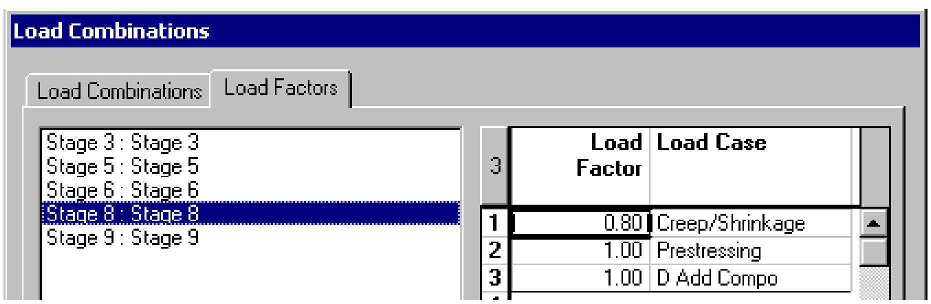
Stage 5 – Addition of Slab. Additional dead loads on beams.



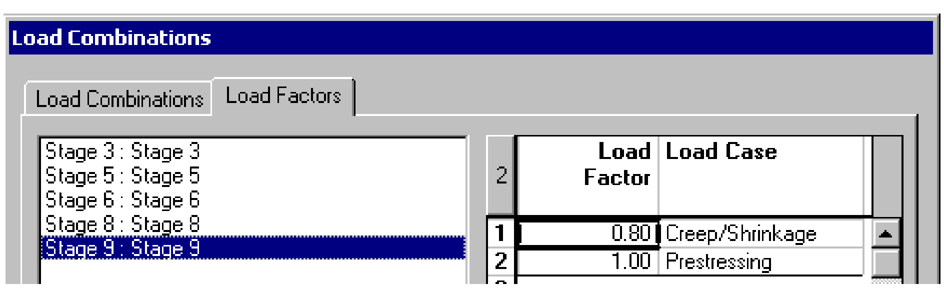
Stage 6 – End of Humid Cure: There is no extra load at this stage. However, include virtual load cases.



Stage 8 – Additional Dead Loads. These loads will be applied on the composite beam.



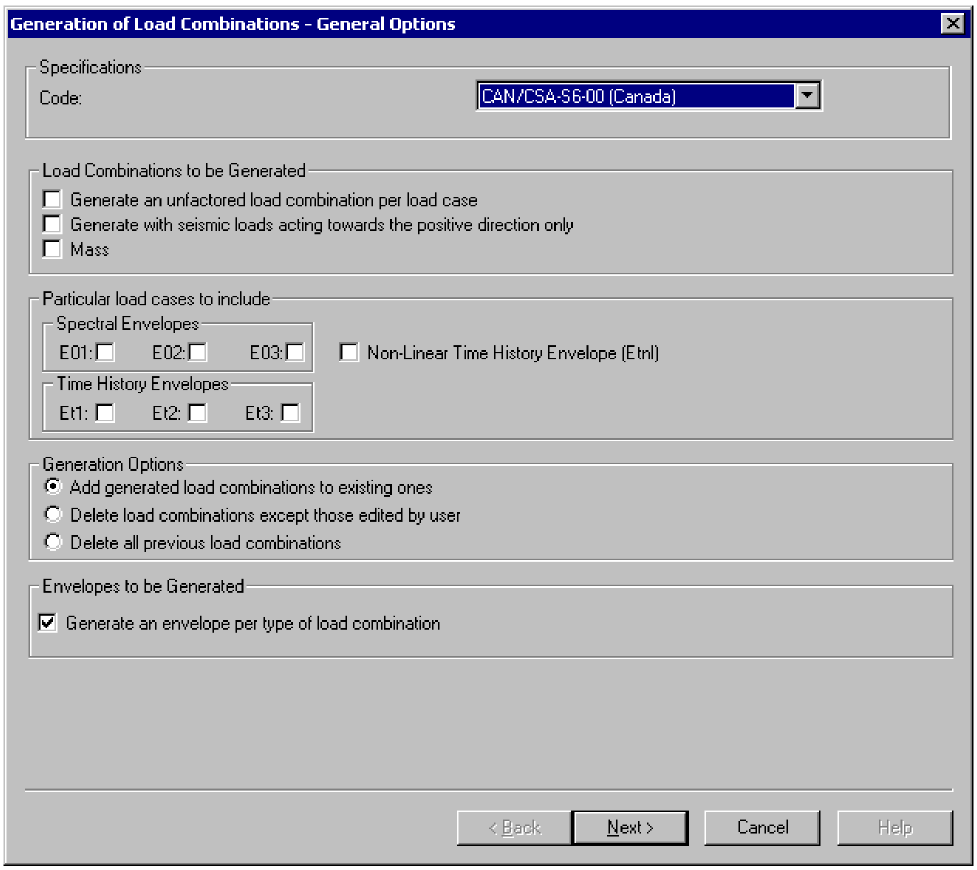
Stage 9 – Long Term: There is no extra load at this stage. However, include virtual load cases.



Generation of Other Load Combinations

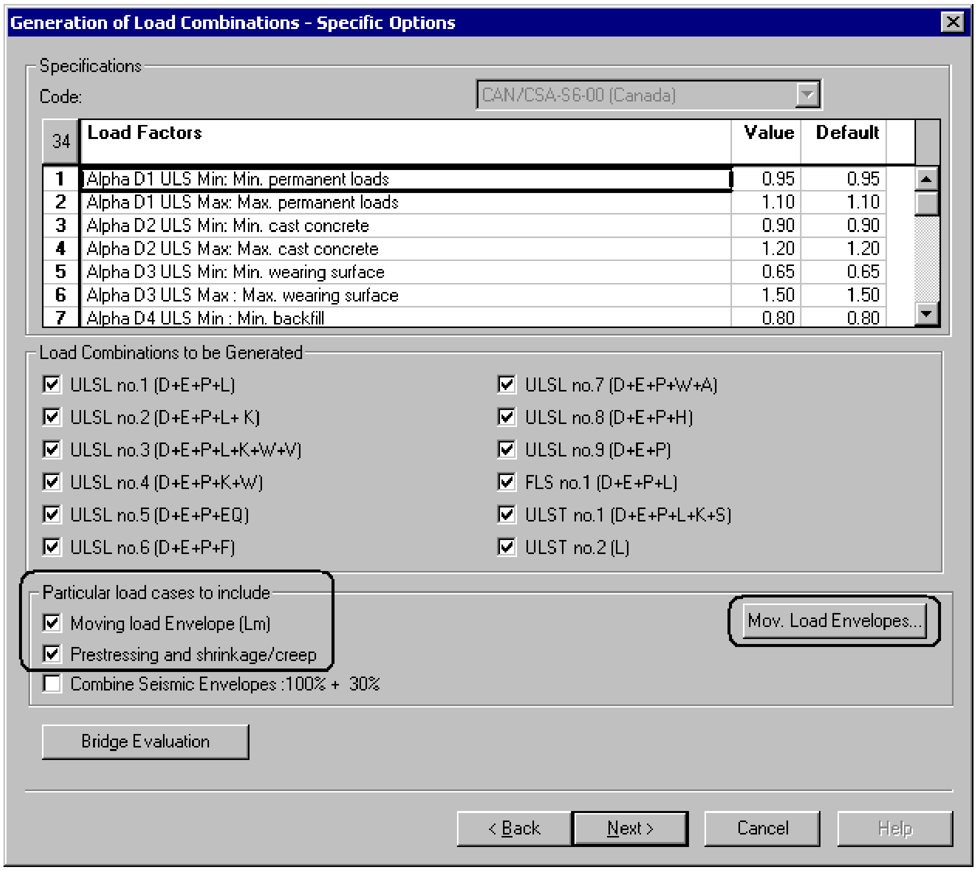
- Use the **Load Combination Generation Wizard** to generate load combinations per S6-00 standard.

- In the **General Options** page of the Wizard, select code S6-00 and activate the option "Add generated load combinations to existing ones" to avoid overwriting construction stages.



- Press *Next.*

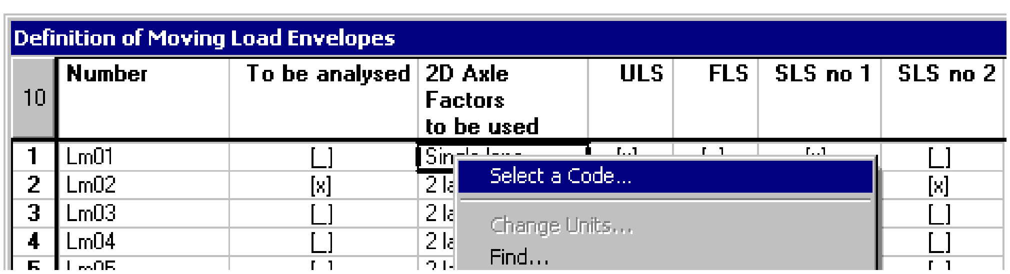
- In the **Specific Options** page, activate particular load cases.



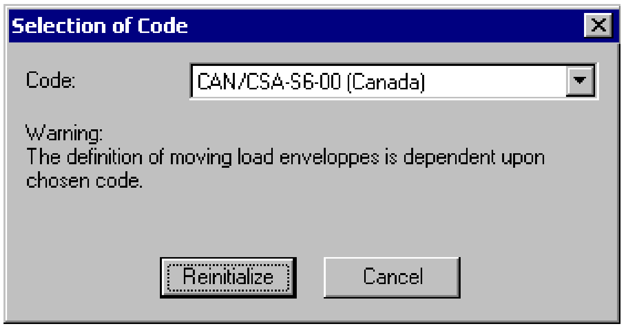
- Press the **Mov. Load Envelopes** button to open the Envelopes spreadsheet for moving load analysis.

This spreadsheet shows the required load combinations and 2D axle factors, as per code. The moving load envelopes must be activated in this spreadsheet.

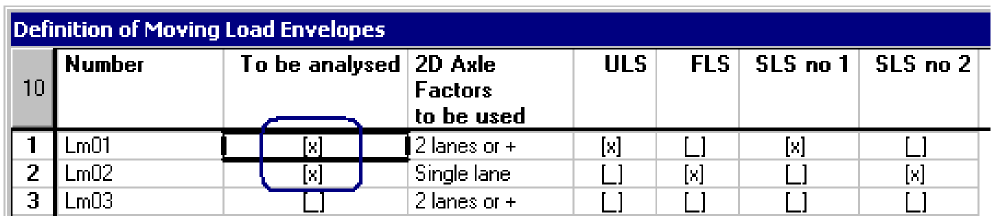
- Click in any cell, right click to open contextual menu, and select the function **Select a Code**.



- Select code S6-00 in the drop down list box and press the button "Reinitialize".

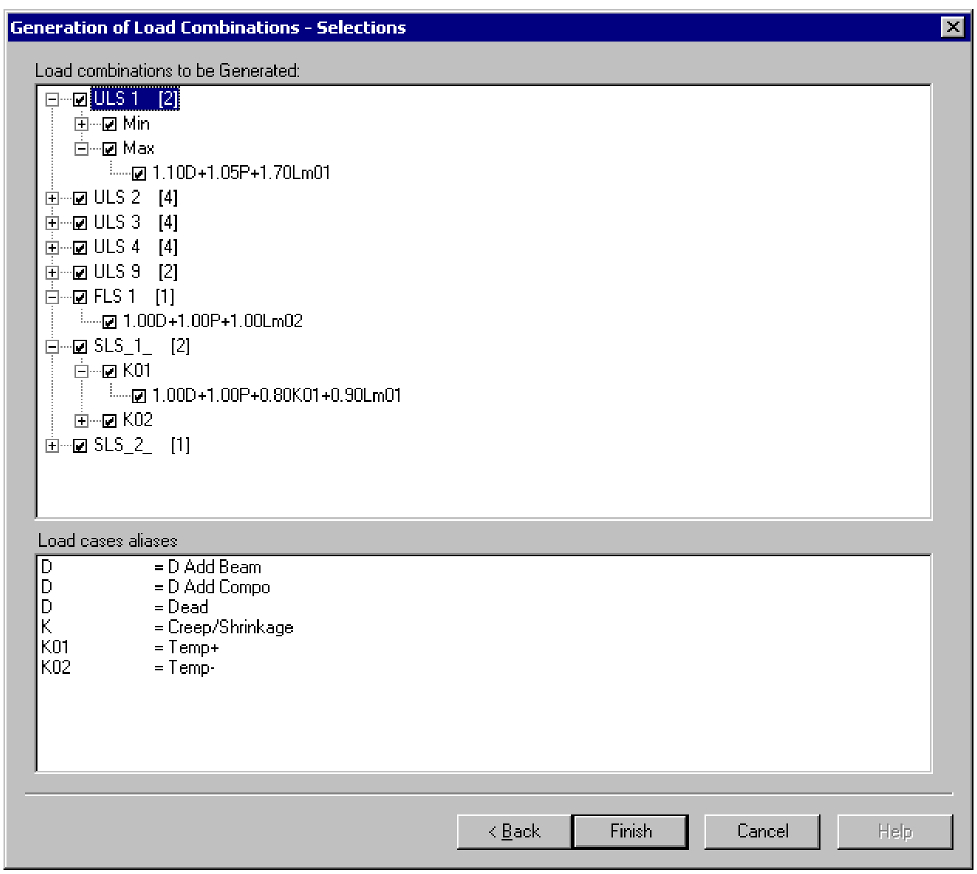


- Activate moving load envelopes Lm01 and Lm02.


Definition of Moving Load Envelopes

10	Number	To be analysed	2D Axle Factors to be used	ULS	FLS	SLS no 1	SLS no 2
1	Lm01	[x]	2 lanes or +	[x]	[ ]	[x]	[ ]
2	Lm02	[x]	Single lane	[ ]	[x]	[ ]	[x]
3	Lm03	[ ]	2 lanes or +	[ ]	[ ]	[ ]	[ ]

- Close the spreadsheet.
- Click *Next* in the **Specific Options** page.



- Press *Finish*.

The definition of load combinations is completed.

Load Combinations

Load Combinations | Load Factors

25	Number	Status	Definition	Stage
1	Stage 3	Construction Stage	Stage 3	3
2	Stage 5	Construction Stage	Stage 5	5
3	Stage 6	Construction Stage	Stage 6	6
4	Stage 8	Construction Stage	Stage 8	8
5	Stage 9	Construction Stage	Stage 9	9
6	ULS 1:max02	ULS 1	1.10D+1.05P+1.70Lm01	0
7	ULS 1:min01	ULS 1	0.95D+0.95P+1.70Lm01	0
8	ULS 2:max05	ULS 2	1.10D+1.05P+1.15K01+1.60Lm01	0
9	ULS 2:max06	ULS 2	1.10D+1.05P+1.15K02+1.60Lm01	0
10	ULS 2:min03	ULS 2	0.95D+0.95P+1.15K01+1.60Lm01	0
11	ULS 2:min04	ULS 2	0.95D+0.95P+1.15K02+1.60Lm01	0
12	ULS 3:max09	ULS 3	1.10D+1.05P+1.00K01+1.40Lm01	0
13	ULS 3:max10	ULS 3	1.10D+1.05P+1.00K02+1.40Lm01	0
14	ULS 3:min07	ULS 3	0.95D+0.95P+1.00K01+1.40Lm01	0
15	ULS 3:min08	ULS 3	0.95D+0.95P+1.00K02+1.40Lm01	0
16	ULS 4:max13	ULS 4	1.10D+1.05P+1.25K01	0
17	ULS 4:max14	ULS 4	1.10D+1.05P+1.25K02	0
18	ULS 4:min11	ULS 4	0.95D+0.95P+1.25K01	0
19	ULS 4:min12	ULS 4	0.95D+0.95P+1.25K02	0
20	ULS 9:max16	ULS 9	1.35D+1.05P	0
21	ULS 9:min15	ULS 9	1.35D+0.95P	0
22	FLS 117	FLS 1	1.00D+1.00P+1.00Lm02	0
23	SLS_1_18	SLS 1	1.00D+1.00P+0.80K01+0.90Lm01	0
24	SLS_1_19	SLS 1	1.00D+1.00P+0.80K02+0.90Lm01	0
25	SLS_2_20	SLS 2	0.90Lm02	0

- Close the spreadsheet.

If you don't want to analyse some load combinations right now, change their statuses to "Not required".

You are now ready to define and place prestressing cables in the continuous beams.

Rebar Placement Window

You must open the *Rebar Placement* window to define cable groups and place cables within continuous system. This window has its own menus and is composed of an elevation view of continuous system. You can create cross-sections, display dimensions and rebars, and also display force and resistance diagrams.

- Activate the “Rebar Placement” mode on Activation toolbar and double click on the continuous system.

VisualDesign will take a few seconds to open the window.

View Options

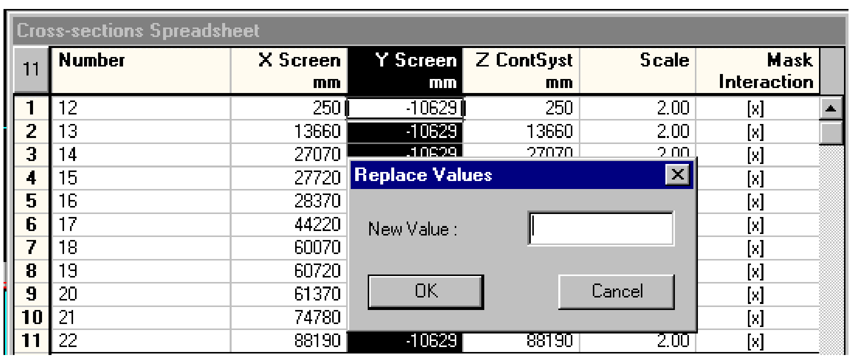
- Open the **View Options** dialog box. Check the *General* and *Dimensions* roots the **Rebar Placement** tab. Press OK.

Cross-Sections

- Generate cross-sections: Select **Automatic Generation of Cross-Sections** in **Rebar Placement** menu. VisualDesign generates cross-sections at each support and at mid-spans.

Editing Cross-Sections

- To move all cross-sections, select the **Cross-sections** spreadsheet in **Rebar Placement** menu. Select column “Y screen” by clicking on its title and right click. Choose function **Replace** in the contextual menu and enter another value. Press Ok.



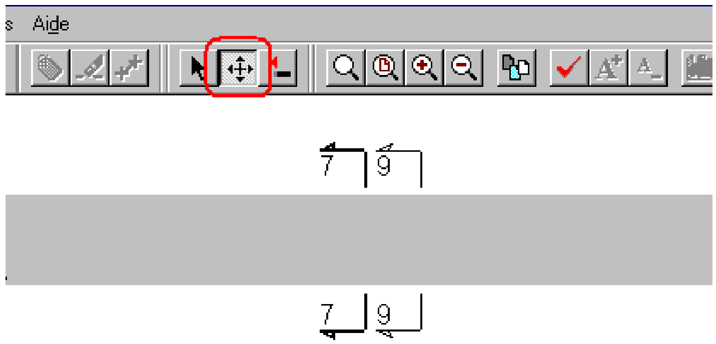
Use the **Zoom** and **Dynamic Pan** functions to increase and move the image on the screen.

To delete a cross-section on screen:

Click on the cross-section outline to highlight it and press the [Delete] key.

To move a line of cut along with the cross-section:

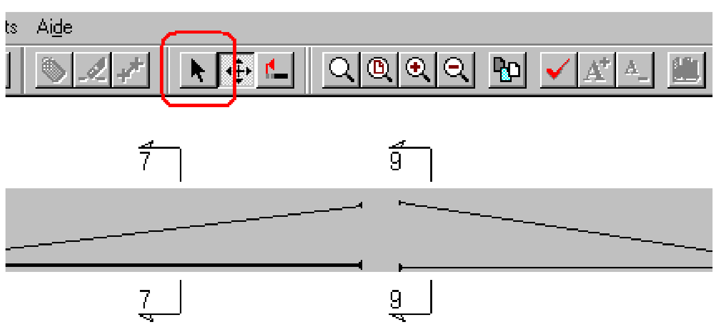
- Activate the **Move** function, as shown below, and click once on a line of cut to highlight it. While keeping down the mouse button, glide the line of cut along continuous system. Release the mouse button.

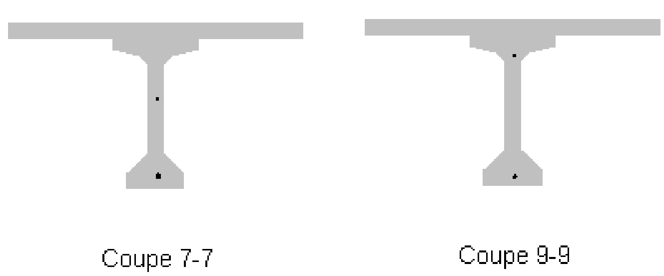


- Go back to **Cursor** mode (located left of the **Move** icon). If you forget to do that, elements that will be touched by your cursor will move. If it happens, use the **Undo** function.

To move the cross-section only:

- Activate the **Move** function and click on the cross-section outline to highlight it. While keeping down the mouse button, glide the cross-section. Release the mouse button. Go back to **Cursor** mode.





Defining Cable Groups and Layouts

Cable Groups

You must define at least one group of cables for a prestressed concrete project (pre-tension or post-tension). In our example, we have many groups because each of them has a different layout.

- Go to **Rebar Placement** menu and select **Cable Groups and Layouts**. Insert lines in the **Groups** spreadsheet. Double click in the "Number" cell and give a name to each group.

X beg and *X end* represent the transverse end positions of sheath(s) or strands according to continuous system local x-axis, for post-tensioning and pre-tensioning.

Cable Groups and Layouts Spreadsheet

Groups | Layouts

6	Number	Post-tensioning Mechanism	Number in width	X beg. mm	X end mm	Number of strands/sheath	Strand
1	Span1-flat	Null	1	-5	5	26	G270:1/2
2	Span1-up	Null	1	-5	5	14	G270:1/2
3	Span3-flat	Null	1	-5	5	26	G270:1/2
4	Span3-up	Null	1	-5	5	14	G270:1/2
5	Span2-flat	Null	1	-5	5	42	G270:1/2
6	Span2-up	Null	1	-5	5	22	G270:1/2

Cable Groups and Layouts Spreadsheet

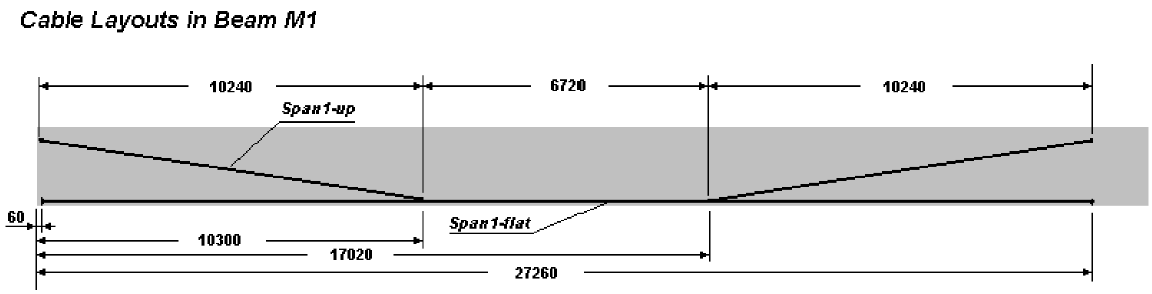
Groups | Layouts

6	Layout	Jacking %	Delta left mm	Delta right mm	Stage	Factor Development Length
1	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00
2	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00
3	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00
4	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00
5	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00
6	Internal with grout	75.00	0	0	Initial Pre-tensioning	50.00

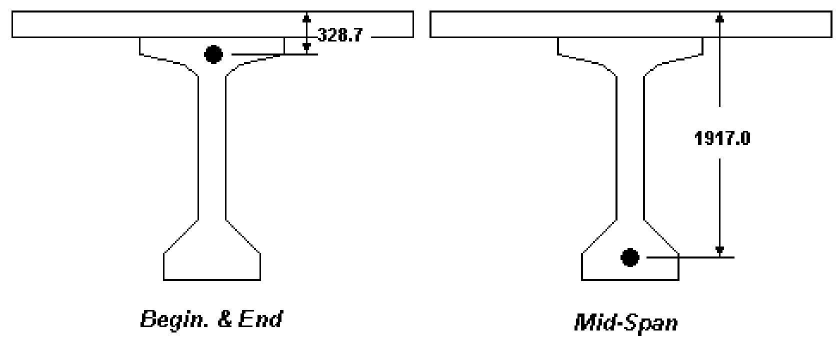
Delta left and *Delta right* are measured from the cone penetration at the left and right end of continuous system (z-axis), after post-tensioning.

Cable Layouts

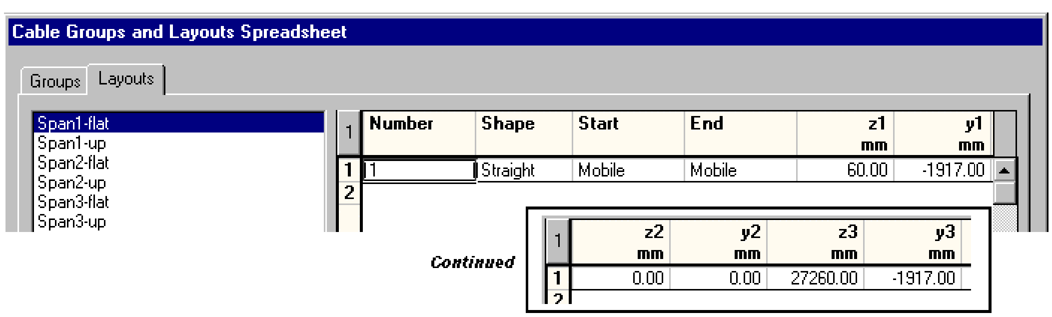
Here are the desired cable layouts for beam M1.



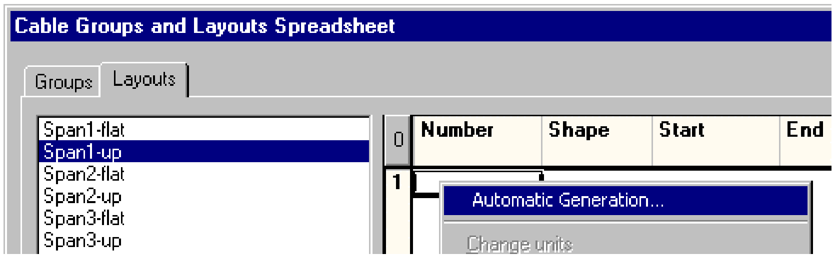
Cable positions (mm) are as follows:



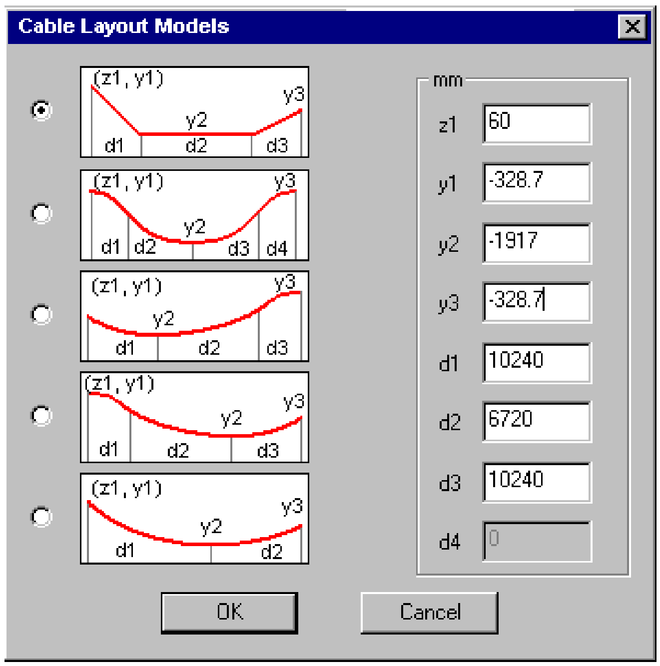
Select the **Layouts** tab. In the left part, highlight the name of the first group and insert a line in the right part. This cable layout is straight and will be positioned at a distance of 1917mm below the top of the composite section. Columns "Start" and "End" represent the cable endings. In our case, the cable endings are mobile so, the *Mobile* option is selected.



- Now, highlight the *Span1-up* group of cables. To help in defining this layout, we are going to use a tool. To call up this tool, click once in any cell (spreadsheet) and right click. Select the **Automatic Generation** function in contextual menu. The **Cable Layout Models** dialog box will open.



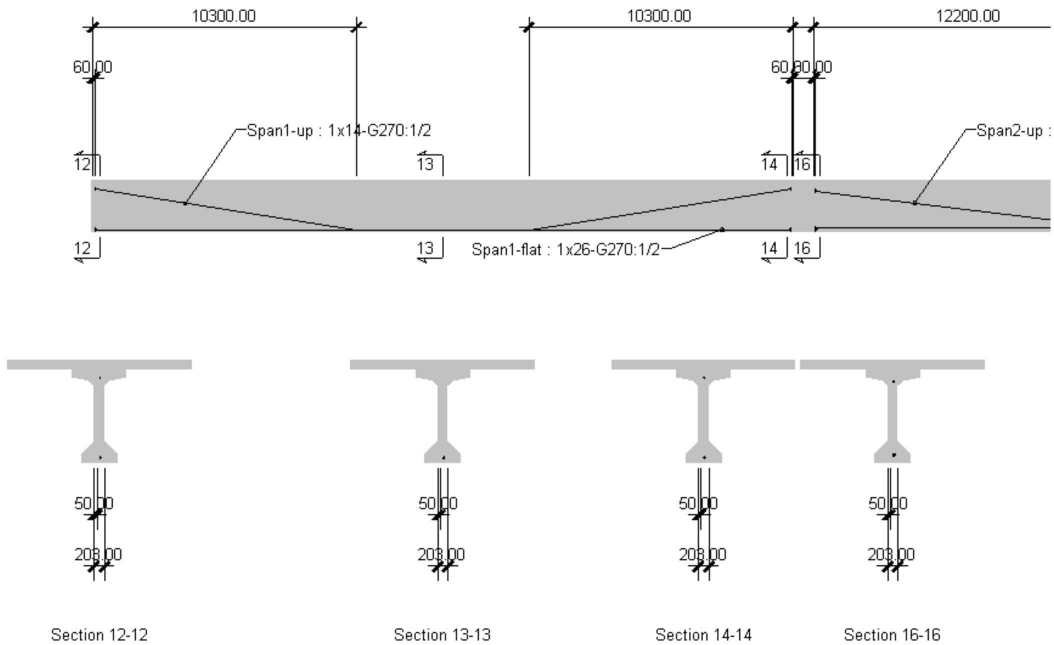
- Activate the model that corresponds to the layout and enter coordinates.



- Press OK.
- You will go back to the **Cable Groups and Layouts** spreadsheets.
- Give a number to each segment and select appropriate cable endings.

3	Number	Shape	Start	End	z1 mm	y1 mm	z2 mm	y2 mm	z3 mm	y3 mm
1	2	Straight	Mobile	Continuous	60.00	-328.70	0.00	0.00	10300.00	-1917.00
2	3	Straight	Continuous	Continuous	10300.00	-1917.00	0.00	0.00	17020.00	-1917.00
3	4	Straight	Continuous	Mobile	17020.00	-1917.00	0.00	0.00	27260.00	-328.70

- Follow the same procedure to define other cable layouts (2 layouts/beam) for beams M3 and M5.

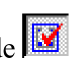
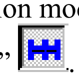


- Close the *Rebar Placement* window.

You are ready to analyse the prestressed composite beam. VisualDesign will verify the resistance with the prestressing cables. It will add main rebars and stirrups where needed.

Design

- Click on the **Analysis and Design**  icon on Tools toolbar or select **Analysis and Design** in **Analysis** menu. The **Design** dialog box will appear on your screen. Press the “Analyse” button. Close the dialog box when design is completed.

The *Rebar Placement* activation mode  will be automatically activated along with the “Continuous System” .

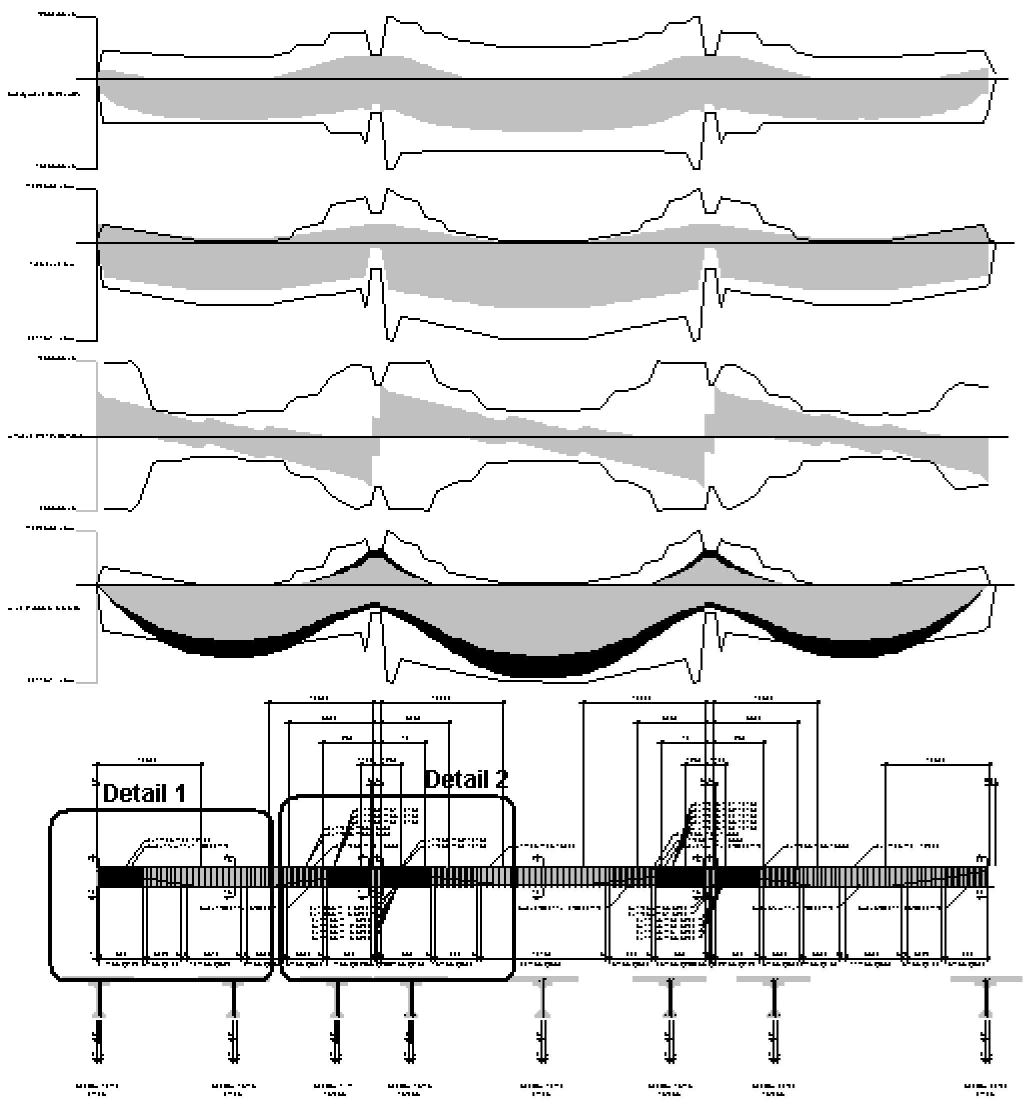
Partial Results

Rebar Placement

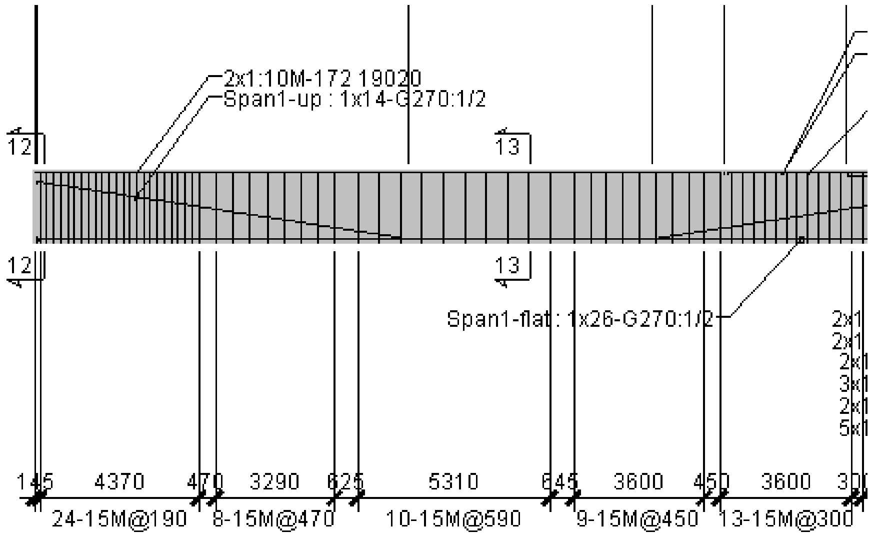
- Double click on the continuous system to open the *Rebar Placement* window or click once and press the **Properties** icon. You will notice that VisualDesign has designed and placed stirrups in the continuous beam, according to Code S6-00 and the *General* method.

Display Force and Resistance Diagrams

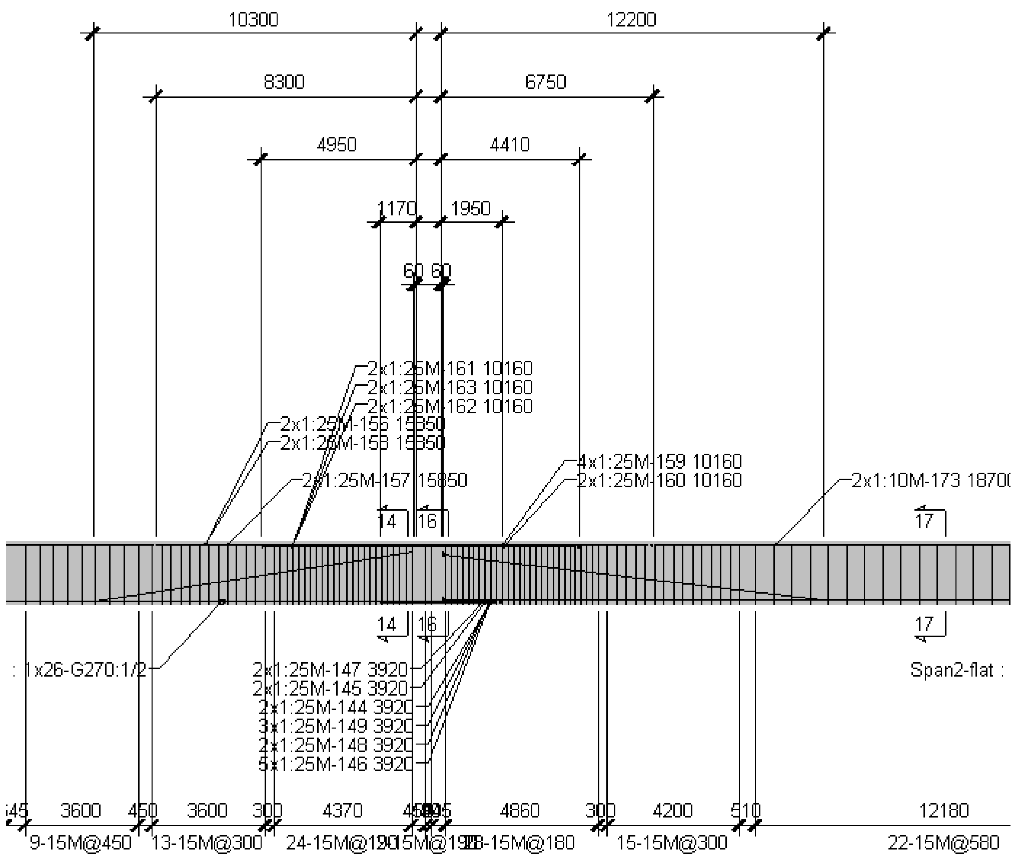
- Select the **View Options** dialog box and expand the *Beam Diagrams* root of **Rebar Placement** tab. Check the boxes corresponding to forces and resistances diagrams that you want to display, as shown below.



Detail 1



Detail 2



Reinforcement and Cables

Double click on any longitudinal rebar, stirrup sequence or cable to consult placement data. The appropriate spreadsheet will open.

Editing Elements

If you modify rebars placement data through spreadsheet (changing rebars dimensions, spacing, etc.), VisualDesign will automatically recalculate new forces and resistances diagrams and redraw them.

We are going to consult detailed results (graphical and numerical) for construction stage load combinations and others. These results are available through the **Results** menu of *Rebar Placement* window.

Graphs

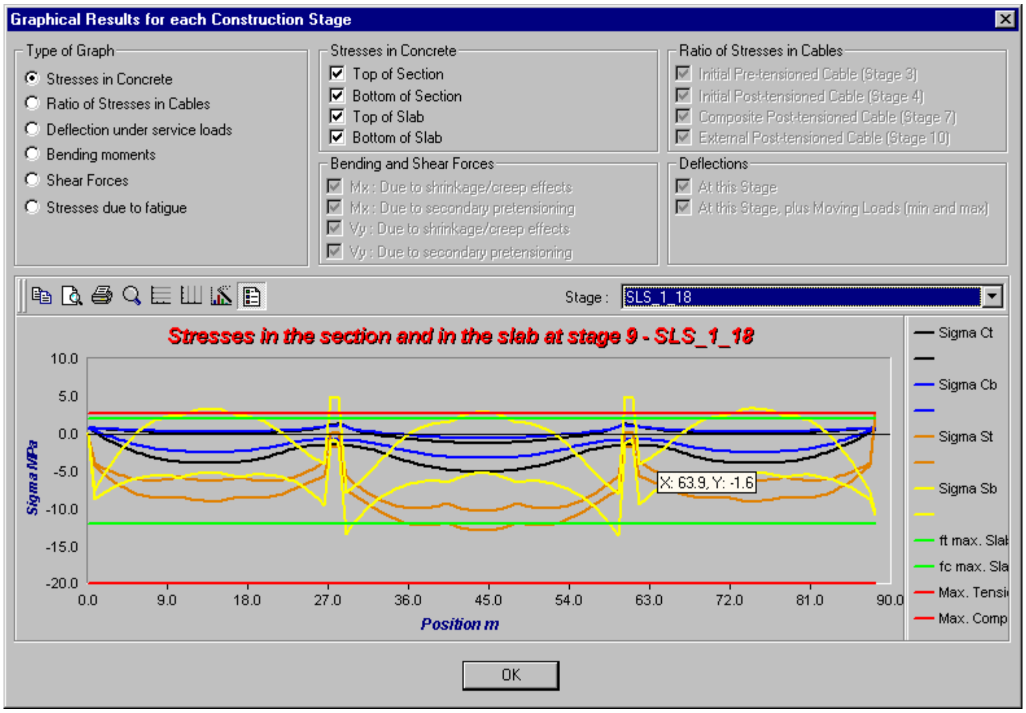
- Select the **Graphs** heading in **Results** menu or press the **Graphs** icon  on View toolbar.
- Activate a type of graph in the **Graphic Results** dialog box shown below. Then, select a serviceability load combination in the drop-down list box.

Note:

Place the cursor on any point on the curve (or elsewhere) and coordinates will be displayed next to your cursor.

To move the legend out of the box, double click on it.

Use the **Graphs** toolbar that is supplied with this dialog box to change the look of the graph, modify the x-axis and y-axis subdivisions, etc. Use the **Print Preview** and **Print** functions to print the graph.



Numerical Results

Prestress Loss in Cables

- Go to **Results** menu and select **Prestress Loss in Cables**. This spreadsheet includes prestress losses <u>AND</u> gains for each construction stage. Total loss, Δp, is available at the last column.

Loss of Prestress in Cables Spreadsheet

6	Number	Stage	ES3 MPa	Δp3 MPa	ES5 MPa	REL5 MPa	CR5 MPa	SH5 MPa	Δp5 MPa	ES6 MPa
1	Span1-flat	Initial Pre-tensioning	-141.91	-141.91	28.23	-16.72	-120.11	-6.67	-115.26	-0.12
2	Span1-up	Initial Pre-tensioning	-108.40	-108.40	16.76	-16.72	-84.06	-6.67	-90.68	-0.13
3	Span2-flat	Initial Pre-tensioning	-207.19	-207.19	43.34	-16.72	-175.36	-6.67	-155.40	-0.43
4	Span2-up	Initial Pre-tensioning	-160.86	-160.86	26.60	-16.72	-126.28	-6.67	-123.06	-0.40
5	Span3-flat	Initial Pre-tensioning	-141.64	-141.64	27.81	-16.72	-121.55	-6.67	-117.12	-0.13
6	Span3-up	Initial Pre-tensioning	-108.41	-108.41	16.84	-16.72	-84.89	-6.67	-91.44	-0.13

Loss of Prestress in Cables Spreadsheet

6	REL6 MPa	CR6 MPa	SH6 MPa	Δp6 MPa	ES8 MPa	Δp8 MPa	ES9 MPa	REL9 MPa	CR9 MPa	SH9 MPa	Δp9 MPa	Δp MPa
1	-0.42	8.38	-0.61	7.22	2.10	2.10	0.00	-19.09	-0.88	-38.62	-58.60	-306.46
2	-0.42	4.24	-0.61	3.08	2.23	2.23	0.00	-19.09	11.81	-38.62	-45.91	-239.68
3	-0.42	13.12	-0.61	11.65	0.29	0.29	0.00	-19.09	7.72	-38.62	-50.00	-400.65
4	-0.42	7.16	-0.61	5.72	1.60	1.60	0.00	-19.09	21.43	-38.62	-36.28	-312.89
5	-0.42	8.41	-0.61	7.24	2.05	2.05	0.00	-19.09	-1.37	-38.62	-59.08	-308.56
6	-0.42	4.26	-0.61	3.10	2.23	2.23	0.00	-19.09	11.63	-38.62	-46.08	-240.60

To know more about the type of prestress losses or gains, refer to On-Line Help, *Chapter 11*, at heading *Results*.

Stresses in Concrete and Prestressing Cables

- Go to **Results** menu and select **Stresses** / **Stage *X***.

For each construction stage, you will find minimum and maximum stresses in concrete (top and bottom of section and slab). Ratio of stresses is also available for prestressing cables. Minimum and maximum deflections are also computed (with or without moving loads).

Stresses under Service Loads Spreadsheet : Stage 6

109	Number	Position m	Section Top σss min MPa	Section Top σss max MPa	Section Bottom σsi min MPa	Section Bottom σsi max MPa	Cable Initial Pre-tension σ/fpu Min.	Cable Initial Pre-tension σ/fpu Max.	Deflection Stage mm	Max Deflection Stage+Truck mm	Min Deflection Stage+Truck mm
1	M1	0.00	-0.07	-0.07	0.07	0.07	0.00	0.00	0.11	0.11	0.11
2	M1	1.36	-4.19	-4.19	-9.01	-9.01	0.62	0.64	9.30	9.30	9.30
3	M1	2.72	-4.82	-4.82	-8.37	-8.37	0.62	0.64	17.88	17.88	17.88
4	M1	4.08	-5.27	-5.27	-7.91	-7.91	0.62	0.64	25.69	25.69	25.69
5	M1	5.44	-5.54	-5.54	-7.63	-7.63	0.62	0.64	32.72	32.72	32.72
6	M1	6.80	-5.62	-5.62	-7.55	-7.55	0.62	0.64	38.89	38.89	38.89

Stresses - Serviceability Limits States

Stresses under Service Loads Spreadsheet : Stage 11

SLS_1_18 | SLS_1_19

109	Number	Position m	Section Top σss min MPa	Section Top σss max MPa	Section Bottom σsi min MPa	Section Bottom σsi max MPa	Slab Top σds min MPa	Slab Top σds max MPa	Slab Bottom σdi min MPa	Slab Bottom σdi max MPa
1	M1	0.00	-0.10	-0.08	0.09	0.15	0.83	0.86	0.84	0.86
2	M1	1.36	-4.83	-4.29	-8.10	-6.40	-0.07	0.71	0.25	0.79
3	M1	2.72	-6.04	-5.05	-7.10	-4.01	-0.85	0.58	-0.29	0.70
4	M1	4.08	-6.97	-5.61	-6.33	-2.09	-1.50	0.47	-0.74	0.62
5	M1	5.44	-7.68	-5.98	-5.78	-0.46	-2.11	0.36	-1.16	0.54
6	M1	6.80	-8.12	-6.15	-5.46	0.72	-2.60	0.27	-1.49	0.48

Intermediate Results

- Go to **Results** menu and select **Intermediate Results.**

This spreadsheet includes properties and results on shrinkage and creep effects in the slab and section, according to (cumulative) days.

Prestressed Concrete Intermediate Results Spreadsheet

25	Stage day	Member	Shape	f'ci section MPa	Eci section MPa	Ix section 10e6mm4	Area section mm²	Compression Limit Section MPa	Tension Limit Section MPa	Creep Section	Shrinkage Section
16	67	M1	Type VI	54.53	30969.62	305220.75	699643.02	32.72	1.48	0.000000	0.000000
17	67	M2	Type VI	54.53	30969.62	305220.75	699643.02	32.72	1.48	0.000000	0.000000
18	67	M3	Type VI	54.53	30969.62	305220.75	699643.02	32.72	1.48	0.000000	0.000000
19	67	M4	Type VI	54.53	30969.62	305220.75	699643.02	32.72	1.48	0.000000	0.000000
20	67	M5	Type VI	54.53	30969.62	305220.75	699643.02	32.72	1.48	0.000000	0.000000
21	10000	M1	Type VI	58.31	31772.49	305220.75	699643.02	20.00	2.83	0.736720	-0.000206
22	10000	M2	Type VI	58.31	31772.49	305220.75	699643.02	20.00	2.83	0.736720	-0.000206
23	10000	M3	Type VI	58.31	31772.49	305220.75	699643.02	20.00	2.83	0.736720	-0.000206

Prestressed Concrete Intermediate Results Spreadsheet

25	f'ci slab MPa	Eci slab MPa	Ix composite 10e6mm4	Area composite mm²	ybi mm	yhi mm	Creep Slab	Shrinkage Slab	Compression Limit Slab MPa	Tension Limit Slab MPa
16	32.72	25646.77	605226.64	1211317.78	1349	680	0.000000	0.000000	12.60	0.92
17	32.72	25646.77	605226.64	1211317.78	1349	680	0.000000	0.000000	12.60	0.92
18	32.72	25646.77	605226.64	1211317.78	1349	680	0.000000	0.000000	12.60	0.92
19	32.72	25646.77	605226.64	1211317.78	1349	680	0.000000	0.000000	12.60	0.92
20	32.72	25646.77	605226.64	1211317.78	1349	680	0.000000	0.000000	12.60	0.92
21	34.98	26268.67	631838.86	1294919.85	1386	643	2.538424	-0.000245	12.00	2.19
22	34.98	26268.67	631838.86	1294919.85	1386	643	2.538424	-0.000245	12.00	2.19
23	34.98	26268.67	631838.86	1294919.85	1386	643	2.538424	-0.000245	12.00	2.19

General Results

- Go to **Results** menu and select the **General Results** spreadsheet. This spreadsheet includes many reinforced concrete design results and some prestressed concrete design results also.

If a line is marked in yellow, it means that there are one or more parameters in this line not following the Code requirements.

General Results Spreadsheet - Continuous System : S_4

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

205	Mcr kN.m	d mm	dv mm	bw mm	As mm²	ρ %	ωp	ϕp.fps.Aps kN	εx
16	8386.91	1915	1882	3600	0.00	0.00	0.02	6808.97	0.001686
17	8427.99	1917	1884	3600	0.00	0.00	0.02	6837.25	0.001887
18	8427.99	1917	1884	3600	0.00	0.00	0.02	6837.25	0.001958
19	8427.99	1917	1884	3600	0.00	0.00	0.02	6837.25	0.002000
20	8427.99	1917	1884	3600	0.00	0.00	0.02	6837.25	0.001986

General Results Spreadsheet - Continuous System : S_4

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

205	Z mm	Vfy Max kN	Vfy Min kN	Design Load %	εx	θ °	β	Vc kN
11	6855	166.05	-883.85	94.54	0.001365	39.32	0.16	299.57
12	7536	235.30	-803.82	88.06	0.001520	40.80	0.16	296.92
13	8216	291.34	-740.64	81.34	0.001646	41.49	0.16	296.72
14	8897	347.26	-677.98	78.98	0.001718	41.95	0.15	297.92
15	9577	429.22	-585.51	72.62	0.001726	41.97	0.15	305.56
16	10258	498.92	-509.10	61.61	0.001686	41.81	0.15	312.68
17	10938	321.60	-681.00	85.06	0.001887	42.58	0.15	300.20
18	11619	376.55	-621.02	78.71	0.001958	42.84	0.14	293.11
19	12299	432.52	-561.69	71.72	0.002000	43.00	0.14	290.04

General Results Spreadsheet - Continuous System : S_4

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force

205	Vc kN	Vs kN	Vr kN	d mm	dv mm	ϕpVp kN
11	299.57	635.30	934.87	1731	1699	-232.48
12	296.92	615.92	912.84	1768	1735	-232.48
13	296.72	613.80	910.53	1805	1772	-232.48
14	297.92	560.49	858.42	1841	1808	-232.48
15	305.56	500.75	806.31	1878	1845	-232.48
16	312.68	513.58	826.26	1915	1882	-232.48
17	300.20	500.44	800.64	1917	1884	0.00
18	293.11	495.86	788.97	1917	1884	0.00
19	290.04	493.09	783.13	1917	1884	0.00

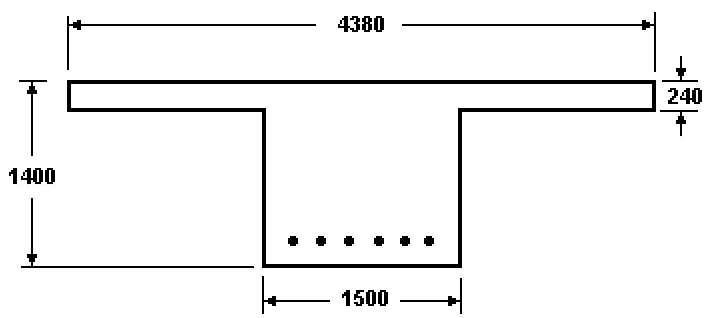
Continuous Beams with Post-tension

Description of the Project

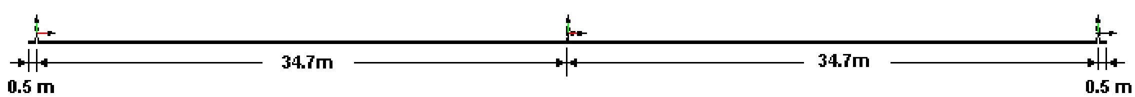
A continuous pre-tensioned beam is composed of a T beam. The specified compressive strength of concrete is 50MPa. Six sheaths will be placed in one layer within continuous beams. Each sheath is composed of 19-3/5" strands of grade 1860 MPa. The diameter of each sheath is 75mm and grout will be injected.

Jacking will be done at 80% of maximum stress fpu. Post-tension will be sequential and applied at initial post-tensioning stage.

Standard T-Beam with Post-Tension



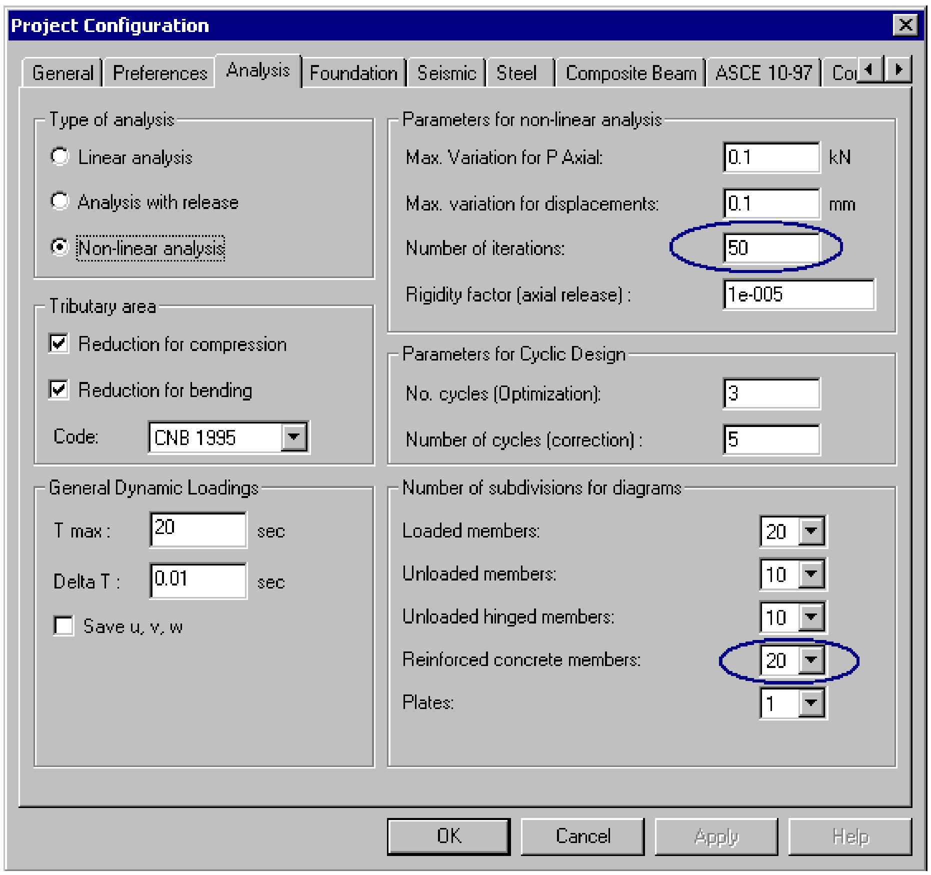
The beam spans are as follows:



Project Configuration

- Go to the **Project Configuration** dialog box (**File** menu) and select the **Preferences** tab. In section *Dialog Box Display*, disable this function for nodes and members to get a quicker editing. Press OK.

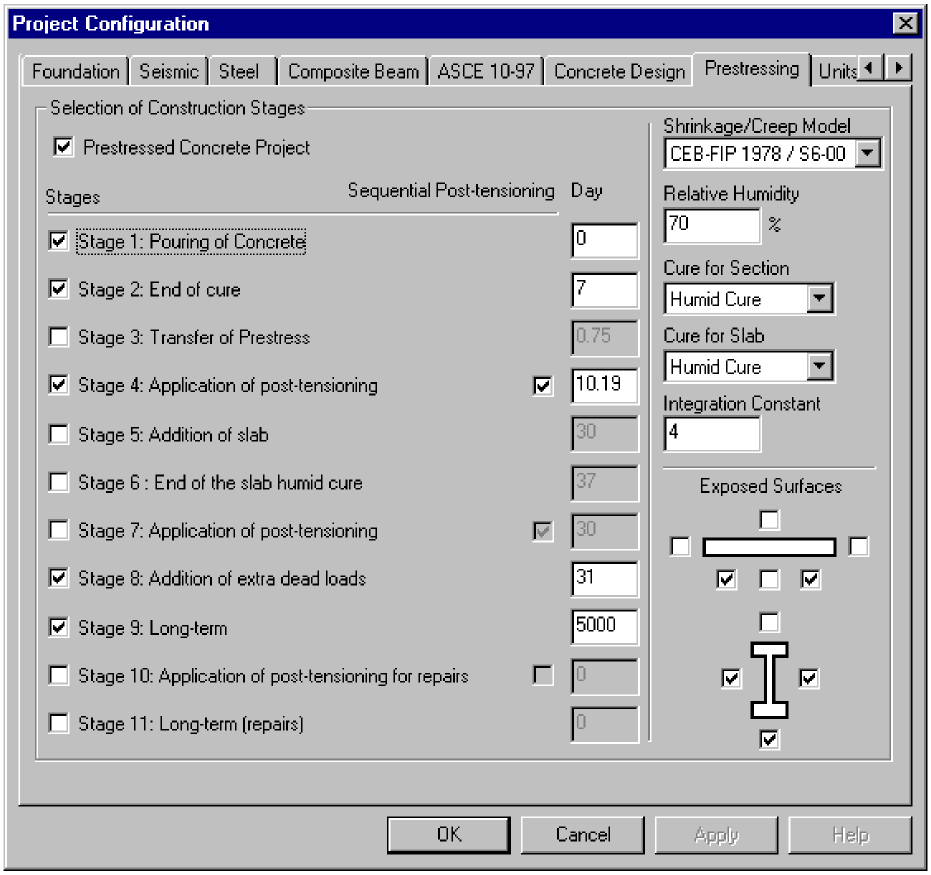
- Now, select the **Analysis** tab and activate a non-linear type of analysis. Increase the number of iterations to 50 for the non-linear analysis. We will set number of subdivisions to 20 for concrete beams and members will be split in ten.



- Select the **Concrete Design** tab. Choose the *General method* for the design of prestressed concrete elements.

- Finally, select the **Prestressing** tab. You must check the *Prestressed Concrete Project* box to activate construction stages below.

  - Check stages 1, 2, 4, 8 & 9 and *Sequential Post-tensioning* at stage 4. Specify the day (cumulative) when each construction stage will be applied. Select a model for shrinkage & creep. The *Exposed Surfaces* section is important to compute concrete shrinkage and creep effects.

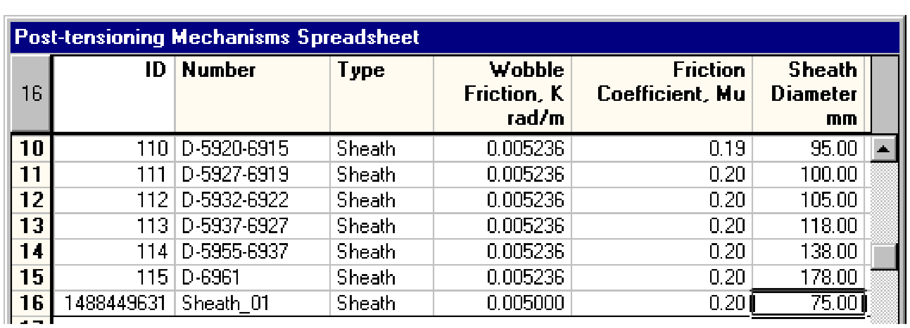


- Press OK to save data and exit the dialog box.

Post-Tension Mechanism

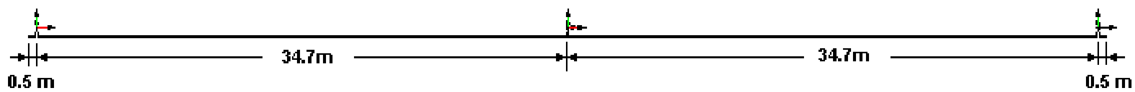
Before selecting the **Post-tensioning Mechanisms** spreadsheet, make sure that the strand steel grade and diameter are included in database (**Common** menu / **Cables**/ **Steel Grades** and / **Strands**)

- Go to **Common** menu / **Cables / Post-Tensioning Mechanisms**. Create a new post-tension mechanism for a 75mm diameter sheath. Insert a line and enter the name *Sheath_01* in the "Number" cell. Select a *Sheath* mechanism in the "Type" cell . Enter a diameter of 75mm. Complete other parameters. Press OK.


Post-tensioning Mechanisms Spreadsheet

16	ID	Number	Type	Wobble Friction, K rad/m	Friction Coefficient, Mu	Sheath Diameter mm
10	110	D-5920-6915	Sheath	0.005236	0.19	95.00
11	111	D-5927-6919	Sheath	0.005236	0.20	100.00
12	112	D-5932-6922	Sheath	0.005236	0.20	105.00
13	113	D-5937-6927	Sheath	0.005236	0.20	118.00
14	114	D-5955-6937	Sheath	0.005236	0.20	138.00
15	115	D-6961	Sheath	0.005236	0.20	178.00
16	1488449631	Sheath_01	Sheath	0.005000	0.20	75.00

Structural Model



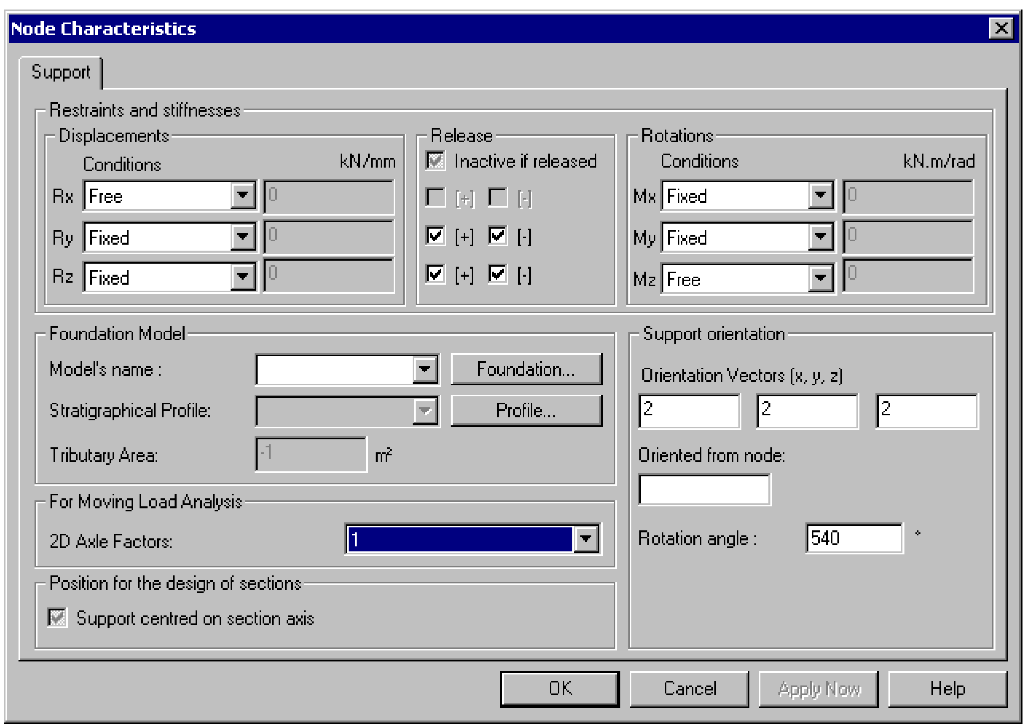
- Select the **Nodes** spreadsheet and insert three lines. Enter nodes coordinates, which are 34.7m apart in the global x-direction. Double-click in the "Type" column and select "Support". Press OK.

- Add members between supports with the **Add** function.

- Select the **Nodes** spreadsheet again and add the end nodes located at 0.5m from ends of continuous beams. Add end members.

- Activate the Support element, select all of them and press the **Properties** icon. Free displacements in the x-direction (Rx) and rotations Mz. Restrain displacements Ry and Rz and rotations Mx and My. Press OK.

If your project includes moving load analysis, select the 2D axles factor that will be applied to supports (2D axles factors must be defined beforehand in the **Loads** menu under **2D Axles Factors**).



- Double-click on the central support and restraint displacement Rx. Press OK.

Personalized T-Beam

The T-Beam is not a composite section; we can create it through the **T Sections** spreadsheet. (If the T sections were to be composite, we would have created it through the AASHTO sections spreadsheet. Refer to On-line Help, Chapter 12 - Topic *T-Sections*.)

- Go to **Common** / **Shapes** / **T Shapes**. Click once on the last line and press [Insert]. Give a name to this section and select a *Concrete* material. Enter values for d, b, t, and w. VisualDesign will automatically calculate other properties.

T Shapes Spreadsheet

351	ID	Metric Designation	Imperial Designation	Material	Canada	US	Europe	Personal	Type
346	12345	ST155X37	ST6X25	Steel	[_]	[x]	[_]	[_]	WT
347	12346	ST190X32	ST7.5X21.45	Steel	[_]	[x]	[_]	[_]	WT
348	12347	ST190X37	ST7.5X25	Steel	[_]	[x]	[_]	[_]	WT
349	12348	ST230X40.7	ST9X27.35	Steel	[_]	[x]	[_]	[_]	WT
350	12349	ST230X52	ST9X35	Steel	[_]	[x]	[_]	[_]	WT
351	2027970292	T1x1400	T1x1400	Concrete	[x]	[x]	[x]	[x]	WT

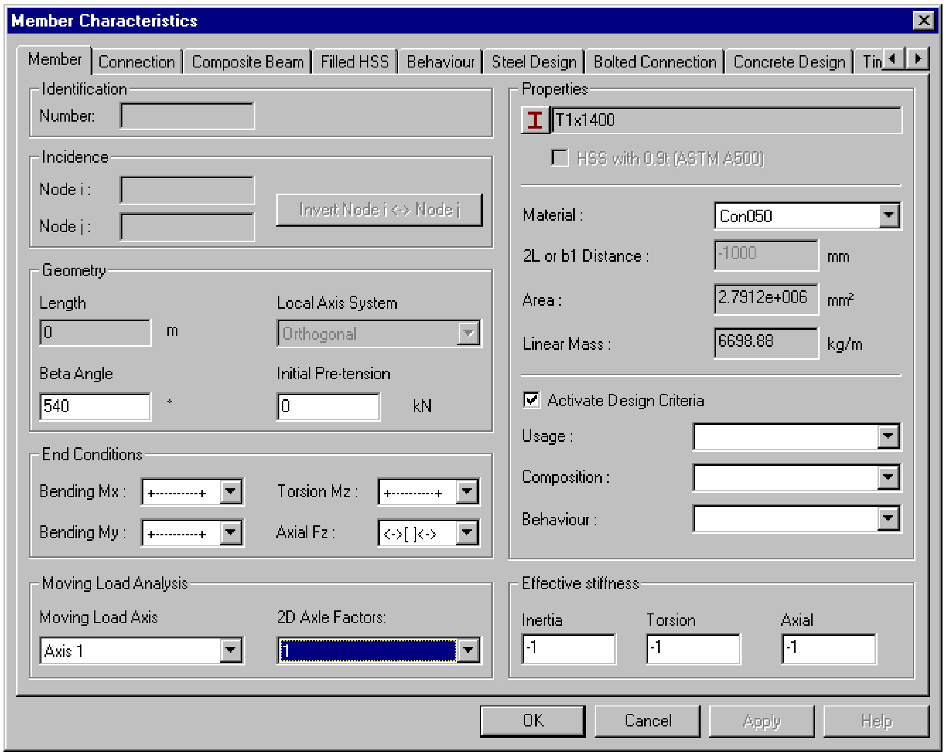
T Shapes Spreadsheet

351	d mm	Nominal d mm	b mm	t mm	w mm	Area mm²	Ix 10e6mm4
346	152.00	155.00	139.00	16.70	17.40	4720.00	10.40
347	191.00	190.00	140.00	15.80	10.40	4060.00	13.70
348	191.00	190.00	143.00	15.80	14.00	4740.00	16.90
349	229.00	230.00	152.00	17.60	11.70	5180.00	25.90
350	229.00	230.00	159.00	17.60	18.10	6630.00	35.20
351	1400.00	1400.00	4380.00	240.00	1500.00	2791199.92	521257.34

Members

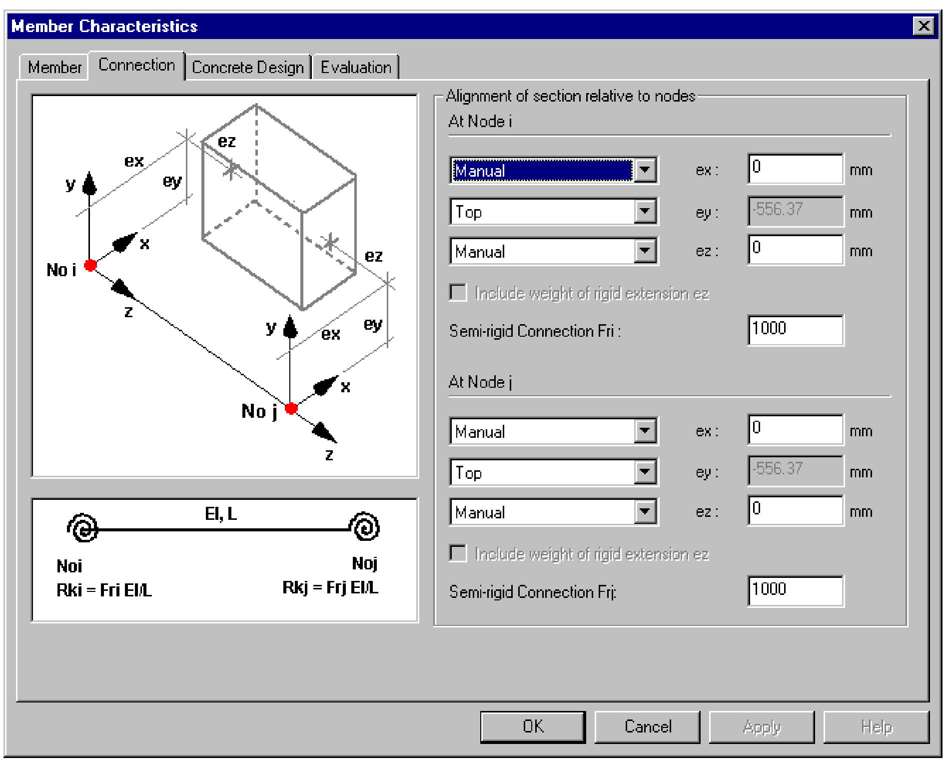
- Activate the member element and select all of them. Press the **Properties** icon to open the **Member Characteristics** dialog box. Click on the I-Beam icon to select the T shape. Then, select a 50MPa concrete material. Activate design criteria. Specify continuous member end conditions (+-----+).

If your project includes moving load analysis, select the moving load axis and specify 2D axles factor. (Refer to the detailed example on 2D Moving Load Analysis)



Alignment

- Go to the **Connection** tab. Align beams at the top.



- Go to the **Concrete Design** tab. Select an option for Vy max. Press OK to save data and close the **Member Characteristics** dialog box.

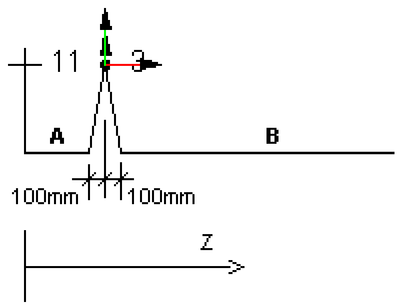
Split Members

Split the members into ten segments using the **Multiple Split** function.

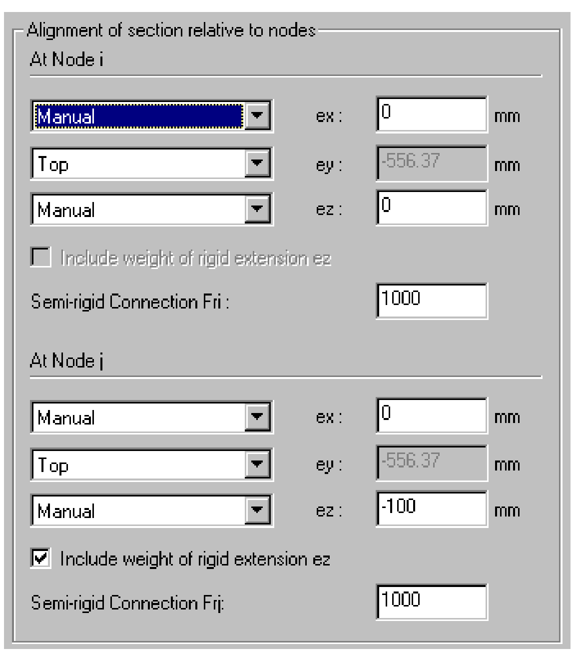
Rigid Extensions "ez"

Rigid extensions must be defined through the **Connection** tab, at each side of a support (where beams connect). VisualDesign needs rigid extensions to calculate rebars and cables development lengths. We will specify 100mm rigid extensions.

Rigid extensions at support #3:



According to member "A" local axis, the rigid extension will be modeled at node j and the value will be negative, as shown below. Activate the option to include the weight of rigid extension.



For member B, rigid extensions will be modeled at node i and j. Values will be respectively +100mm and –100mm.

Do the same to define other rigid extensions (second and third support).

Specifications

The design will be done according to CAN/CSA-S6-00 standard. Go to **Structure/ Specifications/ Concrete** and look at default parameters corresponding to this standard. Modify design options, as you like. The specification will be selected in a further stage, as you will see.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Number	Code	Type of analysis	Maximum Capacity Factor %	Calcul. Method Mr/Vr Positive	Calcul. Meth Mr/Vr Negative
1	A23.3-Design	CAN/CSA-A23.3-95	Design	100.00	Maximize Mr	Maximize Mr
2	A23.3-Vérif.	CAN/CSA-A23.3-95	Verification	100.00	Maximize Mr	Maximize Mr
3	S6-00-Design	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
4	S6-00-Vérif	CAN/CSA-S6-00	Verification	100.00	Maximize Mr	Maximize Mr
5	AASHTO-Design	AASHTO-LRFD-98 Beta	Design	100.00	Maximize Mr	Maximize Mr
6	AASHTO-Vérif.	AASHTO-LRFD-98 Beta	Verification	100.00	Maximize Mr	Maximize Mr

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Epoxy Coated	Longitudinal Optimization	Longitudinal Reinforcement Material	Selection of Longitidunal Rebar
1	[ ]	Weight	G30.18-400R	20M 25M 30M
2	[ ]	Weight	G30.18-400R	20M 25M 30M
3	[ ]	Weight	G30.18-400W	20M 25M 30M
4	[ ]	Weight	G30.18-400R	20M 25M 30M
5	[ ]	Weight	G30.18-400R	# 7 # 8 # 9
6	[ ]	Weight	G30.18-400R	# 7 # 8 # 9

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Slab | Shear Wall

6	Number	Maximum No. of Layers in Tension	Maximum No. of Layers in Compression	Transverse Optimization	Transverse Reinforcement Material	Selection of Transverse Rebar
1	A23.3-Design	4	4	Weight	G30.18-400R	10M 15M
2	A23.3-Vérif.	4	4	Weight	G30.18-400R	10M 15M
3	S6-00-Design	4	2	Weight	G30.18-400W	20M
4	S6-00-Vérif	4	4	Weight	G30.18-400R	10M 15M
5	AASHTO-Design	4	4	Weight	G30.18-400R	# 4 # 5
6	AASHTO-Vérif.	4	4	Weight	G30.18-400R	# 4 # 5

- A maximum of 2 layers of rebars can be placed in the flange of the T-Beam so double-click in the appropriate cell and enter "2".
- Select 20M rebars as transverse reinforcement and press OK.

Continuous Systems

The concrete specification *S6-00 Design* must be selected in the **Continuous Systems** spreadsheet. Concrete cover and other parameters must be specified.

- Go to **Structure / Continuous Systems**. VisualDesign automatically created a continuous system.

- Select the ***S6-00-Design*** specification and enter concrete covers. (When the S6-00 standard is selected, the *Manual* option is automatically chosen. Concrete covers are considered from the outside diameter of stirrups.)

Continuous Systems Spreadsheet

1	Number	Specification	Type	Interaction	Description	Exposure Top	Top Cover mm	Exposure Bottom
1	S_3	S6-00-Design	Beam/Column	Bending	S_3	Manual	60.00	Manual

Continuous Systems Spreadsheet

1	Bottom Cover mm	Exposure Left	Left Cover mm	Exposure Right	Right Cover mm	Crack Control Top kN/m	Crack Control Bottom kN/m	Primary Lateral Resistance
1	50.00	Manual	50.00	Manual	50.00	50000.00	50000.00	[x]

- Press OK to close the spreadsheet.

Load Cases

All load case types and titles that will be used in the project must be defined in the **Load Definition** spreadsheet, including construction stage loads.

- Go to **Loads/ Load Cases** / **Definition**. It is very important to select the right type of load if you plan to use **the Load Combination Generation Wizard**. Here are the load cases that we will need:

Loads Definition

Load Case | Dynamic | Ice

8	Number	Type	Family	Stage	Auto Generation combinaisons
1	Dead	(D1) Prefab Components	N/A	0	[x]
2	Ds Surface	(D3) Wearing Surface	N/A	0	[x]
3	D:Conc deck	(D2) Cast Concrete	N/A	0	[x]
4	D:Barrier	(D1) Prefab Components	N/A	0	[x]
5	K+	(K) Temperature	N/A	0	[x]
6	Diaphragm	(D2) Cast Concrete	N/A	0	[x]
7	D:Curb	(D2) Cast Concrete	N/A	0	[x]
8	K-	(K) Temperature	N/A	0	[x]

Now, apply these load cases graphically to the structure.

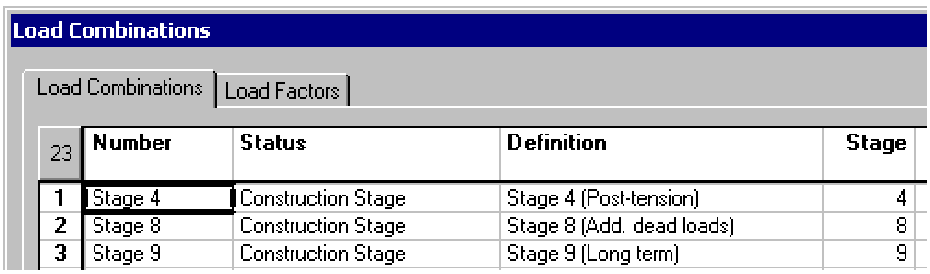
Load Combinations

Construction Stage Load Combinations

Before calling up the **Load Combination Generation Wizard**, we must define construction stage load combinations in the **Load Combination** spreadsheet.

- Go to **Loads** menu and select **Load Combinations / Definition.**

- Insert three lines and define stage load combination numbers, statuses (must be a *Construction Stage* status), and stage numbers.



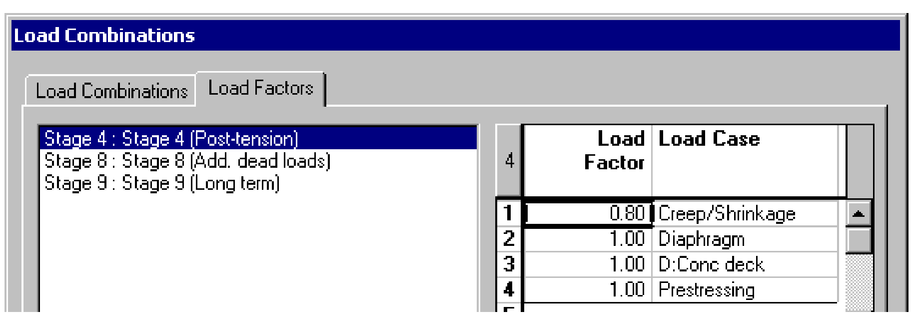
23	Number	Status	Definition	Stage
1	Stage 4	Construction Stage	Stage 4 (Post-tension)	4
2	Stage 8	Construction Stage	Stage 8 (Add. dead loads)	8
3	Stage 9	Construction Stage	Stage 9 (Long term)	9

Load Factors

- Go to the **Load Factors** tab.

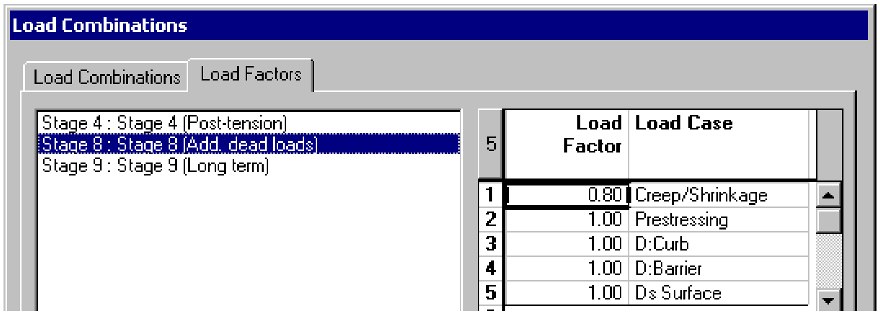
Stage 4 – Application of Post-tensioning

- Highlight *Stage4* load combination in the left part of the dialog box. Place your cursor at line 1 and insert lines. Double click in the *Load Case* cell and select the right type of load case in the drop-down list box. Enter ULS load factors according to CAN/CSA-S6-00.

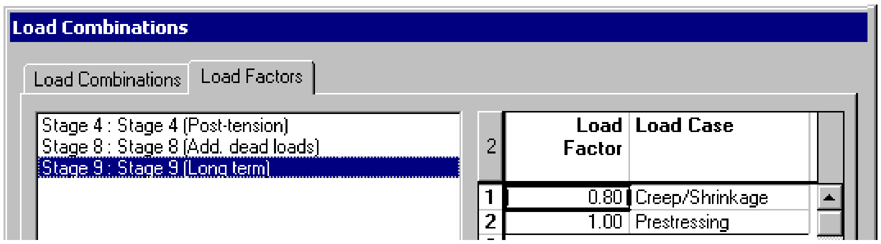


4	Load Factor	Load Case
1	0.80	Creep/Shrinkage
2	1.00	Diaphragm
3	1.00	D:Conc deck
4	1.00	Prestressing

Stage 8 – Additional Dead Load

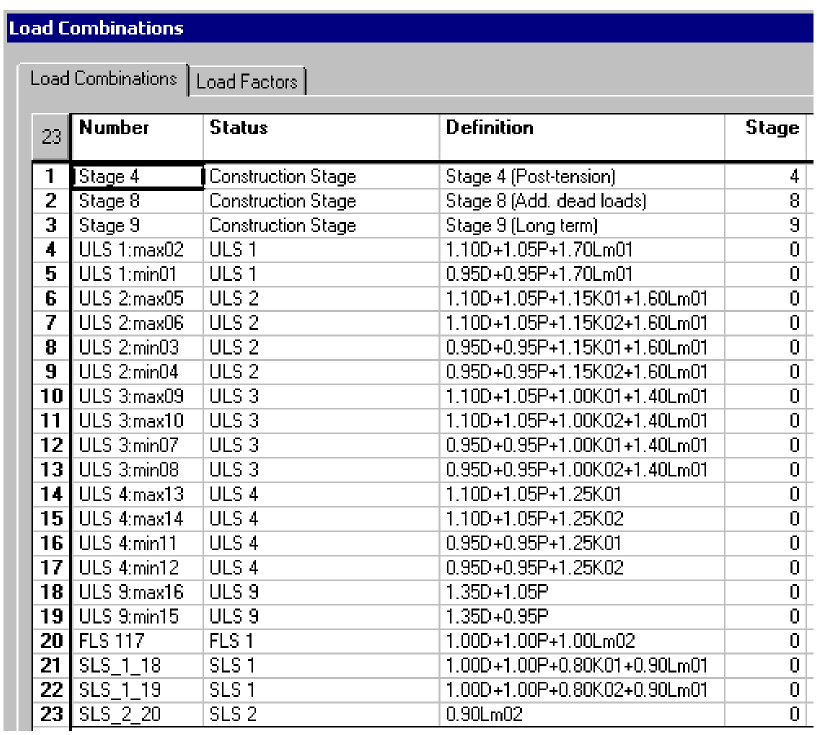


Stage 9 – Long Term



Load Combination Generation Wizard

- Use the **Load Combination Generation Wizard** to generate other load combinations according to S6-00 standard. Include "Prestressing - Shrinkage/Creep" and "Moving Load Envelope" as special loadings. Activate option "Add generated load combinations to existing ones".


Load Combinations

Load Combinations | Load Factors

23	Number	Status	Definition	Stage
1	Stage 4	Construction Stage	Stage 4 (Post-tension)	4
2	Stage 8	Construction Stage	Stage 8 (Add. dead loads)	8
3	Stage 9	Construction Stage	Stage 9 (Long term)	9
4	ULS 1:max02	ULS 1	1.10D+1.05P+1.70Lm01	0
5	ULS 1:min01	ULS 1	0.95D+0.95P+1.70Lm01	0
6	ULS 2:max05	ULS 2	1.10D+1.05P+1.15K01+1.60Lm01	0
7	ULS 2:max06	ULS 2	1.10D+1.05P+1.15K02+1.60Lm01	0
8	ULS 2:min03	ULS 2	0.95D+0.95P+1.15K01+1.60Lm01	0
9	ULS 2:min04	ULS 2	0.95D+0.95P+1.15K02+1.60Lm01	0
10	ULS 3:max09	ULS 3	1.10D+1.05P+1.00K01+1.40Lm01	0
11	ULS 3:max10	ULS 3	1.10D+1.05P+1.00K02+1.40Lm01	0
12	ULS 3:min07	ULS 3	0.95D+0.95P+1.00K01+1.40Lm01	0
13	ULS 3:min08	ULS 3	0.95D+0.95P+1.00K02+1.40Lm01	0
14	ULS 4:max13	ULS 4	1.10D+1.05P+1.25K01	0
15	ULS 4:max14	ULS 4	1.10D+1.05P+1.25K02	0
16	ULS 4:min11	ULS 4	0.95D+0.95P+1.25K01	0
17	ULS 4:min12	ULS 4	0.95D+0.95P+1.25K02	0
18	ULS 9:max16	ULS 9	1.35D+1.05P	0
19	ULS 9:min15	ULS 9	1.35D+0.95P	0
20	FLS 117	FLS 1	1.00D+1.00P+1.00Lm02	0
21	SLS_1_18	SLS 1	1.00D+1.00P+0.80K01+0.90Lm01	0
22	SLS_1_19	SLS 1	1.00D+1.00P+0.80K02+0.90Lm01	0
23	SLS_2_20	SLS 2	0.90Lm02	0

Rebar Placement Window

You must open the *Rebar Placement* window to place cables within continuous system. This window has its own menus and is composed of an elevation view of continuous system. You can create cross-sections, display dimensions and rebars, and display forces and resistances diagrams as you will see further on.

- Activate the "Rebar Placement" mode on Activation toolbar and double click on continuous system.

VisualDesign will take a few seconds to open the window.

View Options

- Open the **View Options** dialog box. Check the *General* and *Dimensions* roots in the **Rebar Placement** tab. Press OK.

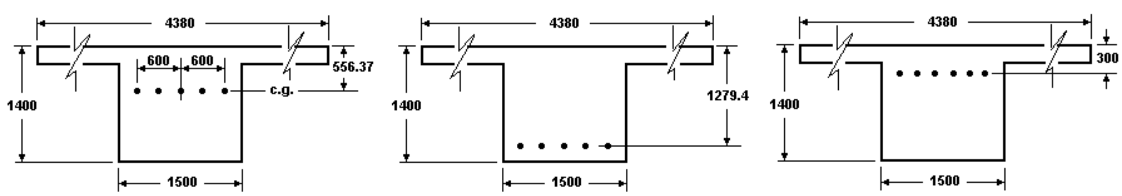
Cross-Sections

VisualDesign generates cross-sections at supports and mid-spans with the function **Automatic Generation of Cross-sections** (**Rebar Placement** menu).

You will notice that two cross-sections are overlapped at intermediate supports. To delete a cross-section, click on its outline to highlight it and press the [Delete] key.

Group of Cables

Position of Sheaths in the T-Beam



End of Layout | At Mid-Spans | At Central Support

- Go to **Rebar Placement** menu and select **Cable Groups and Layouts**. Insert a line and enter/select parameters as follows:

Cable Groups and Layouts Spreadsheet

Groups | Layouts

	Number	Post-tensioning Mechanism	Number in width	X beg. mm	X end mm	Number of strands/sheath	Strand
1	1	Sheath_01	6	-600.00	600.00	19	G270:3/5

Cable Groups and Layouts Spreadsheet

Groups | Layouts

	Layout	Jacking %	Delta left mm	Delta right mm	Stage	Factor Development Length
1	Internal with grout	80.00	10.00	10.00	Initial Post-tensioning	50.00

X beg and *X end* represent the transverse end positions of sheaths/strands according to continuous system local x-axis for post-tensioning and pre-tensioning.

Delta left and *Delta right* are measured from the cone penetration at the left and right end of continuous system (z-axis), after post-tensioning.

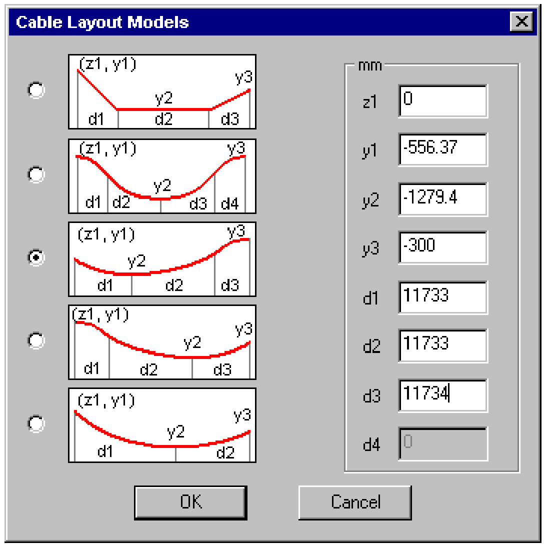
Cable Layout

The completed layout must be as follows:

Cables must be positioned from their centre of gravity to continuous system longitudinal axis. The latter is located at the top of the section. The centre of gravity of the section is located at –556.37mm from the top of the section, as we saw in the **Connection** tab.

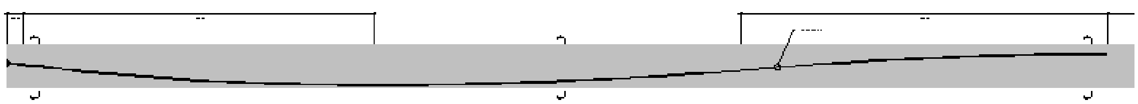
N.B. At mid-spans, dimension 1279.4mm was obtained considering a concrete cover of 75mm, M15 stirrups (diameter of 16.2mm), and sheath diameter, which is equal to 75mm.

- Go to the **Layouts** tab.
- Open the **Cable Layout Models** dialog box (Click a cell, right click, and select **Automatic Generation**).
- Activate the third model and enter coordinates. This model corresponds to the cable layout located in the first span.



- Press OK. You will go back to the **Layouts** tab.

- VisualDesign has created three segments for this layout and has calculated inflexion points. Give a number to each segment and select a *Mobile* cable ending at the beginning of the layout. Press OK.



The cable layout is now completed for the first span, and is displayed on screen:

- Define the cable layout for the second span. Open the **Cable Layout Models** dialog box and activate the forth model. Enter coordinates and, once back in the **Layouts** tab, select *Mobile* endings at each end of this layout.

The final layout coordinates are the following:

6	Number	Shape	Start	End	z1 mm	y1 mm	z2 mm	y2 mm	z3 mm	y3 mm
1	1	Parabolic	Mobile	Continuous	0.00	-556.37	7190.00	-1098.64	14380.00	-1279.40
2	2	Parabolic	Continuous	Continuous	14380.00	-1279.40	19790.00	-1152.15	25200.00	-770.41
3	3	Parabolic	Continuous	Continuous	25200.00	-770.41	30200.00	-417.60	35200.00	-300.00
4	4	Parabolic	Continuous	Continuous	35200.00	-300.00	40200.00	-417.60	45200.00	-770.41
5	5	Parabolic	Continuous	Continuous	45200.00	-770.41	50610.00	-1152.15	56020.00	-1279.40
6	6	Parabolic	Continuous	Mobile	56020.00	-1279.40	63210.00	-1098.64	70400.00	-556.37

- Close the *Rebar Placement* window.

You are ready to analyze the prestressed composite beam and design stirrups.

Design

- Click on the **Analysis and Design** icon on Tools toolbar or select **Analysis and Design** in **Analysis** menu. The **Design** dialog box will appear on your screen. Press the “Analyze” button. Close the dialog box when design is completed.

Detailed Results

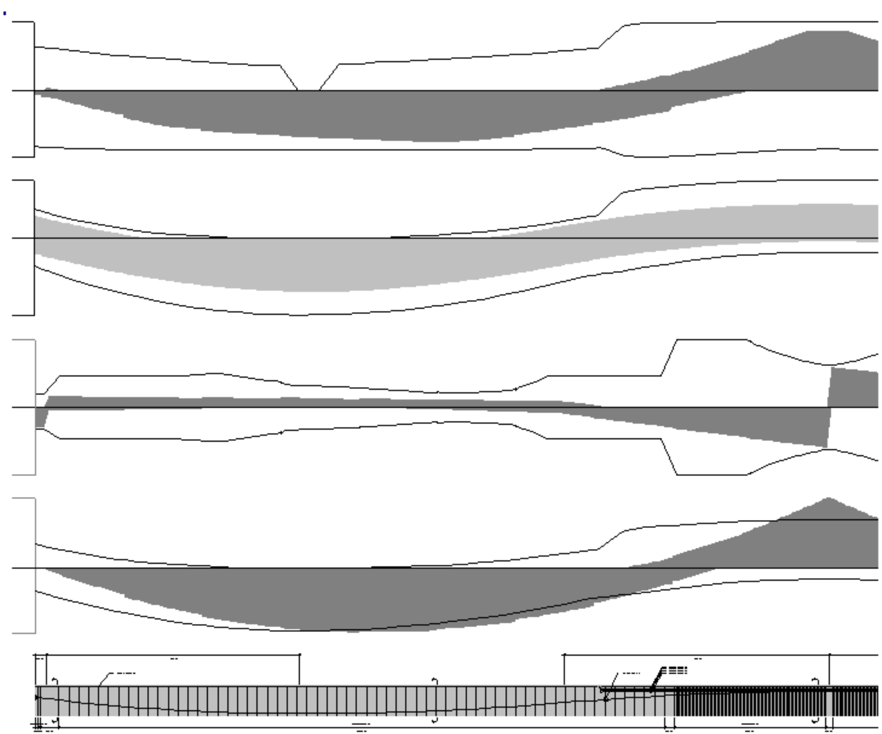
Rebar Placement window

- Activate the *Rebar Placement* mode and double-click on the continuous system to open the Rebar Placement window.

You will notice that VisualDesign has designed and placed stirrups in continuous beams, according to Code S6-00 and the *General* method.

Display Force and Resistance Diagrams

- Select the **View Options** dialog box and expand the *Beam Diagrams* root of **Rebar Placement** tab. Check the boxes corresponding to the force and resistance diagrams that you want to display, as shown below.

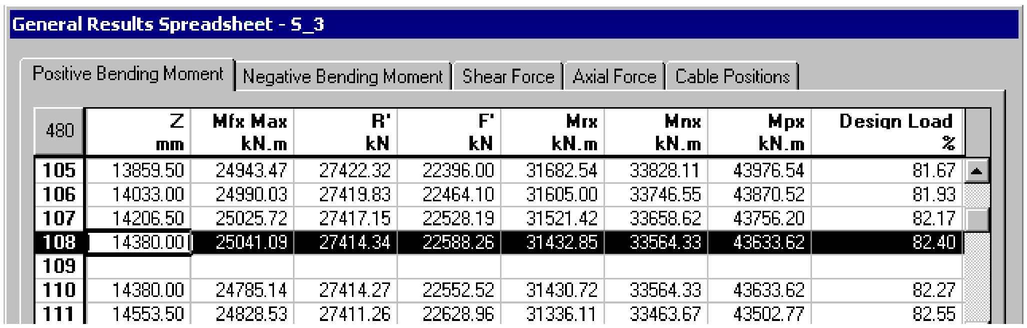


We can see that the cable layout is not appropriate because it does not take the shape of the bending moment diagram. The design in the zone of negative bending moment is not right, either.

We have to find the position of maximum bending moment. Then, a new cable layout will be defined with this position.

- Open the **General Results** spreadsheet (**Results** menu) and look in the **Positive Bending Moment** tab.

Maximum bending moment is located at 14380mm from origin of continuous system. This is the first coordinate that we are looking for.

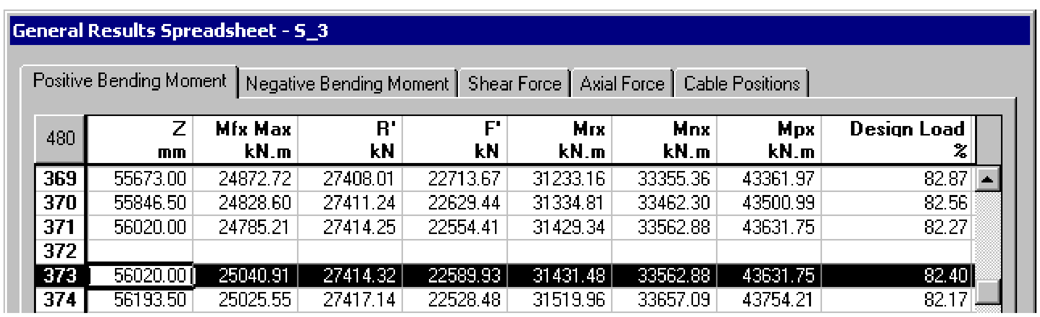


General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

480	Z mm	Mfx Max kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %
105	13859.50	24943.47	27422.32	22396.00	31682.54	33828.11	43976.54	81.67
106	14033.00	24990.03	27419.83	22464.10	31605.00	33746.55	43870.52	81.93
107	14206.50	25025.72	27417.15	22528.19	31521.42	33658.62	43756.20	82.17
108	14380.00	25041.09	27414.34	22588.26	31432.85	33564.33	43633.62	82.40
109								
110	14380.00	24785.14	27414.27	22552.52	31430.72	33564.33	43633.62	82.27
111	14553.50	24828.53	27411.26	22628.96	31336.11	33463.67	43502.77	82.55

The maximum bending moment in the second beam is located at 56020mm from origin of continuous system.



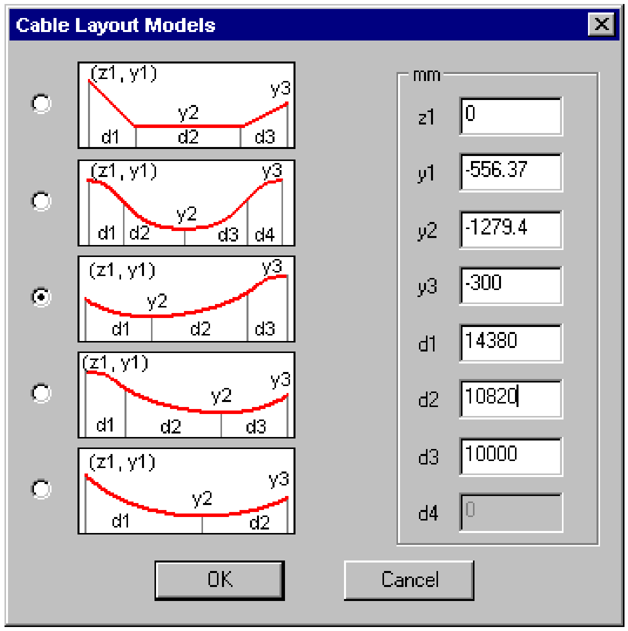
General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

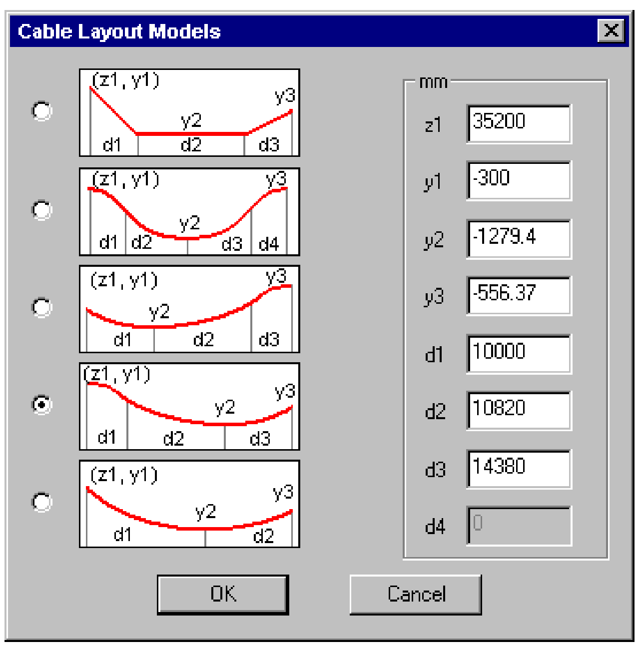
480	Z mm	Mfx Max kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %
369	55673.00	24872.72	27408.01	22713.67	31233.16	33355.36	43361.97	82.87
370	55846.50	24828.60	27411.24	22629.44	31334.81	33462.30	43500.99	82.56
371	56020.00	24785.21	27414.25	22554.41	31429.34	33562.88	43631.75	82.27
372								
373	56020.00	25040.91	27414.32	22589.93	31431.48	33562.88	43631.75	82.40
374	56193.50	25025.55	27417.14	22528.48	31519.96	33657.09	43754.21	82.17

Modification of Cable Layouts

- Uncheck the *Beam Diagrams* root in the **Rebar Placement** tab of **View Options** dialog box to avoid recalculations of diagrams while you edit.
- Open the **Cable Groups and Layouts** spreadsheet. Select the **Layouts** tab and delete all the lines. Open the **Cable Layout Models** dialog box and activate the third model. Enter new coordinates, as follows:



- Define the cable layout for the second beam, placing the lowest point at a distance of 56020mm from origin. Press OK.



- Select appropriate cable endings in the **Layouts** tab and close the spreadsheet.
- Close the *Rebar Placement* window.
- Launch the design again.
- Double click on the continuous system to open the *Rebar Placement* window.

- Select the **View Options** dialog box and display forces and resistances diagrams again.

Partial Results

Design Loads

- Stay in VisualDesign main window and activate the *Design Results* mode on Activation toolbar.

- Go to **Results / Structure Design / Concrete** and look at calculated design loads for this continuous system.

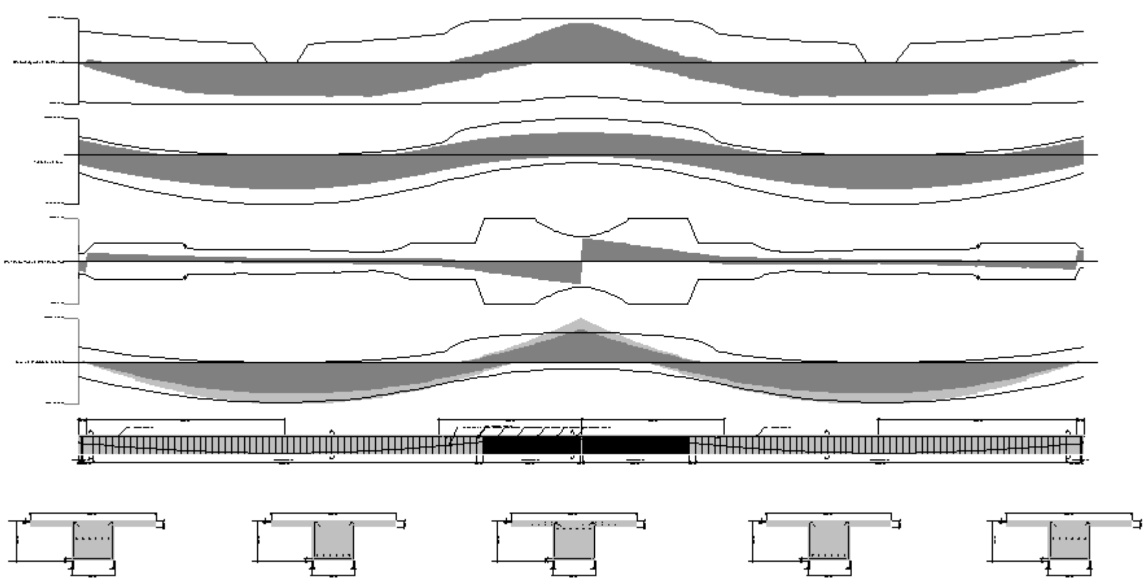
Design Results for Continuous Systems

Beam / Joist | Column | Shear Wall | Slab

1	Number	Design load Positive moment %	Design load Negative moment %	Design load Shear force %	Cracking Positive moment %	Cracking Negative moment %
1	S_3	83.11	98.92	95.49	0.00	0.00

Graphical Results

- Activate the *Rebar Placement* mode on Activation toolbar and double click on continuous system.



The design is adequate.

We are going to consult detailed results (graphical and numerical) for construction stage load combinations and other serviceability load combinations. These results are available through the **Results** menu of *Rebar Placement* window.

Graphs

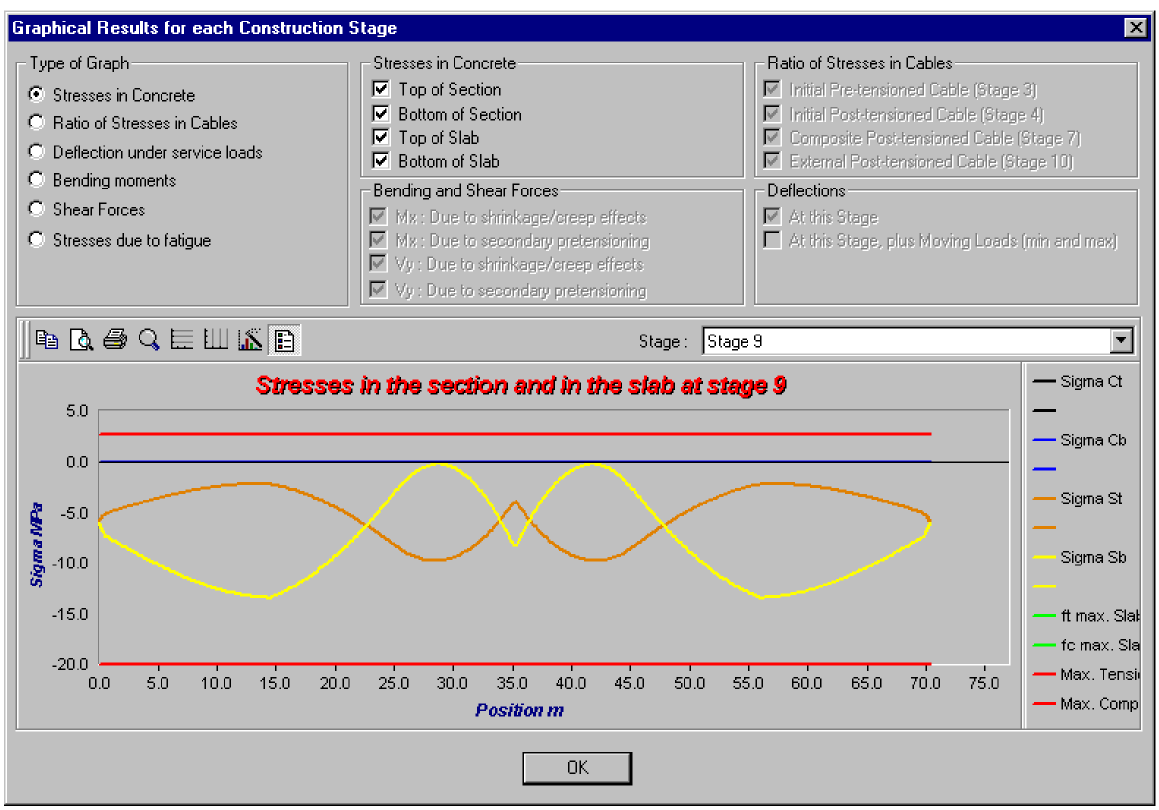
Graphic results are available for Serviceability load combinations.

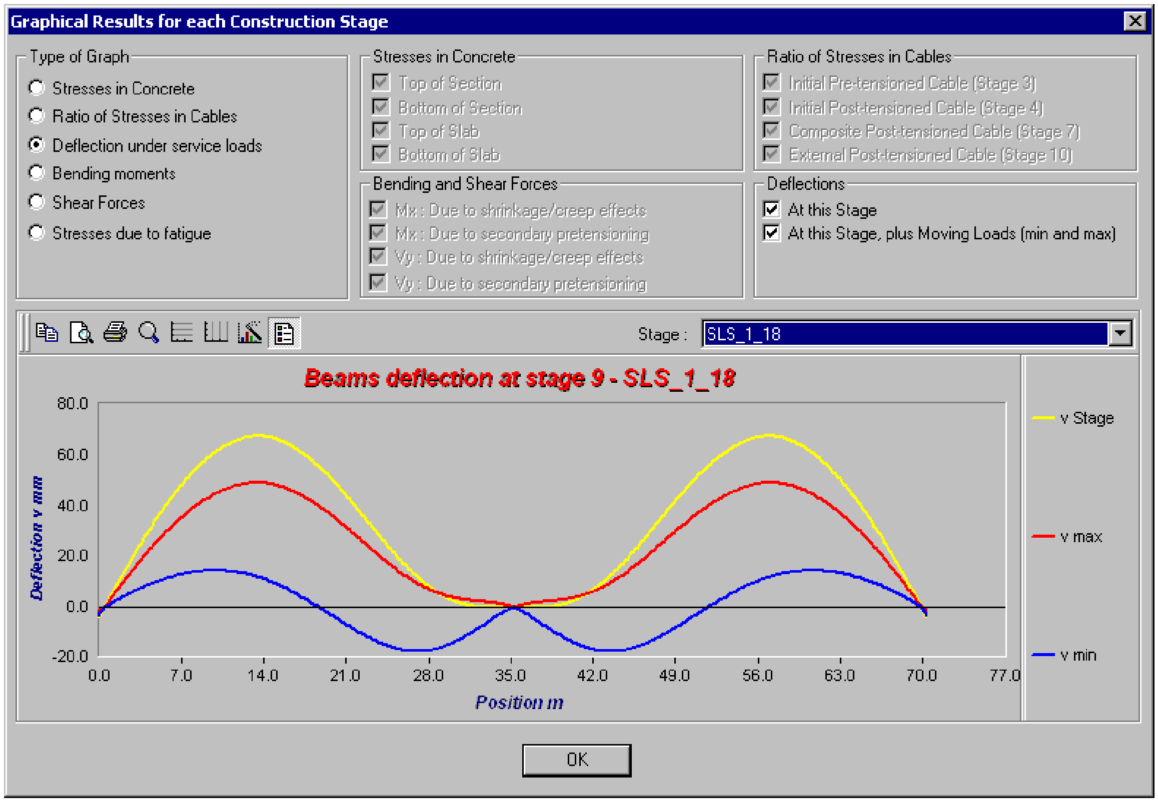
- Go to **Results** / **Graphs** to open the **Graphical Results** dialog box or click the icon  on View toolbar.
- In the **Graphs** dialog box, select a graph and select a load combination.

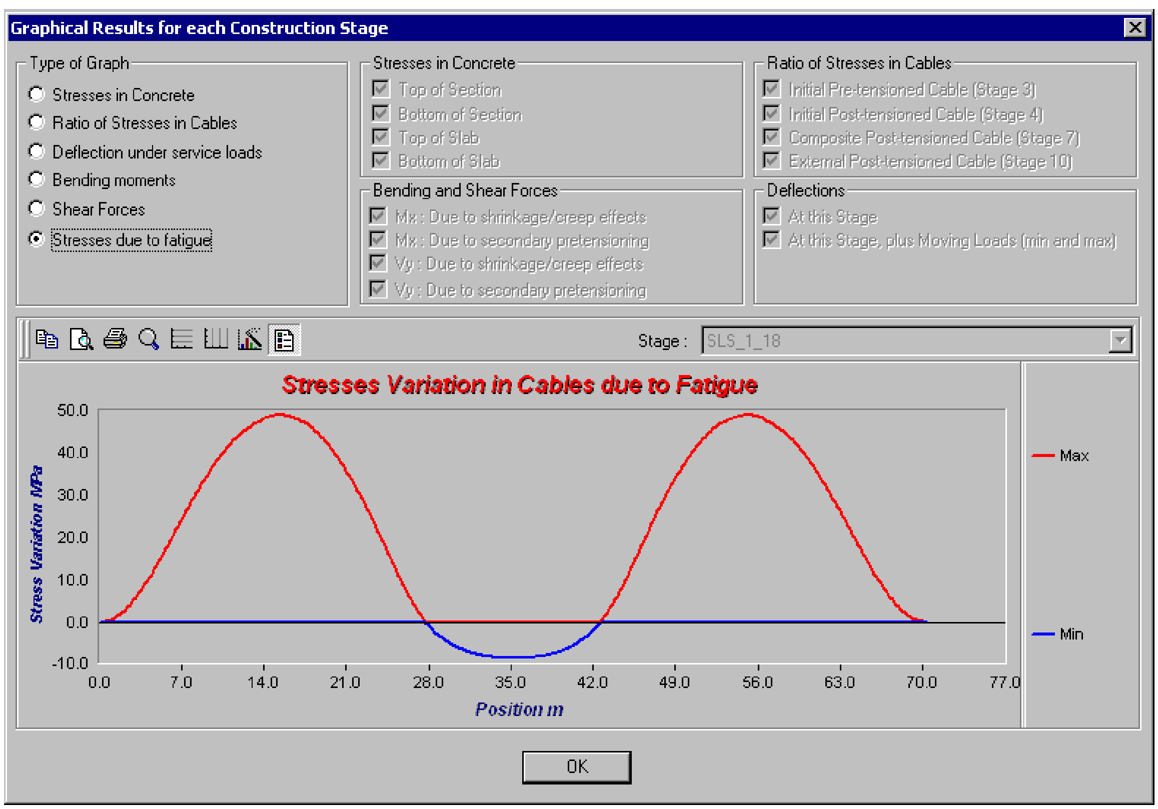
To move the legend out of the box, double click on it.

Place the cursor on any point on the curve and coordinates will be displayed next to your cursor.

Use the **Graphs** toolbar that is supplied with this dialog box to change the look of the graph and x-axis and y-axis subdivisions. Use the **Print Preview** and **Print** functions to print the graph.




Graphical Results for each Construction Stage
Type of Graph
Stresses in Concrete
Ratio of Stresses in Cables
Deflection under service loads
Bending moments
Shear Forces
Stresses due to fatigue
Stresses in Concrete
Top of Section
Bottom of Section
Top of Slab
Bottom of Slab
Ratio of Stresses in Cables
Initial Pre-tensioned Cable (Stage 3)
Initial Post-tensioned Cable (Stage 4)
Composite Post-tensioned Cable (Stage 7)
External Post-tensioned Cable (Stage 10)
Bending and Shear Forces
Mx : Due to shrinkage/creep effects
Mx : Due to secondary pretensioning
Vy : Due to shrinkage/creep effects
Vy : Due to secondary pretensioning
Deflections
At this Stage
At this Stage, plus Moving Loads (min and max)
Stage : SLS_1_18
Beams deflection at stage 9 - SLS_1_18
Deflection v mm
Position m
v Stage
v max
v min
OK


Graphical Results for each Construction Stage
Type of Graph
Stresses in Concrete
Ratio of Stresses in Cables
Deflection under service loads
Bending moments
Shear Forces
Stresses due to fatigue
Stresses in Concrete
Top of Section
Bottom of Section
Top of Slab
Bottom of Slab
Ratio of Stresses in Cables
Initial Pre-tensioned Cable (Stage 3)
Initial Post-tensioned Cable (Stage 4)
Composite Post-tensioned Cable (Stage 7)
External Post-tensioned Cable (Stage 10)
Bending and Shear Forces
Mx : Due to shrinkage/creep effects
Mx : Due to secondary pretensioning
Vy : Due to shrinkage/creep effects
Vy : Due to secondary pretensioning
Deflections
At this Stage
At this Stage, plus Moving Loads (min and max)
Stage : SLS_1_18
Stresses Variation in Cables due to Fatigue
Stress Variation MPa
Position m
Max
Min
OK

Numerical Results

Prestress Loss in Cables

- Go to **Results** menu and select **Prestress Loss in Cables**. This spreadsheet includes prestress losses <u>AND</u> gains for each construction stage. Total loss, Δp, is equal to -200.34 MPa.

Loss of Prestress in Cables Spreadsheet

1	Number	Stage	ES4 MPa	Δp4 MPa	ES8 MPa	REL8 MPa	CR8 MPa
1	1	Initial Post-tensioning	-30.26	-30.26	5.98	-5.41	-37.90

Loss of Prestress in Cables Spreadsheet

1	SH8 MPa	Δp8 MPa	REL9 MPa	CR9 MPa	SH9 MPa	Δp9 MPa	Δp MPa
1	-0.44	-37.77	-24.74	-75.91	-31.65	-132.31	-200.34

Stresses in Concrete and Prestressing Cables for each Construction stage

- Go to **Results** / **Stresses** / **Stage *X***. This spreadsheet supplies minimum and maximum stresses in concrete, at the top (σss) and bottom (σsi) of the T-Beam. Stress ratios are also given for post-tensioning cables. Mask useless columns.

Stresses under Service Loads Spreadsheet : Stage 4

483	Number	Position m	Section Top σss min MPa	Section Top σss max MPa	Section Bottom σsi min MPa	Section Bottom σsi max MPa	Cable Initial Post-tension σ/fpu Min.	Cable Initial Post-tension σ/fpu Max.	Deflection Stage mm	Max Deflection Stage+Truck mm	Min Deflection Stage+Truck mm
380	26	0.87	-0.53	-0.53	-18.17	-18.17	0.71	0.71	32.98	32.98	32.98
381	26	1.04	-0.52	-0.52	-18.17	-18.17	0.71	0.71	32.96	32.96	32.96
382	26	1.21	-0.52	-0.52	-18.15	-18.15	0.71	0.71	32.92	32.92	32.92
383	26	1.39	-0.53	-0.53	-18.11	-18.11	0.71	0.71	32.87	32.87	32.87
384	26	1.56	-0.54	-0.54	-18.07	-18.07	0.71	0.71	32.81	32.81	32.81
385	26	1.74	-0.55	-0.55	-18.04	-18.04	0.71	0.71	32.73	32.73	32.73
386	26	1.91	-0.55	-0.55	-18.01	-18.01	0.70	0.70	32.64	32.64	32.64
387	26	2.08	-0.55	-0.55	-17.99	-17.99	0.70	0.70	32.53	32.53	32.53
388	26	2.26	-0.56	-0.56	-17.96	-17.96	0.70	0.70	32.41	32.41	32.41

Intermediate Results

- Go to **Results** menu and select **Intermediate Results**. This spreadsheet includes properties and results on shrinkage and creep effects, for all construction stages, according to cumulated days.

Prestressed Concrete Intermediate Results Spreadsheet

66	Stage day	Member	Shape	f'ci section MPa	Eci section MPa	Ix section 10e6mm4	Area section mm²	Compression Limit Section MPa	Tension Limit Section MPa	Creep Section	Shrinkage Section
41	31	27	T1x1400	50.70	30124.40	521257.34	2791199.92	30.42	1.42	0.158437	-0.000002
42	31	28	T1x1400	50.70	30124.40	521257.34	2791199.92	30.42	1.42	0.158437	-0.000002
43	31	29	T1x1400	50.70	30124.40	521257.34	2791199.92	30.42	1.42	0.158437	-0.000002
44	31	4	T1x1400	50.70	30124.40	521257.34	2791199.92	30.42	1.42	0.158437	-0.000002
45	5000	10	T1x1400	58.28	31766.79	521257.34	2791199.92	20.00	2.83	1.122911	-0.000168
46	5000	2	T1x1400	58.28	31766.79	521257.34	2791199.92	20.00	2.83	1.122911	-0.000168
47	5000	22	T1x1400	58.28	31766.79	521257.34	2791199.92	20.00	2.83	1.122911	-0.000168
48	5000	23	T1x1400	58.28	31766.79	521257.34	2791199.92	20.00	2.83	1.122911	-0.000168

General Results

- Go to **Results** menu and select the **General Results** spreadsheet. Lines that are marked in yellow mean that one or more parameters in these lines do not follow the Code requirements.

The Positive Bending Moment tab

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	Member Number	Z mm	Mfx Max kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %	fcr MPa	Mcr kN.m	d mm	dv mm
111	25	14380.00	25545.48	27589.72	22326.87	32303.40	34482.74	44827.56	80.92	2.83	23272.03	1279.63	1170.85
112	25	14553.50	25566.93	27589.42	22339.87	32299.46	34478.52	44822.08	80.97	2.83	23241.37	1279.49	1170.72
113	25	14727.00	25588.38	27588.91	22357.25	32288.11	34466.49	44806.43	81.04	2.83	23204.59	1279.10	1170.33
114	25	14900.50	25601.95	27588.19	22373.03	32269.36	34446.66	44780.65	81.10	2.83	23163.01	1278.45	1169.68
115	25	15074.00	25595.59	27587.27	22380.65	32243.21	34419.04	44744.75	81.13	2.83	23116.64	1277.54	1168.77
116	25	15247.50	25591.35	27586.14	22391.56	32209.65	34383.62	44698.70	81.17	2.83	23065.51	1276.37	1167.60
117	25	15421.00	25581.99	27584.80	22403.00	32168.70	34340.40	44642.52	81.22	2.83	23009.62	1274.94	1166.17

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	bw mm	As mm²	ρ %	φp	φp.fps.Aps kN	εx	θ °	β	c/d	c/d Max	Stresses Var. Rebars MPa	Ieff	ICr
111	1500.00	400.00	0.02	0.30	27431.81	0.000645	31.48	0.20	0.20	0.50	239.46	1.00	0.95
112	1500.00	400.00	0.02	0.30	27431.35	0.000660	31.69	0.20	0.20	0.50	240.33	1.00	0.94
113	1500.00	400.00	0.02	0.30	27430.65	0.000676	31.92	0.20	0.20	0.50	241.44	1.00	0.93
114	1500.00	400.00	0.02	0.30	27429.75	0.000692	32.15	0.20	0.20	0.50	242.58	1.00	0.91
115	1500.00	400.00	0.02	0.30	27428.63	0.000704	32.33	0.20	0.20	0.50	243.46	1.00	0.90
116	1500.00	400.00	0.02	0.30	27427.29	0.000718	32.53	0.20	0.20	0.50	244.62	1.00	0.89
117	1500.00	400.00	0.02	0.30	27425.74	0.000732	32.74	0.20	0.20	0.50	245.88	1.00	0.87

The Negative Bending Moment tab

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	Member Number	Z mm	Mfx Min kN.m	R' kN	F' kN	Mrx kN.m	Mnx kN.m	Mpx kN.m	Design Load %	fcr MPa	Mcr kN.m	d mm	dv mm
239	5	34762.99	-24701.38	26158.77	25859.10	21910.75	24864.62	32324.01	98.85	2.83	17164.27	1138.19	837.61
240	5	34931.48	-25380.60	26150.62	25859.10	21917.95	24872.29	32333.97	98.89	2.83	17152.84	1138.63	838.14
241	5	35099.98	-26065.92	26140.26	25859.10	21918.63	24872.69	32334.50	98.92	2.83	17137.41	1138.87	838.50
242													
243	3	35300.00	-26065.97	26140.25	25859.10	21918.63	24872.69	32334.49	98.92	2.83	17137.40	1138.87	838.50
244	3	35468.50	-25380.77	26150.61	25859.10	21917.95	24872.29	32333.97	98.89	2.83	17152.83	1138.63	838.14
245	3	35637.00	-24701.54	26158.77	25859.10	21910.75	24864.62	32324.01	98.85	2.83	17164.27	1138.19	837.61
246	3	35805.50	-24028.57	26164.58	25859.10	21897.00	24849.66	32304.55	98.83	2.83	17171.47	1137.55	836.90

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	bw mm	As mm²	ρ %	ϖp	ϕp.fps.Aps kN	εx	θ °	β	c/d	c/d Max	leff	ICr
239	1500.00	16800.00	0.98	0.32	20110.78	0.002000	39.27	0.12	0.63	0.50	1.00	0.35
240	1500.00	16800.00	0.98	0.32	20102.63	0.002000	39.12	0.12	0.63	0.50	1.00	0.35
241	1500.00	16800.00	0.98	0.32	20092.27	0.002000	38.99	0.12	0.63	0.50	1.00	0.34
242												
243	1500.00	16800.00	0.98	0.32	20092.26	0.002000	39.00	0.12	0.63	0.50	1.00	0.34
244	1500.00	16800.00	0.98	0.32	20102.62	0.002000	39.12	0.12	0.63	0.50	1.00	0.35
245	1500.00	16800.00	0.98	0.32	20110.78	0.002000	39.27	0.12	0.63	0.50	1.00	0.35
246	1500.00	16800.00	0.98	0.33	20116.58	0.002000	39.42	0.12	0.63	0.50	1.00	0.36

The Shear Force tab

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	Member Number	Z mm	Vfy Max kN	Vfy Min kN	Design Load %	εx	θ °	β	Vcy kN	Vsy kN	Vry kN	dy mm	dvy mm	ϕpVp kN
193	15	27566.00	2128.88	410.66	57.04	-0.003741	27.00	0.41	2760.85	971.17	3732.03	859.88	1008.00	1130.96
194	15	27739.50	2193.27	464.67	58.77	-0.003611	27.00	0.41	2760.85	971.17	3732.03	871.15	1008.00	1103.82
195	15	27913.00	2257.53	518.56	60.49	-0.003480	27.00	0.41	2760.85	971.17	3732.03	882.18	1008.00	1076.73
196	15	28086.50	2317.75	570.40	62.10	-0.003348	27.00	0.41	2760.85	971.17	3732.03	892.95	1008.00	1049.72
197	15	28260.00	2381.81	624.09	24.27	-0.003215	27.00	0.41	2760.85	7051.06	9811.91	903.36	1008.00	1023.08
198														
199	30	28260.00	2381.81	624.09	24.10	-0.003214	27.00	0.41	2760.85	7121.92	9882.78	903.60	1008.00	1022.46
200	30	28433.50	2445.77	676.98	24.75	-0.003079	27.00	0.41	2760.85	7121.92	9882.78	913.75	1008.00	995.89
201	30	28607.00	2509.60	729.56	25.39	-0.002943	27.00	0.41	2760.85	7121.92	9882.78	923.78	1008.00	969.07

The Cable Positions tab

General Results Spreadsheet - S_3

Positive Bending Moment | Negative Bending Moment | Shear Force | Axial Force | Cable Positions

483	Member Number	Z mm	1 y mm	1 Slope °	1 Area mm²
63	22	7093.00	-1093.73	-2.92	15959.97
64	22	7266.50	-1102.47	-2.85	15959.97
65	22	7440.00	-1110.90	-2.78	15959.97
66					
67	23	7440.00	-1111.09	-2.78	15959.97
68	23	7613.50	-1119.31	-2.71	15959.97
69	23	7787.00	-1127.41	-2.64	15959.97
70	23	7960.50	-1135.31	-2.57	15959.97

On-Line Help:

Press the **F1** key to open VisualDesign On-line Help and obtain the description of columns included in the displayed spreadsheet.

Summary of Procedure

1. Project Configuration

- Go to **File / Project Configuration** and select the **Analysis** tab. Specify a greater number of iterations, such as 50, because of construction stages. Specify the number of subdivisions for concrete members.

- Go to the **Concrete Design** tab. Select the *General Method* for concrete design.

- Go to the **Prestressing** tab. Check the "Prestressed Concrete Project" box and activate appropriate construction stages.

2. Concrete Specification and Selection of Reinforcement

- Select the **Concrete Specifications** spreadsheet in the **Structure** menu. Consult the design specification and modify default parameters, if needed. Select transverse reinforcement that will be used for the design of stirrups.

3. Slab – Composite beam

- For composite beams, go to **Structure** menu and create a slab in the **Slabs** spreadsheet.

4. Member Characteristics Dialog Box

- Select a shape and material and activate design criteria in the **Member** tab of **Member Characteristics** dialog box..

- For a composite section (AASHTO, NEBT or T shape), select *Composite Beam* in the "Composition" field. Then, in the **Composite Beam** tab, select the slab that you created before. Choose the beam end conditions on strong axis during construction stages 1 to 5. ***Never check option "Add dead load of slab".***

- Model members' rigid extensions in the **Connection** tab and align members at the top. Include the mass of rigid extensions for all members.

- In the **Concrete Design** tab, select an option for the calculation of Vy: *at the face of support* or at *d or dv of support.*

5. Continuous Systems Spreadsheet

- Go to Structure / **Continuous Systems.** Select a concrete specification. Enter concrete covers and specify the cracking parameters.

6. Load Cases and Load Combinations

- Go to **Loads**/**Load Cases** / **Definition**. Enter load case title and types that will be apply during construction stages.

- Go to **Loads / Load Combinations / Definition.** Define construction stage load combinations. Select a *Construction stage* status for each one. Click in the "Stage" cell and specify the stage number.

- Select the **Load Factors** tab. For each stage load combination, select appropriate load cases (virtual loads also) in the *Load Case* cell. Enter load factors.

- Use the **Load Combination Generation Wizard** to create other load combinations as per selected building code or standard.

7. Definition of Strands and Post-Tensioning Mechanisms

- Go to **Common** menu. Make sure that cable steel grades and strands are listed in the spreadsheets.

- Open the **Post-tension Mechanisms** spreadsheet and define the post-tension mechanism that will be used in your project.

8. Group of Cables and Cable Layouts

- Activate the “Rebar Placement” mode and double-click on any continuous system to open the *Rebar Placement* window.

- Select **Cable Groups and Layouts** in the **Rebar Placement** menu. Insert appropriate number of lines to define each cable group, and complete the required information. Specify the tension at jacking and pre- or post-tensioning stage where it will be applied.

- Select the **Layouts** tab and enter data yourself or use the **Cable Layout Models** dialog box, which is a tool accessible through the contextual menu (mouse right click, command **Automatic Generation**).

- In the Layouts tab, give a name to each segment of cable layout. Select the appropriate cable endings at each end of segments (Mobile, Fixed or Continuous). If there is some cable degradation, enter a percentage of deterioration in the *Deterioration* column.

- Close the *Rebar Placement* window.

9. Design

- Launch the design. The Moving Load Analysis will be launched automatically. VisualDesign™ will verify or design the prestressed concrete beam according to the prestressing cables that were placed in the continuous system.

10. Consult Design Results

- Activate the "Design Results" mode and go to **Results / Structure Design / Concrete** and consult calculated design loads for continuous systems.

- Activate the "Rebar Placement" mode and double-click on a continuous system to open the *Rebar Placement* window.

- Open the **View Options** dialog box and check the *Dimensions* and *General* roots. Expand the *Beam Diagrams* root in the **Rebar Placement** tab and choose the diagrams that you want to look at. (Notice: Some results are included in the **General Results** spreadsheet only if the corresponding diagram is displayed on screen, such as *Variation of stresses in rebars* and *Cracking*)..

- Go to **Results** menu. Look at results in the form of graphs by selecting **Graphs**. Numerical results are available in the following spreadsheets: **Prestress Loss in cables**, **Stresses** in cables and in concrete, **Intermediate Results**, and **General Results**.

E X A M P L E S 1 4 , 1 5 & 1 6

Foundation Design

Footings

Soil-Structure Interaction

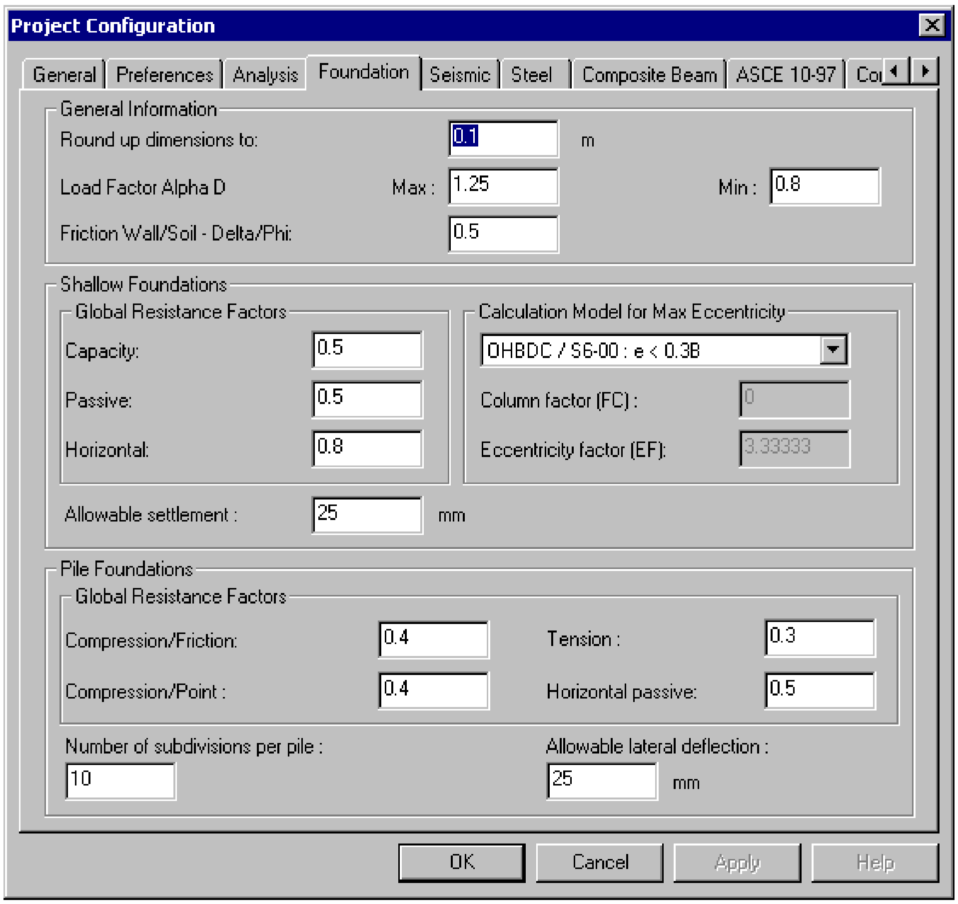
Piles

General – Foundation Design

Project Configuration – *Foundation* tab

The first step is to configure the foundation parameters. Go to **File / Project Configuration** and select the **Foundation** tab.

Among other parameters, users must specify the global resistance factor for shallow and deep foundations. Once that analysis is completed, this factor must be multiplied to the ultimate bearing capacity of foundation.



Soils

Make sure that types of soils that you will be using in your project are included in VisualDesign. Three spreadsheets are available, namely: **Cohesive Soils**, **Granular Soils**, and **Rocks**. They are located in **Common**/**Soils** menu. To add a soil in the database, select the appropriate spreadsheet, insert a line and complete the required parameters.

Important Parameters

You must, at least, define the following parameters:

	Calculation of Bearing Capacity	Calculation of Settlements
COHESIVE SOILS	Name of the soil Undrained shear resistance (not necessary if qult is known) Saturated and humid unit weights	Young's modulus Poisson's ratio
GRANULAR SOILS	Name of the soil Effective angle of internal friction Saturated and Humid Unit Weights	Young's modulus Poisson's ratio

Stratigraphical Profile

Define the stratigraphical profile over which will lay the foundation before selecting any foundation model. A geotechnical study may show that more than one profile is required. You must indicate, for each stratigraphical profile, the elevation of natural ground and water table. Enter data in the **Stratigraphical Profiles** spreadsheet (**Structure** menu).

Soil Layers

Usually, a stratigraphical profile is composed of many soil layers. The **Layers** tab allows you to define soil layers for each stratigraphical profile.

Specifications

A standard or building code must be selected in appropriate specifications spreadsheets (deep or shallow) and the type of analysis (design or check) must be specified. Then, a specification is assigned to each foundation model, deep or shallow, through **Foundation Models**.

Shallow Foundations

Enter the footing maximum dimensions if you plan to design the footings. For a check of foundations, enter current dimensions.

Definition of Foundation Models

The selection of a foundation model must be done according to theoretical and empirical calculation models. To learn more about these, refer to On-Line Help *Chapter 7 Foundation Design*.

Through Spreadsheets and Dialog Boxes

Data can be entered in the **Shallow Foundations** spreadsheet and the **Deep Foundations** spreadsheet, which are located in **Structure** / **Foundation Models**. Define as many models as you think there will be in the project.

We recommend that you enter data in the dialog box because it contains more information and it is much easier this way. To open the dialog box, click in any cell (**Shallow Foundations** or **Deep Foundations** spreadsheets), right click, and select *Details* in the contextual menu.

In the dialog box, data are divided into several tabs: *Foundation Model*, *Footing, Column*, *Design*, *Piles,* and *Piles Layout*.

Use the Foundation Modelling Wizard

Create foundation models in a quick way by using the **Wizard**. Users have to specify lesser parameters when this **Wizard** is used. VisualDesign creates a stratigraphical profile and a specification for the user. When the foundation model is completed, data are transferred in appropriate spreadsheets (**Shallow** or **Deep Foundation** spreadsheet) and can be modified afterwards. There is no limit to the number of models created with the **Wizard**.

Assigning Foundation Models to supports

Foundation models must be assigned to one support node or more. Usually, the same model is assigned to supports that carry approximately the same load to the foundation. A foundation model is assigned to a support through the **Support** tab of **Node Characteristics** dialog box.

Soil/Structure Interaction

Soil/structure interaction can be considered so that internal forces created by footing settlements are redistributed into the structure. The program performs iterations until it reaches convergence between calculated settlements and calculated forces to attain force/settlement compatibility.

VisualDesign calculates the footing settlements using the secant modulus of soil. However, to consider this modulus, the option "Secant modulus, K" must be selected as degrees of freedom, in the **Support** tab, for each foundation support.

You must create an *Interaction* type of load case and generate serviceability load combinations. You will find an example of such analysis further on.

Static Analysis

Shallow Foundations

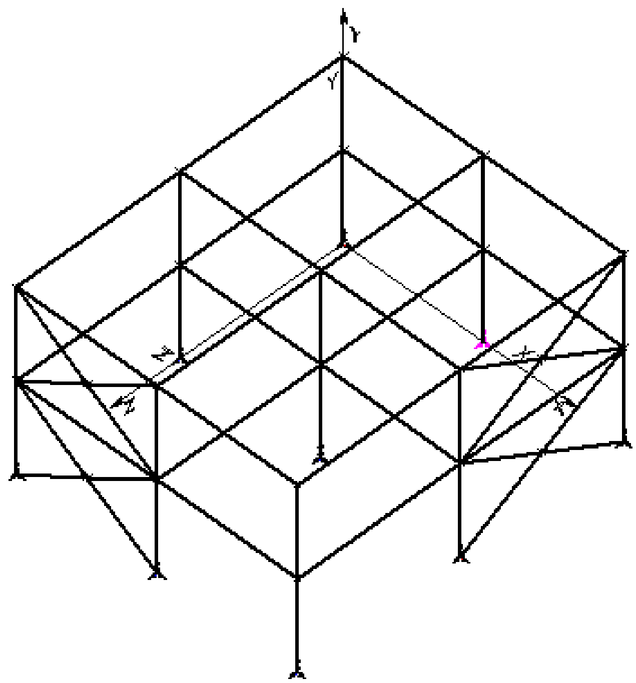
In Design mode, the footings are optimized according to the bearing capacity (ultimate limit states) or settlement (serviceability limit states). Eccentricities caused by moments or eccentric loads are taken into account. Eccentricities may be limited as specified. Reinforcement is supplied in a results spreadsheet and can be edited.

Deep Foundations

Deep foundations (piles) cannot be designed. They are verified using the selected steel shape and maximum length of piles, which is specified in the **Deep Foundation Specifications** spreadsheet.

Design of Shallow Foundations

Shallow foundations will be defined and designed for the following steel building:



Project Configuration

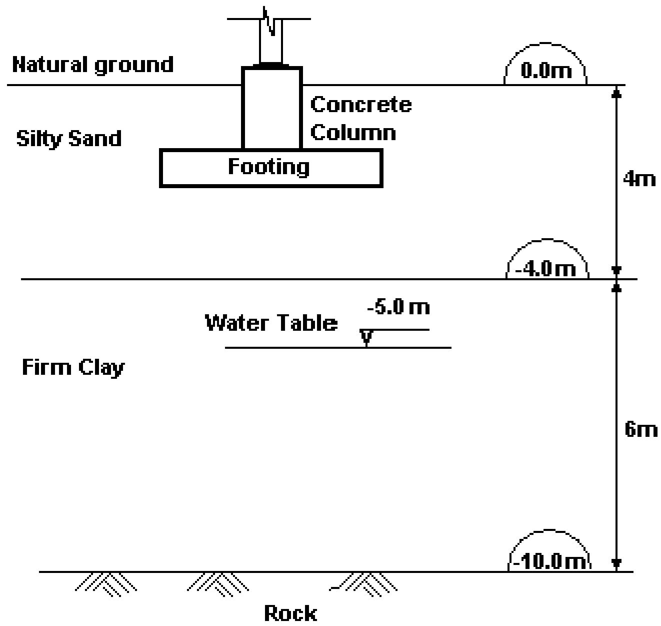
We keep the default values included in the **Foundation** tab of **Project Configuration** dialog box.

Soil Parameters

Geotechnical Study

A geotechnical study is recommended to obtain the stratigraphical profile below foundations. Results can vary much from a type of soil to another.

In our case, the study supplied the following profile:



Soils spreadsheets

Silty sand and firm clay are composing the stratigraphical profile. They are cohesive soils. Select **Common** /**Soils** /**Cohesive**. We added these soils in the database (by inserting lines at the end of the spreadsheet) and entered properties supplied by the geotechnical study.

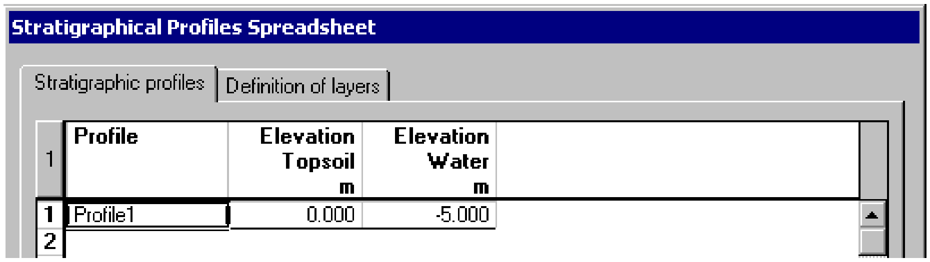
Cohesive Soils Spreadsheet

8	Name	Undrained Shear Resistance, Cu kPa	ϕ °	q ultimate kPa	γ Humid kN/m³	γ Saturated kN/m³	E MPa	µ	qc kPa	Ip	N1-60
1	Clay[1]Soft	40.00	0.00	250.00	16.00	17.00	20.00	0.40	0.00	1.00	3
2	Clay[2]Medium	75.00	0.00	500.00	16.00	17.00	35.00	0.50	0.00	1.00	10
3	Clay[3]Stiff	100.00	0.00	750.00	16.00	17.00	75.00	0.50	0.00	1.00	20
4	Clay[4]VeryStiff	200.00	0.00	1000.00	16.00	17.00	150.00	0.50	0.00	1.00	35
5	Clay[5]Hard	300.00	0.00	1500.00	16.00	17.00	100.00	0.50	0.00	1.00	35
6	Clay[0]VerySoft	10.00	0.00	50.00	16.00	17.00	2.00	0.40	0.00	1.00	3
7	Firm Clay	200.00	0.00	500.00	15.70	15.70	75.00	0.30	0.00	0.00	15
8	Silty Sand	7.50	36.00	287.28	18.50	18.50	75.00	0.45	0.00	0.00	0

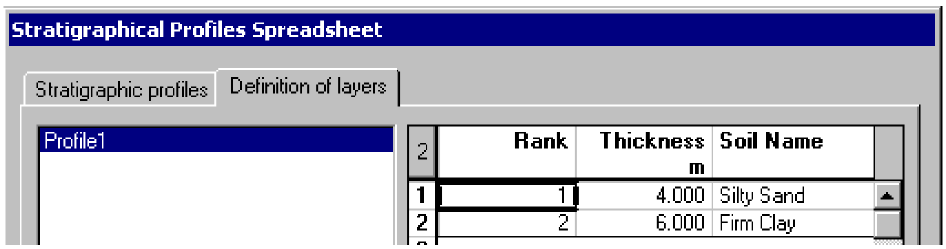
Click OK to save data and exit the spreadsheet.

Stratigraphical Profile

Select **Structure** / **Stratigraphical Profiles**. Insert a line, give a name to the profile, and enter elevation of natural ground and water table.



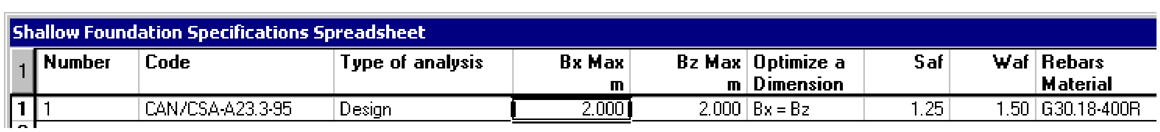
- Select the **Layer Definition** tab and insert two lines in the spreadsheet. For each layer, enter its rank (rank 1 is the first layer below natural ground) and thickness. Double click in the *Soil Name* cell and select type of soils that is composing the layer. Click OK.



Specifications for Shallow Foundations

A design code or standard must be selected in the specification spreadsheet and the type of analysis, design or check, must be specified.

- Go to **Structure / Specifications / Shallow Foundations**. Insert a line and give a name to this specification. Select the option *Design* as type of analysis, and enter the footing maximum dimensions, which is 2m x 2m. Select code A23.3 and rebars material.

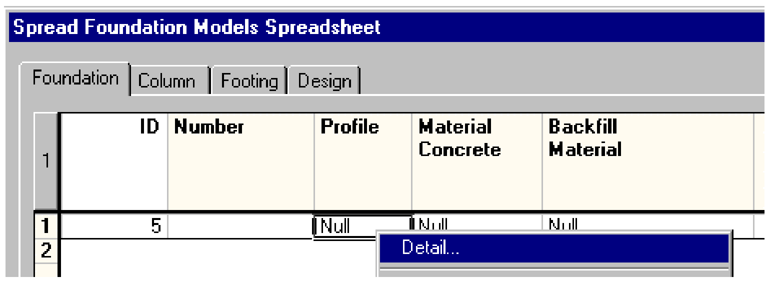


This specification will be selected while defining foundation models.

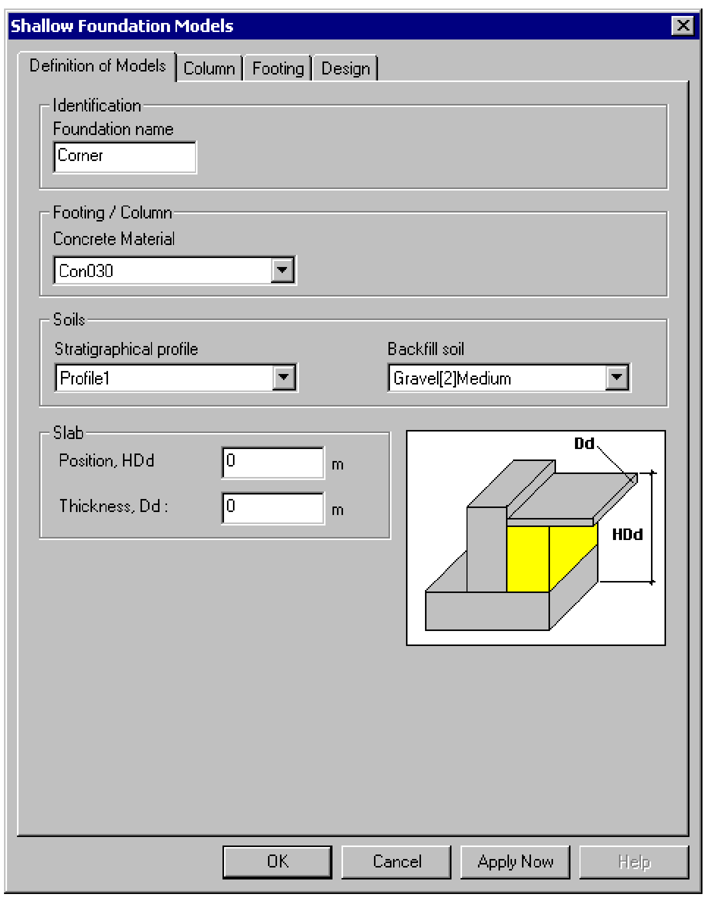
Shallow Foundation Models

Three types of footings will be defined: Corner footings (4), intermediate footings (4), and the centre one (1).

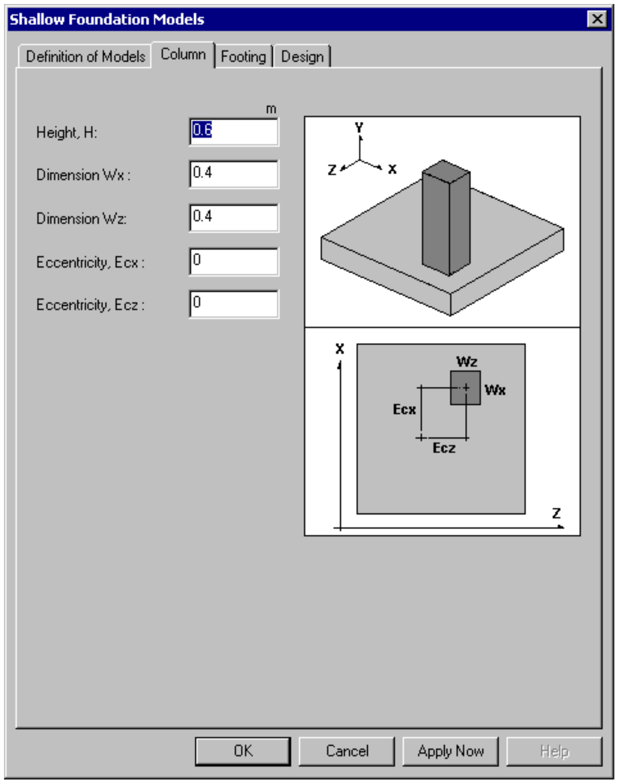
- Select **Structure** / **Foundation Models**/ **Shallow**. Insert a line. To open the dialog box, click in any cell, right click, and select the command *Detail* in contextual menu.



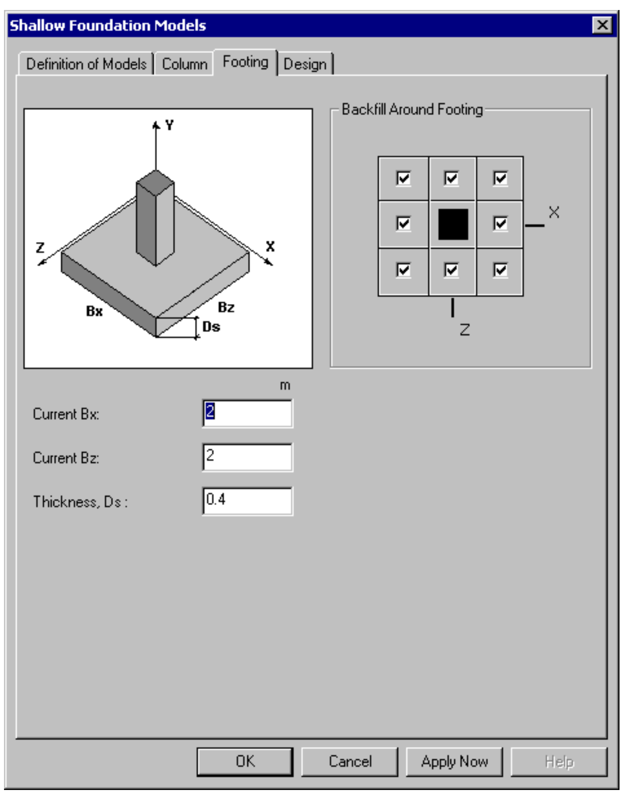
- In the first tab, give a name to this model, choose the concrete material, select the stratigraphical profile, and specify the backfill soil above corner footings.



- Select the **Column** tab and enter the dimensions of concrete column.



- Select the **Footing** tab.

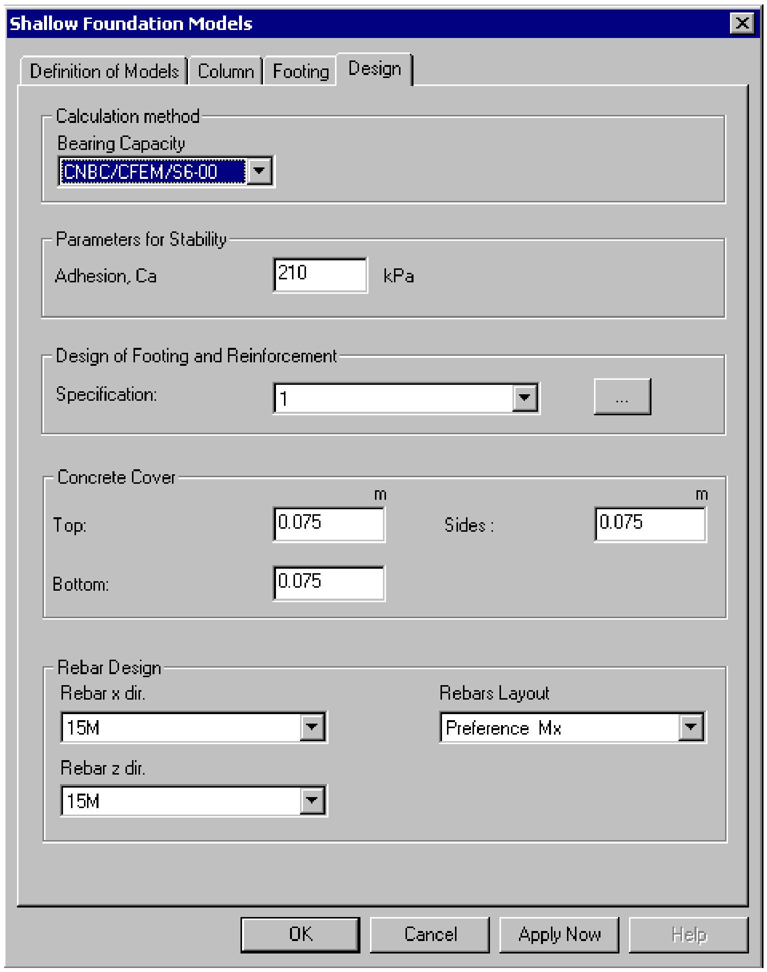


- Define the footing dimensions and thickness. *Bx* and *Bz* are specified as maximum dimensions for the design in the **Shallow Foundations Specifications** spreadsheet.

- Specify the backfill around the footing by checking appropriate boxes.

If bearing capacity is insufficient, dimensions must be increased in the specification and another analysis must be carried on until bearing capacity is OK.

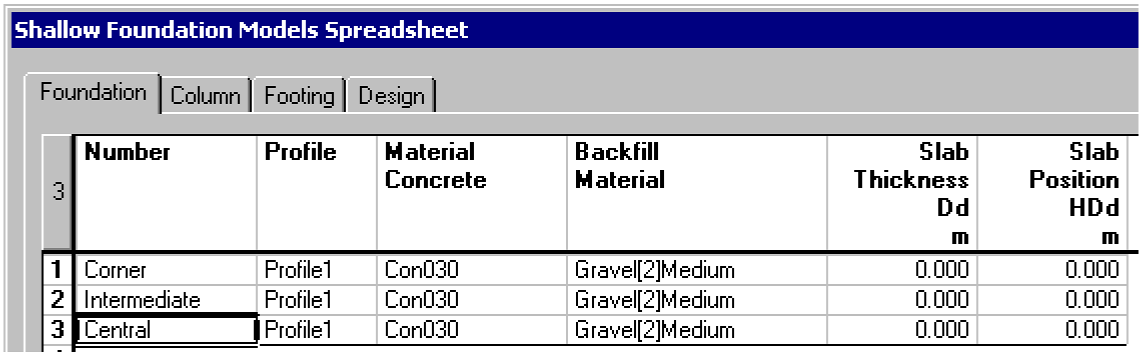
- Select the **Design** tab. Enter required parameters needed for the design.



- Click OK.

Back in the **Shallow Foundation Models** spreadsheet, we will copy the first model to create the second and third model. To do this operation quickly, highlight the line, right click, and choose **Duplicate** in contextual menu. Change the names of foundation models only because parameters are the same as the corner footing.

Click OK to exit the spreadsheet.



Shallow Foundation Models Spreadsheet

Foundation | Column | Footing | Design

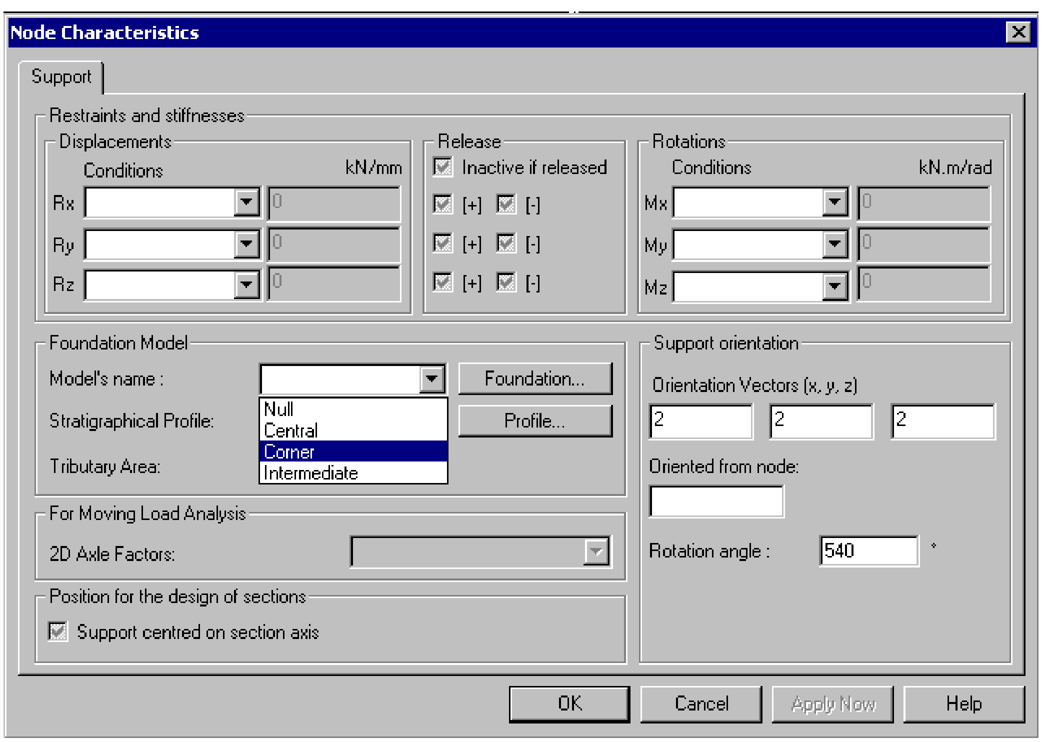
3	Number	Profile	Material Concrete	Backfill Material	Slab Thickness Dd m	Slab Position HDd m
1	Corner	Profile1	Con030	Gravel[2]Medium	0.000	0.000
2	Intermediate	Profile1	Con030	Gravel[2]Medium	0.000	0.000
3	Central	Profile1	Con030	Gravel[2]Medium	0.000	0.000

The next step is to assign foundation models to supports.

Assigning Foundation Models to Supports

Corner Footings

- Activate the *Support* icon on Elements toolbar and click once on each corner support while keeping the [Ctrl] key down. Select the **Properties** function
- Select the name of foundation model in the drop-down list box and click OK.

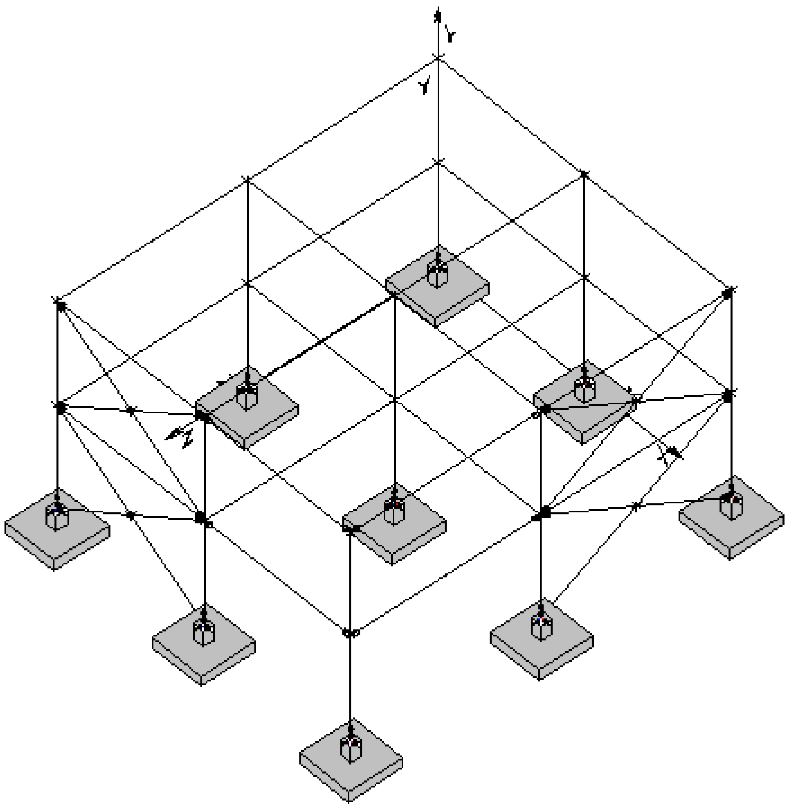


- Do the same to assign other foundation models to supports.

View Options

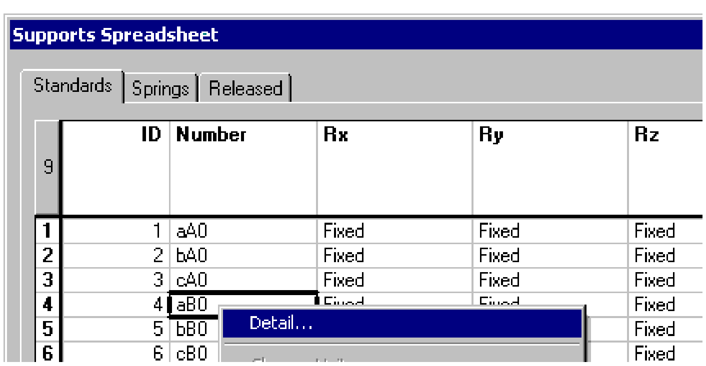
Footing Dimensions and Models

To display footings on screen, open the **View Options** dialog box and activate the *Foundation* box in the **View** tab. You can also display dimensions and names. Press OK.



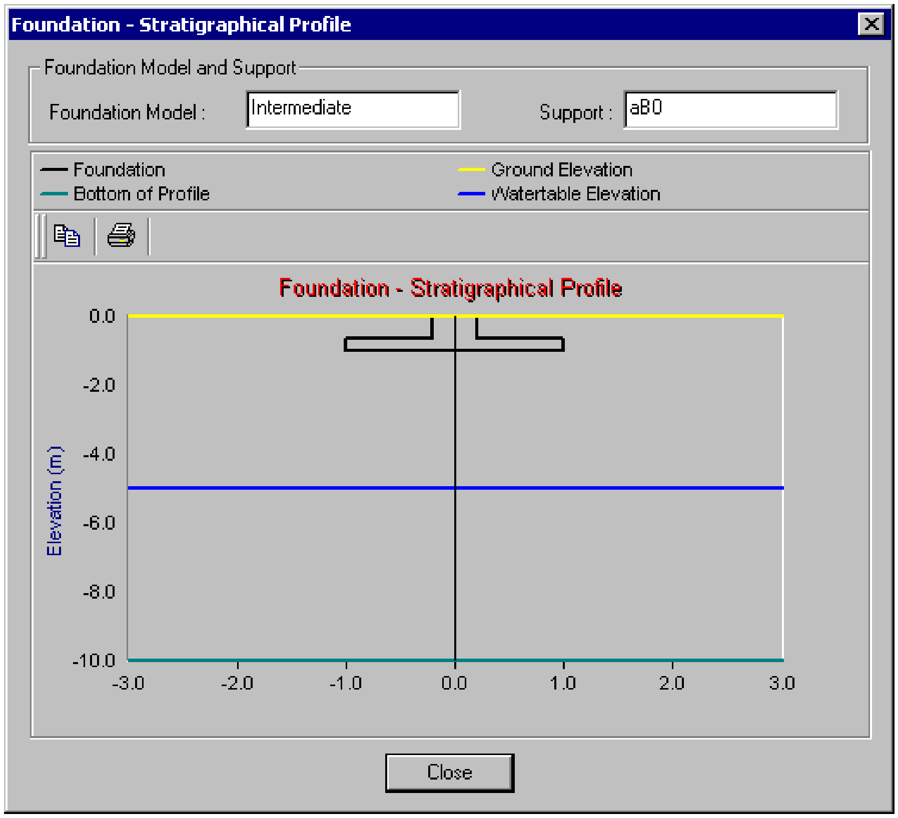
Display the Stratigraphical Profile

You can display the stratigraphical profile under a selected foundation support, through the **Supports** spreadsheet (**Structure** menu).



- Click in a cell (in the line corresponding to the support that interests you), right click, and select the function *Details* in the contextual menu.

The stratigraphical profile will be displayed in the following dialog box. Double-click in the "Legend" box and move it elsewhere.



Static Analysis or Design

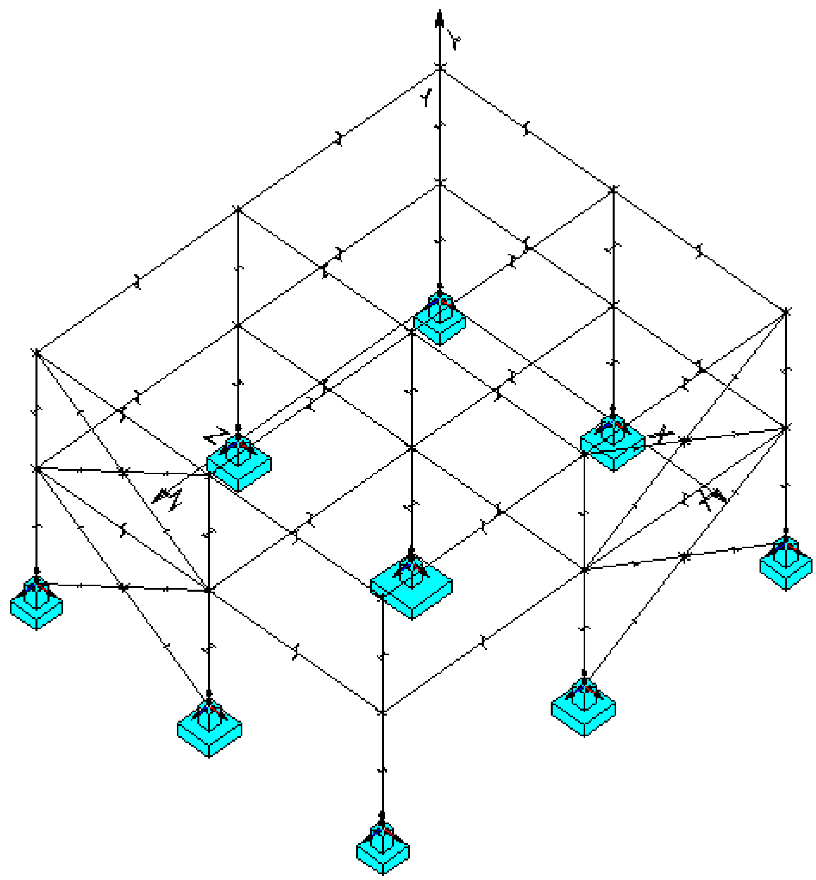
- Launch the static analysis by pressing icon  on Tools toolbar. VisualDesign will iterate until an optimum solution is found for each load combination and each support.

If you own a Design module (Steel, Concrete, or Timber), launch the design by clicking the icon . The foundation design will be part of the design process.

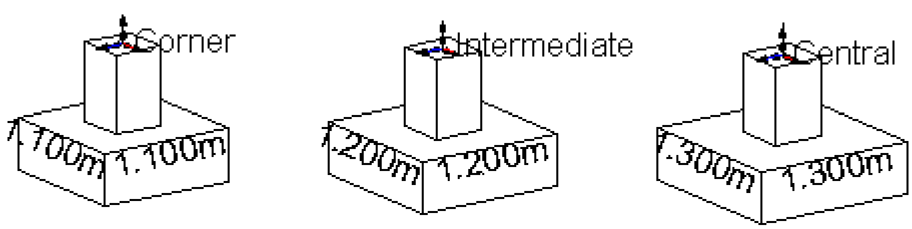
Graphical Results

View Options

- Open the **View Options** dialog box and check options "Dimensions" and "Name of Model" in the **View** tab.

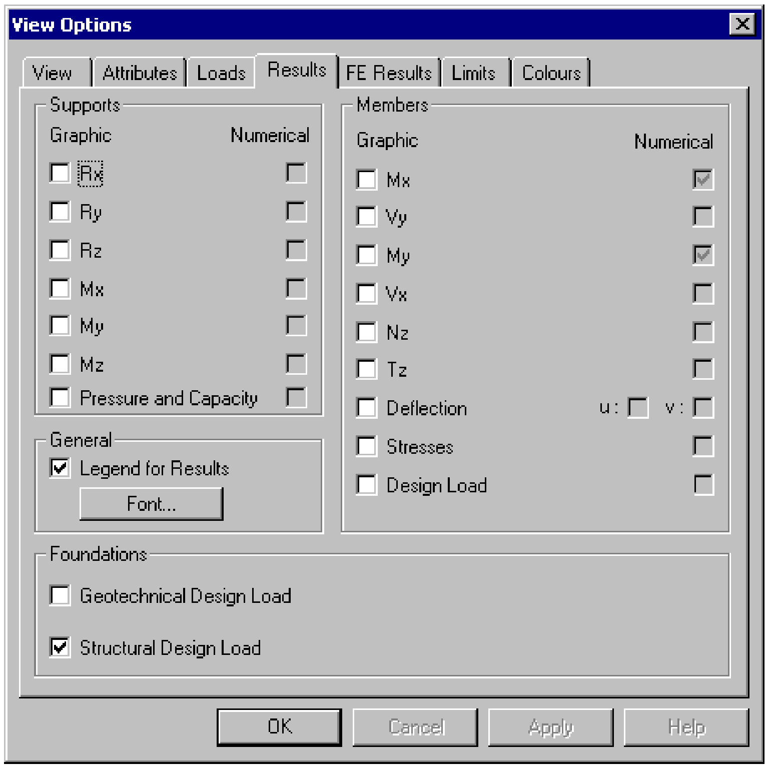


The designed footings are as follows:



Structural and Geotechnical Design Load

Activate the *Design Results* mode and open the **View Options** dialog box. Select the **Results** tab and activate the geotechnical or structural design load.



The table below includes structural and geotechnical design load obtained through the coloured legend.

Model	**Structural Design Load**	**Geotechnical Design Load**
Corner	Between 40% and 60%	Between 0% and 90%
Intermediate	Between 80% and 90%	Between 40% and 90%
Centre	100% and +	Between 80% and 90%

Numerical Results

Exact computed design loads are available in **Results / Foundations** menu:

- Select **Footings** and look at the calculated bearing capacity and geotechnical design loads.
- Select **Footing Reinforcement** to obtain structural design loads, minimum thickness, and calculated reinforcing bars.

Bearing Capacity and Geotechnical Design Loads

- Go to **Results / Foundations / Footings**. The **Shallow Foundation Results** spreadsheet presents geotechnical results for each support and load combination. It includes many results: Stability (sliding and overturning), reactions, bending moments, and much more.

N. B. Some columns have been masked.

Shallow Foundation Results Spreadsheet

20	Support	Load Combination	Bx m	Bz m	Bx Effective m	Bz Effective m	Bearing Capacity qr kPa	Calculated Pressure qf kPa	Rfx kN	Rfy kN	Rfz kN	Mfx kN.m	Mfz kN.m	Design Load %
1	aA0	DL1	1.100	1.100	1.089	0.995	881.06	295.09	0.69	319.92	7.05	16.76	-1.70	33.49
2	aA0	DL2	1.100	1.100	1.086	0.960	861.43	196.21	0.59	204.66	6.02	14.28	-1.44	22.78
3	aC0	DL1	1.100	1.100	0.680	1.062	600.17	498.98	75.67	360.06	-2.97	-6.89	-75.67	83.14
4	aC0	DL2	1.100	1.100	0.561	1.050	522.12	404.20	64.20	238.13	-2.55	-5.89	-64.20	77.42
5	cA0	DL1	1.100	1.100	1.086	0.687	600.71	477.90	-0.93	356.42	73.66	73.66	2.47	79.56
6	cA0	DL2	1.100	1.100	1.082	0.569	522.67	381.90	-0.80	234.96	62.43	62.43	2.10	73.07
7	cC0	DL1	1.100	1.100	1.100	1.100	940.93	318.10	0.01	384.82	0.03	0.04	-0.00	33.81
8	cC0	DL2	1.100	1.100	1.100	1.100	940.89	215.04	0.00	260.15	0.03	0.03	-0.00	22.86
9														
10	aB0	DL1	1.200	1.200	1.172	1.194	927.36	470.17	3.55	658.28	0.60	1.91	-9.08	50.70
11	aB0	DL2	1.200	1.200	1.169	1.194	925.48	356.70	3.07	497.72	0.50	1.59	-7.72	38.54
12	bA0	DL1	1.200	1.200	1.197	1.127	899.39	519.49	-0.25	700.76	10.90	25.65	1.00	57.76
13	bA0	DL2	1.200	1.200	1.197	1.118	894.33	399.01	-0.23	533.85	9.34	21.91	0.86	44.62
14	bC0	DL1	1.200	1.200	0.963	1.048	776.68	629.81	-75.45	635.57	-21.30	-48.18	75.45	81.09
15	bC0	DL2	1.200	1.200	0.932	1.027	757.11	498.71	-63.96	477.34	-18.26	-41.29	63.96	65.87
16	cB0	DL1	1.200	1.200	1.172	0.974	752.17	528.00	-3.16	602.64	-68.17	-68.24	8.31	70.20
17	cB0	DL2	1.200	1.200	1.168	0.943	728.75	407.72	-2.75	449.19	-57.68	-57.75	7.09	55.95
18														
19	bB0	DL1	1.300	1.300	1.299	1.298	940.76	826.22	-0.13	1393.60	0.19	1.08	0.68	87.82
20	bB0	DL2	1.300	1.300	1.299	1.298	940.76	671.23	-0.13	1132.10	0.16	0.89	0.59	71.35

Shallow Foundation Results Spreadsheet

20	Hrs kN	Hri kN	Hf Dir. x kN	Hr Dir. x kN	Design Load Sliding Dir. x %	Hf Dir. z kN	Hr Dir. z kN	Design Load Sliding z-Dir. %	Ratio for Eccentricity Dir. x %	Ratio for Eccentricity Dir. z %	Uplift Ratio %	Result
1	389.23	389.23	0.69	409.44	0.17	7.05	409.44	1.72	1.61	15.88	0.00	Ok
2	322.23	322.23	0.59	342.44	0.17	6.02	342.44	1.76	2.13	21.14	0.00	Ok
3	412.56	412.56	75.67	432.77	17.49	2.97	432.77	0.69	63.69	5.80	0.00	Ok
4	341.69	341.69	64.20	361.89	17.74	2.55	361.89	0.70	81.70	7.50	0.00	Ok
5	410.44	410.44	0.93	430.65	0.21	73.66	430.65	17.10	2.10	62.63	0.00	Ok
6	339.85	339.85	0.80	360.05	0.22	62.43	360.05	17.34	2.71	80.52	0.00	Ok
7	426.95	426.95	0.01	426.95	0.00	0.03	447.16	0.01	0.00	0.03	0.00	Ok
8	354.49	354.49	0.00	354.49	0.00	0.03	374.69	0.01	0.00	0.03	0.00	Ok
9												
10	624.54	624.54	3.55	646.18	0.55	0.60	646.18	0.09	3.83	0.81	0.00	Ok
11	531.21	531.21	3.07	552.86	0.56	0.50	552.86	0.09	4.31	0.89	0.00	Ok
12	649.23	649.23	0.25	670.88	0.04	10.90	670.88	1.62	0.40	10.17	0.00	Ok
13	552.21	552.21	0.23	573.86	0.04	9.34	573.86	1.63	0.45	11.40	0.00	Ok
14	611.33	611.33	75.45	632.98	11.92	21.30	632.98	3.36	32.98	21.06	0.00	Ok
15	519.37	519.37	63.96	541.01	11.82	18.26	541.01	3.38	37.22	24.03	0.00	Ok
16	592.20	592.20	3.16	613.84	0.52	68.17	613.84	11.11	3.83	31.46	0.00	Ok
17	503.00	503.00	2.75	524.65	0.52	57.68	524.65	10.99	4.38	35.71	0.00	Ok
18												
19	1093.93	1093.93	0.13	1117.01	0.01	0.19	1117.01	0.02	0.13	0.20	0.00	Ok
20	941.94	941.94	0.13	965.03	0.01	0.16	965.03	0.02	0.13	0.20	0.00	Ok

Bearing capacities and geotechnical design loads are OK. Close the spreadsheet.

Footing Reinforcement

Go to **Results** / **Foundations / Footing Reinforcement**. The spreadsheet supplies the required number of rebars, length and spacing, for each footing model. Look at footing shear forces and resistance (Qf and Qr), punching shear (vf and vc), and minimum required thickness (Hmin).

Yellow columns are editable. Double click to modify rebar dimensions. Then, VisualDesign will automatically recalculate spacing and resistances in the spreadsheet.

Design of Footing Reinforcement Spreadsheet

3	Name	Quantity	Bx m	Bz m	Hs m	Hmin. m	Material Concrete	Steel	Rebars Layout	Minimum Cover m	Side Cover m	Saf	Waf
1	Corner	4	1.100	1.100	0.400	0.360	Con030	G30.18-400R	Preference Mx	0.075	0.075	1.25	1.50
2	Intermediate	4	1.200	1.200	0.400	0.415	Con030	G30.18-400R	Preference Mx	0.075	0.075	1.25	1.50
3	Central	1	1.300	1.300	0.400	0.483	Con030	G30.18-400R	Preference Mx	0.075	0.075	1.25	1.50

Design of Footing Reinforcement Spreadsheet

3	Rebar dim. x dir.	No. Rebars x dir.	Spacing c/c Rebars x-dir. m	ρ x %	Length x Dir. m	d x dir. m	Mfx kN.m/m	Mrx kN.m/m	Qfx kN/m	Qrx kN/m	Rebar dim. z dir.	No. Rebars z dir.
1	15M	5	0.234	0.29	0.950	0.317	33.52	57.25	18.06	205.66	15M	5
2	15M	5	0.259	0.26	1.050	0.317	47.13	55.15	48.90	205.66	15M	5
3	15M	8	0.162	0.39	1.150	0.317	83.58	85.39	109.79	205.66	15M	9

Design of Footing Reinforcement Spreadsheet

3	No. Rebars z dir.	Spacing c/c Rebars z-dir. m	ρ z %	Length z Dir. m	d z Dir. m	Mfz kN.m/m	Mrz kN.m/m	Qfz kN/m	Qrz kN/m	vf MPa	vc MPa	Design Load %	Message
1	5	0.234	0.30	0.950	0.301	32.78	57.25	26.23	197.69	0.57	1.31	58.56	Ok
2	5	0.259	0.28	1.050	0.301	44.05	55.15	54.51	197.69	0.96	1.31	85.46	Ok
3	9	0.142	0.46	1.150	0.301	83.63	96.06	123.07	197.69	1.59	1.31	121.24	vf > vc

The central footing is not adequate. The punching shear is too high and the structural design load is 124%. We can see that the minimum required thickness for this footing is 483mm.

- Go back to the **Shallow Foundation Models** spreadsheet and specify a thickness of 475mm for the central footing. Launch another static analysis.

Design of Footing Reinforcement Spreadsheet

3	Name	Quantity	Bx m	Bz m	Hs m	Hmin. m	Material Concrete	Steel	Rebars Layout	Minimum Cover m
1	Corner	4	1.100	1.100	0.400	0.355	Con030	G30.18-400R	Preference Mx	0.075
2	Intermediate	4	1.100	1.100	0.400	0.398	Con030	G30.18-400R	Preference Mx	0.075
3	Central	1	1.200	1.200	0.475	0.465	Con030	G30.18-400R	Preference Mx	0.075

Design of Footing Reinforcement Spreadsheet

3	Mfz kN.m/m	Mrz kN.m/m	Qfz kN/m	Qrz kN/m	vf MPa	vc MPa	Design Load %	Message
1	31.79	57.25	25.43	197.69	0.49	1.31	57.07	Ok
2	38.27	57.25	30.61	197.69	0.89	1.31	72.07	Ok
3	76.63	88.41	22.99	233.48	1.11	1.31	86.68	Ok

The thickness is sufficient and dimensions have dropped to 1.2m x1.2m.

Soil-Structure Interaction

General

You can consider soil-structure interaction in a structural analysis and shallow foundation design. An *Interaction* type of load must be defined in the **Loads Definition** spreadsheet and included in appropriate load combinations.

To obtain settlements, foundation supports must be modeled using the option *Secant modulus K*, which is available as degrees of freedom in the **Support** tab of **Node Characteristics** dialog box. With this option, VisualDesign calculates settlements using the secant modulus of soils that are part of the stratigraphical profile, for each footing. The calculated soil stiffness is supplied in the **Support** tab when a foundation model is assigned to a foundation support.

When the soil-structure interaction is considered, the program performs iterations until it reaches compatibility between footing settlements, stresses in the structure and support reactions caused by footing settlements.

The iterative process is stopped when the number of iterations exceeds the maximum specified. Then, a message can be displayed on screen such as: "The solution is acceptable but footing dimensions could not be optimized." The iterative process rarely requires more than five iterations. If more than five iterations are required, the solution cannot be optimized. However, results are valid since calculated capacities are greater than 100%.

Shallow Foundation Design with Soil/Structure Interaction

In the next example, soil-structure interaction will be considered. We will use the structural model that we previously analysed (the last one).

Foundation parameters and footing models are already defined. The next step is to model foundation supports, define the *Interaction* type of load, and include this load in appropriate serviceability load combinations.

Foundation Supports

Secant Modulus, K

- Use the option *Secant modulus K* for appropriate degrees of freedom to model spring supports under the footings and automatically obtain the soil stiffness. This soil stiffness is required for the calculation of footing settlements.

N. B. In this example, foundation models are already assigned to foundation supports.

- Activate the Support icon, select all supports and press the short cut keys [Ctrl]+H to open the **Supports** spreadsheet. Use the **Replace** function of contextual menu and modify degrees of freedom to *Secant modulus K*.

Supports Spreadsheet

Standards | Springs | Released

9	Number	Rx	Ry	Rz	Mx	My	Mz	Foundation
1	aA0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Corner
2	bA0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Intermediate
3	cA0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Corner
4	aB0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Intermediate
5	bB0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Central
6	cB0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Intermediate
7	aC0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Corner
8	bC0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Intermediate
9	cC0	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Secant mod. K	Corner

Linear stiffness (Kx, Ky, and Kz) and torsional rotation (Krx, Kry, and Krz) are automatically calculated and written in the **Spring**s tab.

Supports Spreadsheet

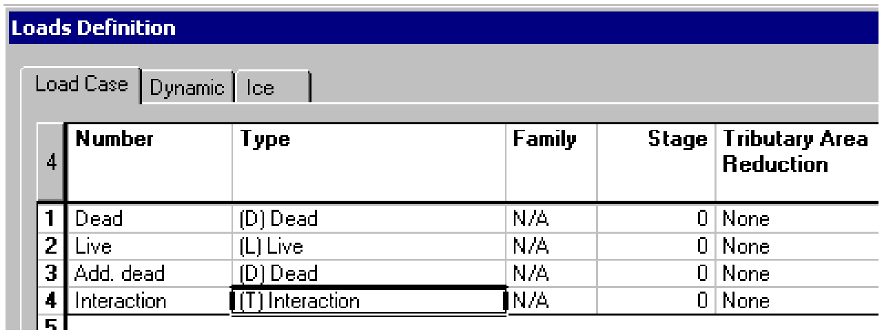
Standards | Springs | Released

9	Number	Kx kN/mm	Ky kN/mm	Kz kN/mm	Krx kN.m/rad	Kry kN.m/rad	Krz kN.m/rad	Tributary Area m²	Profile
1	aA0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
2	bA0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
3	cA0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
4	aB0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
5	bB0	1.10	84.50	1.10	1267.43	3.6979e+008	1267.43	0.00	Null
6	cB0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
7	aC0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
8	bC0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null
9	cC0	0.98	77.50	0.98	976.82	3.6979e+008	976.82	0.00	Null

- Close the spreadsheet.

Interaction Load Case

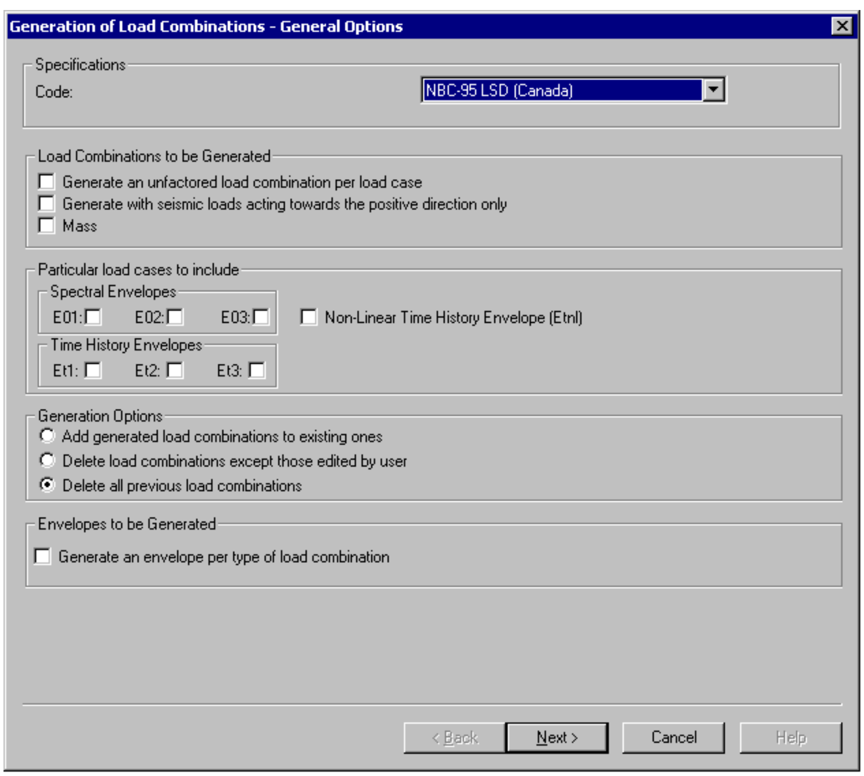
- Select the **Loads Definition** spreadsheet and insert a line. Give a number (name) to this load case, double-click in the "Type" column, expand the appropriate root, and select *Interaction*.

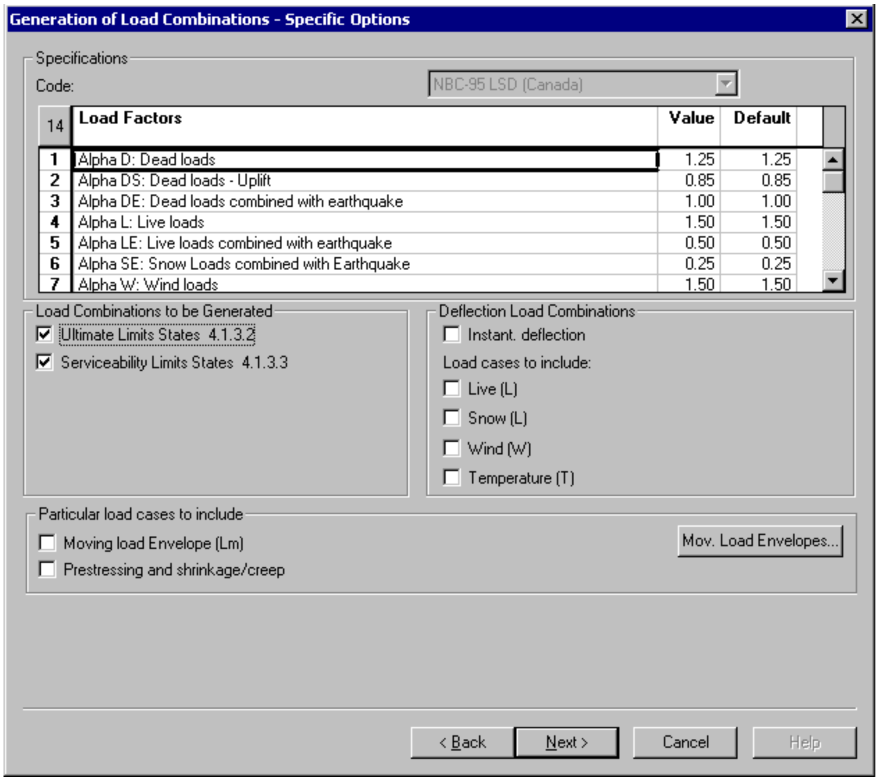


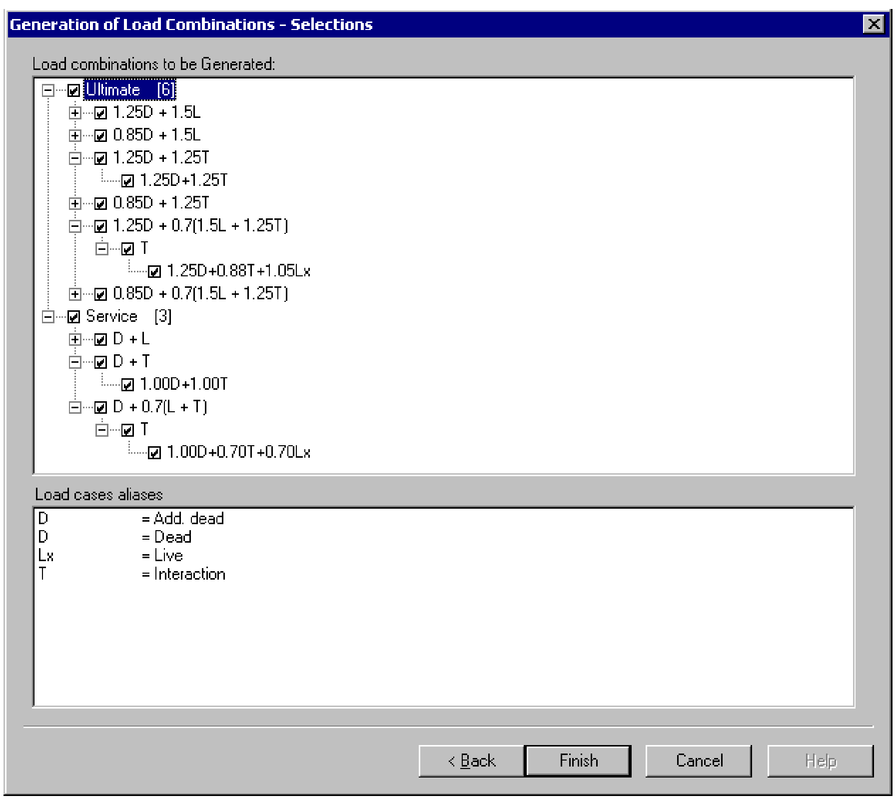
- Close the spreadsheet.

Load Combinations

Serviceability load combinations are required to consider soil/structure interaction. Use the **Load Combination Generation Wizard** to generate load combinations according to the Canadian National Building Code.




Generation of Load Combinations - Specific Options
Specifications
Code:
NBC-95 LSD (Canada)
14
Load Factors
Value
Default
1 Alpha D: Dead loads 1.25 1.25
2 Alpha DS: Dead loads - Uplift 0.85 0.85
3 Alpha DE: Dead loads combined with earthquake 1.00 1.00
4 Alpha L: Live loads 1.50 1.50
5 Alpha LE: Live loads combined with earthquake 0.50 0.50
6 Alpha SE: Snow Loads combined with Earthquake 0.25 0.25
7 Alpha W: Wind loads 1.50 1.50
Load Combinations to be Generated
Ultimate Limits States 4.1.3.2
Serviceability Limits States 4.1.3.3
Deflection Load Combinations
Instant. deflection
Load cases to include:
Live (L)
Snow (L)
Wind (W)
Temperature (T)
Particular load cases to include
Moving load Envelope (Lm)
Prestressing and shrinkage/creep
Mov. Load Envelopes...
< Back
Next >
Cancel
Help


Generation of Load Combinations - Selections
Load combinations to be Generated:
Ultimate [6]
1.25D + 1.5L
0.85D + 1.5L
1.25D + 1.25T
1.25D+1.25T
0.85D + 1.25T
1.25D + 0.7(1.5L + 1.25T)
T
1.25D+0.88T+1.05Lx
0.85D + 0.7(1.5L + 1.25T)
Service [3]
D + L
D + T
1.00D+1.00T
D + 0.7(L + T)
T
1.00D+0.70T+0.70Lx
Load cases aliases
D = Add. dead
D = Dead
Lx = Live
T = Interaction
< Back
Finish
Cancel
Help

The following load combinations were generated:

Load Combinations

Load Combinations | Load Factors

9	Number	Status	Definition	Stage
1	DL1	Ultimate	1.25D+1.50Lx	0
2	DL2	Ultimate	0.85D+1.50Lx	0
3	DLT5	Ultimate	1.25D+0.88T+1.05Lx	0
4	DLT6	Ultimate	0.85D+0.88T+1.05Lx	0
5	DT3	Ultimate	1.25D+1.25T	0
6	DT4	Ultimate	0.85D+1.25T	0
7	DL7	Service	1.00D+1.00Lx	0
8	DLT9	Service	1.00D+0.70T+0.70Lx	0
9	DT8	Service	1.00D+1.00T	0

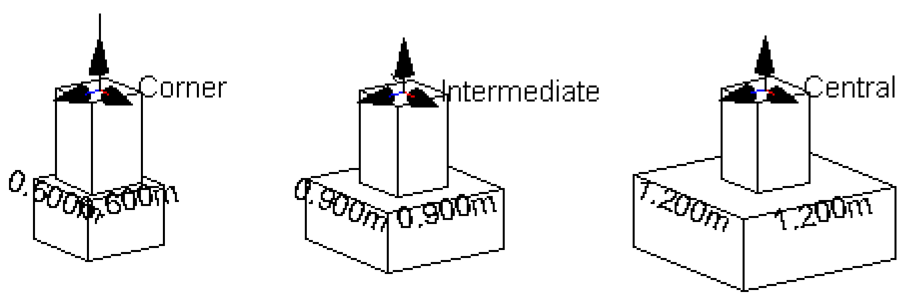
Static Analysis or Design

- Launch the static analysis or design.

Results

Footing Dimensions and

The corner and intermediate footings are smaller than before (static analysis without soil/structure interaction).



Design Loads

- Activate the *Design Results* mode.
- Open the **View Options** dialog box, go to **Results** tab and activate the display of foundation design loads.

The geotechnical and structural design loads are the same as before.

Model	Structural Design Load	Geotechnical Design Load
Corner	Between 20% and 40%	Between 70% and 90%
Intermediate	Between 40% and 60%	Between 70% and 90%
Centre	Between 90% and 100%	Between 90% and 100%

Settlements (Footings)

- Go to **Results** / **Foundations** / **Footings.** Settlements are calculated at each support, for each serviceability load combinations:

Shallow Foundation Results Spreadsheet

83	Model	Support	Load Combination	Bx m	Bz m	Bx Effective m	Bz Effective m	Bearing Capacity qr kPa	Calculated Pressure qf kPa	Settlement dy mm
75	Central	bB0	DL1	1.200	1.200	1.173	1.174	986.03	945.87	0.00
76	Central	bB0	DL2	1.200	1.200	1.174	1.174	986.35	801.83	0.00
77	Central	bB0	DT3	1.200	1.200	1.174	1.174	986.48	442.10	0.00
78	Central	bB0	DT4	1.200	1.200	1.174	1.174	986.48	299.35	0.00
79	Central	bB0	DLT5	1.200	1.200	1.174	1.174	986.38	793.98	0.00
80	Central	bB0	DLT6	1.200	1.200	1.174	1.174	986.46	650.80	0.00
81	Central	bB0	DL7	1.200	1.200	1.174	1.174	986.45	688.42	-10.85
82	Central	bB0	DT8	1.200	1.200	1.174	1.174	986.48	353.61	-5.39
83	Central	bB0	DLT9	1.200	1.200	1.174	1.174	986.49	586.92	-9.21

Bearing capacities are OK.

At the far right of this spreadsheet, you will find geotechnical design loads and others, including the settlement ratio according to allowable settlements (25mm).

Shallow Foundation Results Spreadsheet

83	Design Load %	Design Load Sliding Dir. x %	Design Load Sliding z-Dir. %	Design Load Settlement %	Result
1	72.23	0.28	1.04	0.00	Ok
2	60.70	0.23	0.93	0.00	Ok
3	34.98	0.16	0.62	0.00	Ok
4	23.65	0.12	0.48	0.00	Ok
5	60.93	0.23	0.91	0.00	Ok
6	49.54	0.20	0.82	0.00	Ok
7	52.64	0.21	0.84	14.98	Ok
8	27.97	0.14	0.54	7.67	Ok
9	45.23	0.19	0.76	12.79	Ok

Footing Reinforcements and Structural Design Loads

- **Results / Foundations / Footing reinforcements**.

Design of Footing Reinforcement Spreadsheet

3	Name	Quantity	Bx m	Bz m	Hs m	Hmin. m	Material Concrete	Steel	Rebars Layout	Minimum Cover m
1	Corner	4	0.600	0.600	0.400	0.214	Con030	G30.18-400R	Preference Mx	0.075
2	Intermediate	4	0.900	0.900	0.400	0.331	Con030	G30.18-400R	Preference Mx	0.075
3	Central	1	1.200	1.200	0.475	0.462	Con030	G30.18-400R	Preference Mx	0.075

Design of Footing Reinforcement Spreadsheet

3	Mfz kN.m/m	Mrz kN.m/m	Qfz kN/m	Qrz kN/m	vf MPa	vc MPa	Design Load %	Message	Steel kg	Concrete m³
1	4.05	19.20	0.00	197.69	0.33	1.31	25.32	Ok	16.96	0.96
2	24.39	42.67	0.00	197.69	0.72	1.31	57.15	Ok	37.68	1.68
3	74.54	88.41	22.36	233.48	1.08	1.31	98.48	Ok	21.43	0.78

The design is OK.

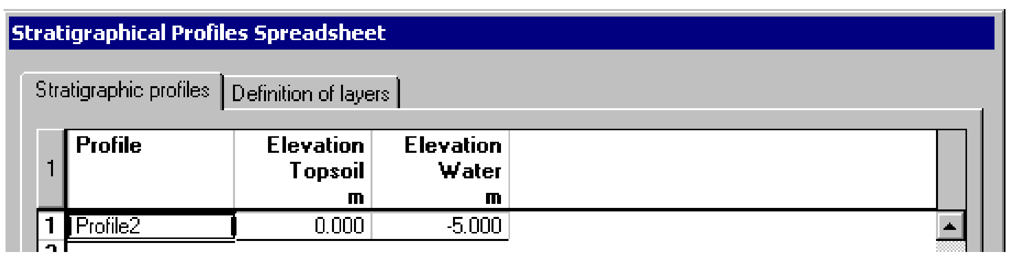
Pile Foundations

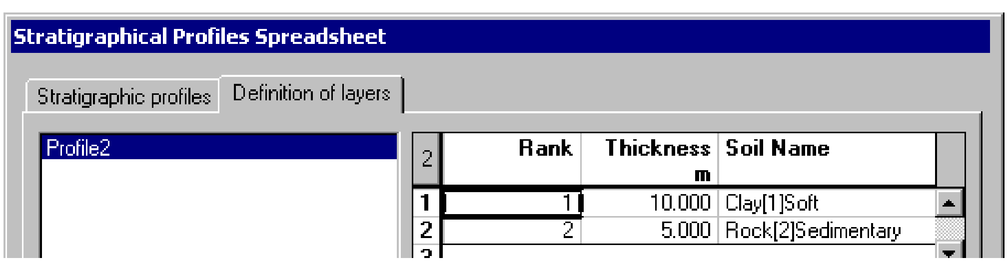
We will add piles below the footings that we designed (shallow foundations, page 176). However, footing dimensions will be increased to 3m x 3m and thickness will be fixed to 500mm because of punching shear and shear forces developed around pile heads. The stratigraphical profile will also be modified.

The procedure to define pile foundations is the same as for shallow foundations except that more tabs (Definition of piles) are included in the **Deep Foundation Models** dialog box.

Stratigraphical Profile

- Open the **Stratigraphical Profiles** spreadsheet and enter the following data.

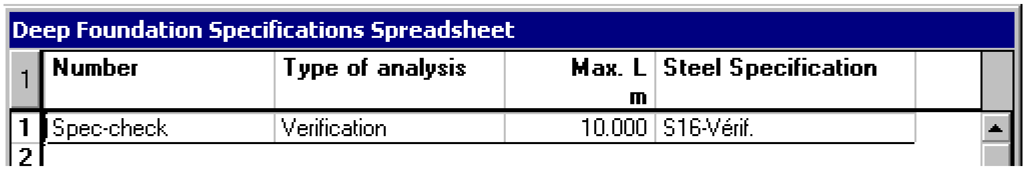




Specifications for Pile Foundations

Piles cannot be structurally designed. Piles will be verified according to the chosen steel shape, maximum length, and steel specification.

- Select **Structure** / **Specifications / Deep Foundations**. Insert a line and give a name to the specification. Enter the maximum pile length and select a steel specification that will be used for the structural verification of the pile.



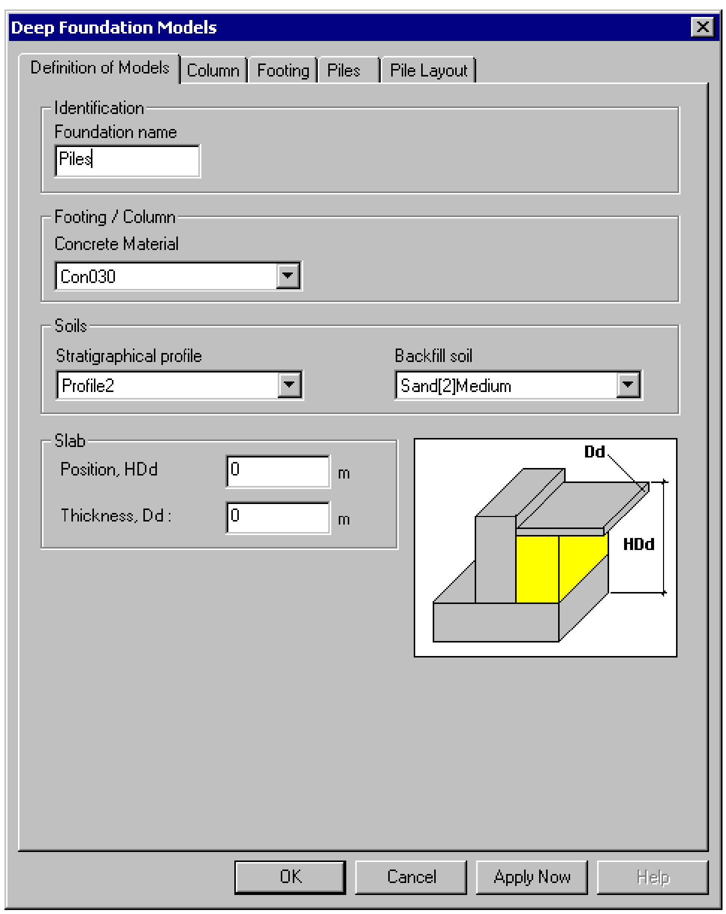
- Close the spreadsheet.

Definition of Foundation Model

- Select **Structure** / **Foundation Models** / **Deep**. Insert a line in the spreadsheet and open the dialog box. (Select a cell, right click and choose *Detail* in the contextual menu.)

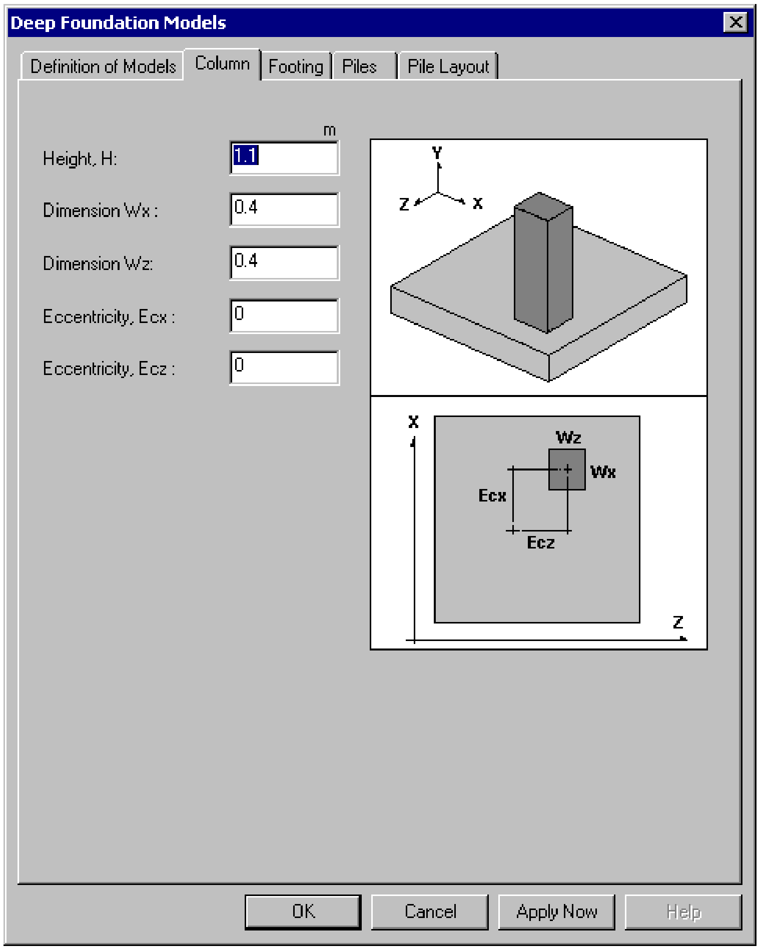
The Definition tab

- Give a name to the model, select the concrete material, stratigraphical profile, and backfill soil among lists box.



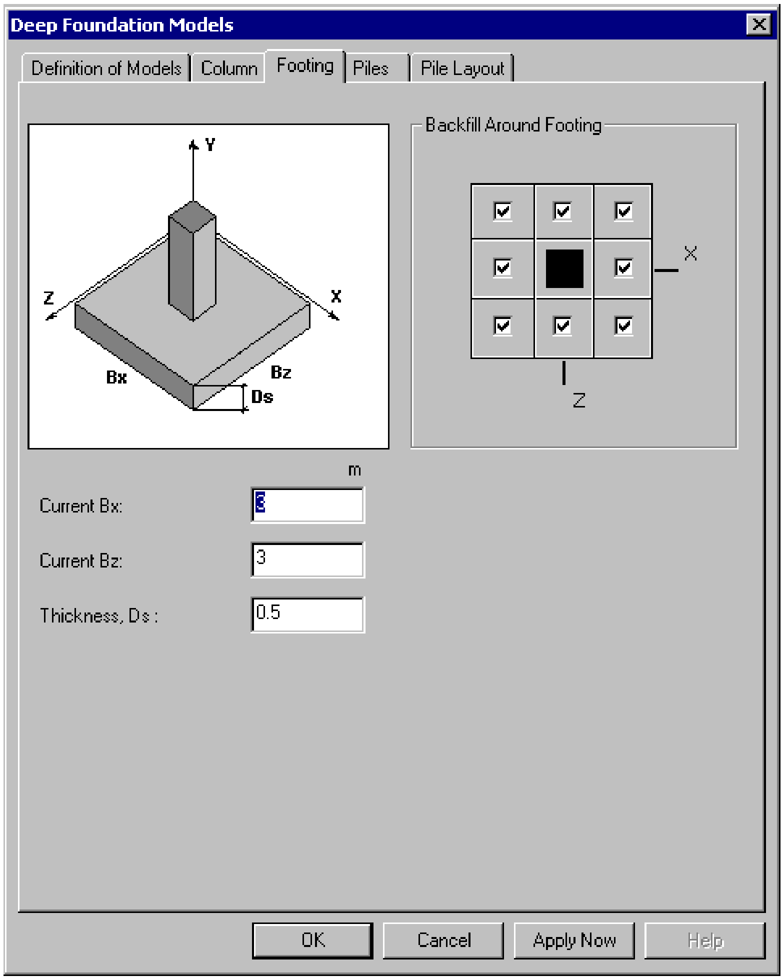
The Column tab

- Go to the **Column** tab and enter the concrete column dimensions and height according to global axis system.



The Footing tab

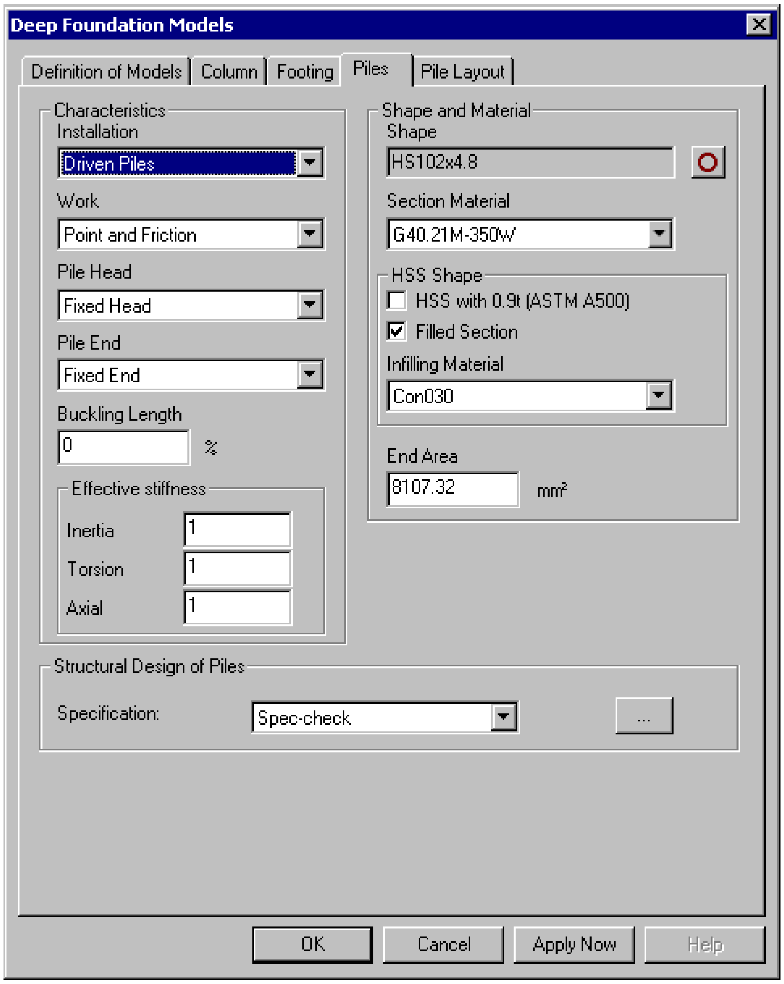
- Select the **Footing** tab and enter the footing dimensions and thickness. Specify the location of backfill over the footing.



The Piles tab

- Select the **Piles** tab. Many parameters must be specified for the verification of pile foundations. To know more, press F1 while the dialog box is open and VisualDesign On-line Help will open at this specific topic.

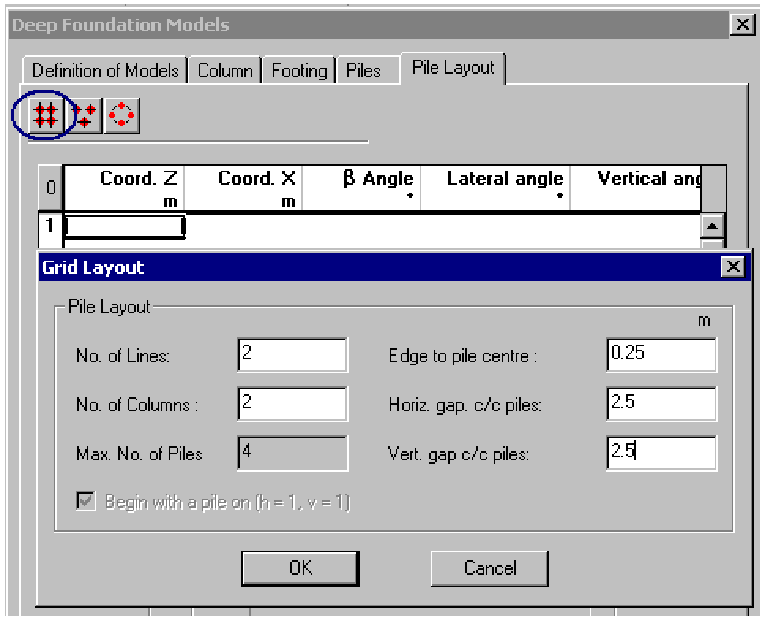
VisualDesign will calculate the geotechnical capacity and structural strength of piles according to the deep foundation specification and the steel shape chosen in this tab.



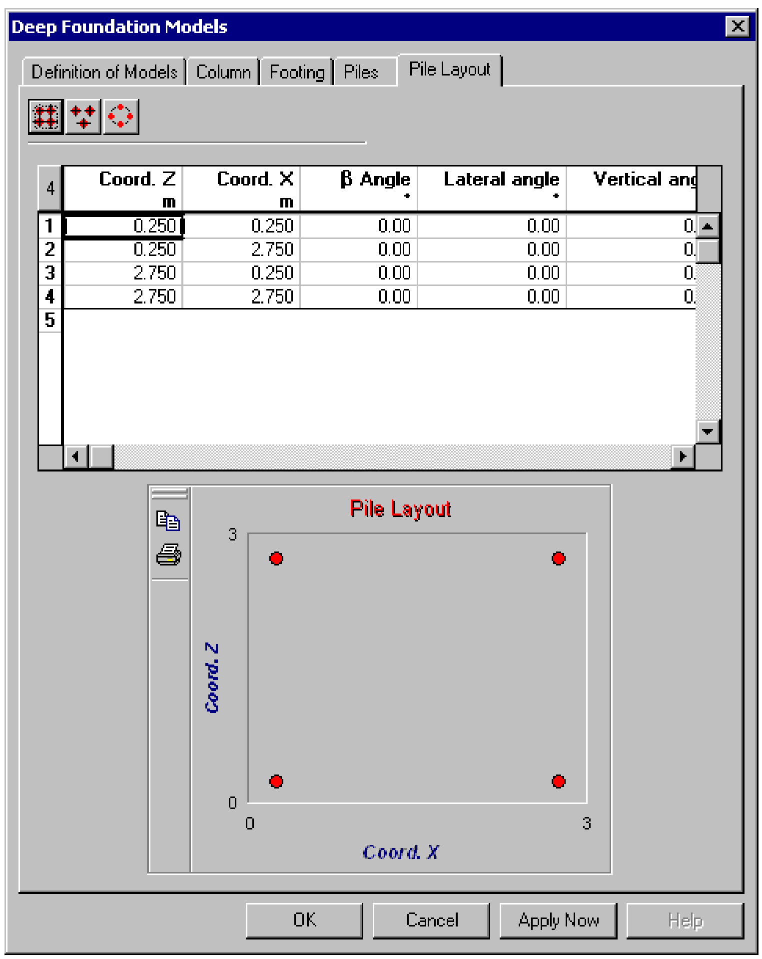
The Pile Layout tab

- Select the **Pile Layout** tab and define the pile layout under the footing. To do that quickly, use one of the three buttons posted in the upper part of the dialog box. Each button represents a layout and opens a specific dialog box. In our example, we pressed the first button.

- In the **Grid Layout** dialog box, enter the number of pile lines (longitudinal) and columns (transverse). Specify the edge distance, which is from border to centre of pile.



- Click OK to close the **Grid Layout** dialog box.



Piles' coordinates are written in the spreadsheet and the layout is displayed in a box. The layout can be printed.

The **Pile Layout** spreadsheet includes other parameters: *Beta angle* of steel section, *Lateral Angle* and *Vertical Angle*, which are the pile slope relatively to xz and xy planes.

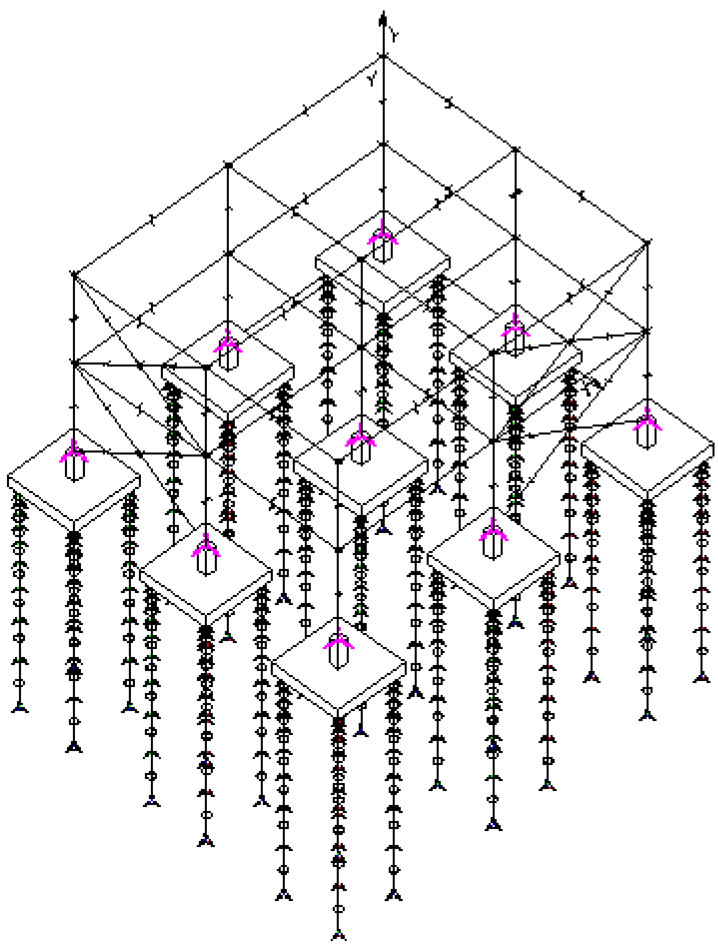
- Click OK to exit the dialog box and save data.

Assigning the Model to Supports

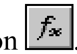
- Activate the *Support* icon. Select all supports and press the **Properties** icon. Select the *Piles* model in the **Support** tab and click OK.

View Options

Select the **View Options** dialog box and display foundation dimensions and name through the **View** tab. Click OK.



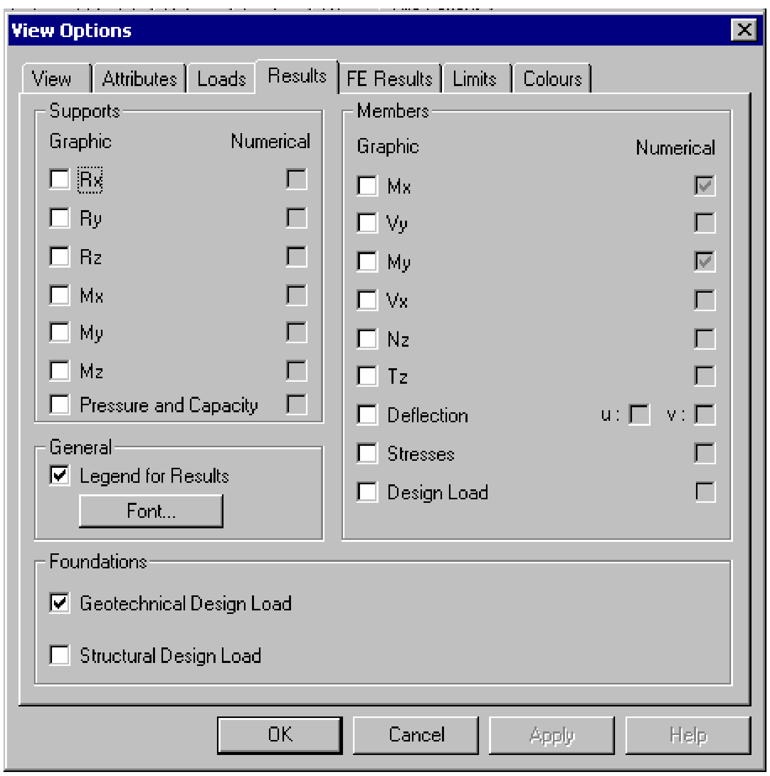
Static Analysis

- Launch the static analysis by pressing icon . VisualDesign will iterate until an optimum solution is found for each load combination and each support. If you possess one of design modules (steel, timber, or concrete), press the Design  icon.

Results

Piles' Structural and Geotechnical Design Load

- Verify the piles' design load through the coloured legend. Select The Design results mode on Activation toolbar and open the **View Options** dialog box. Activate a type of results and corresponding legend in the **Results** tab. Click OK.



We can see that the design of piles is OK.

Geotechnical Results

- Select **Results / Foundation / Piles**. This spreadsheet includes geotechnical results at each support, for each load combination.

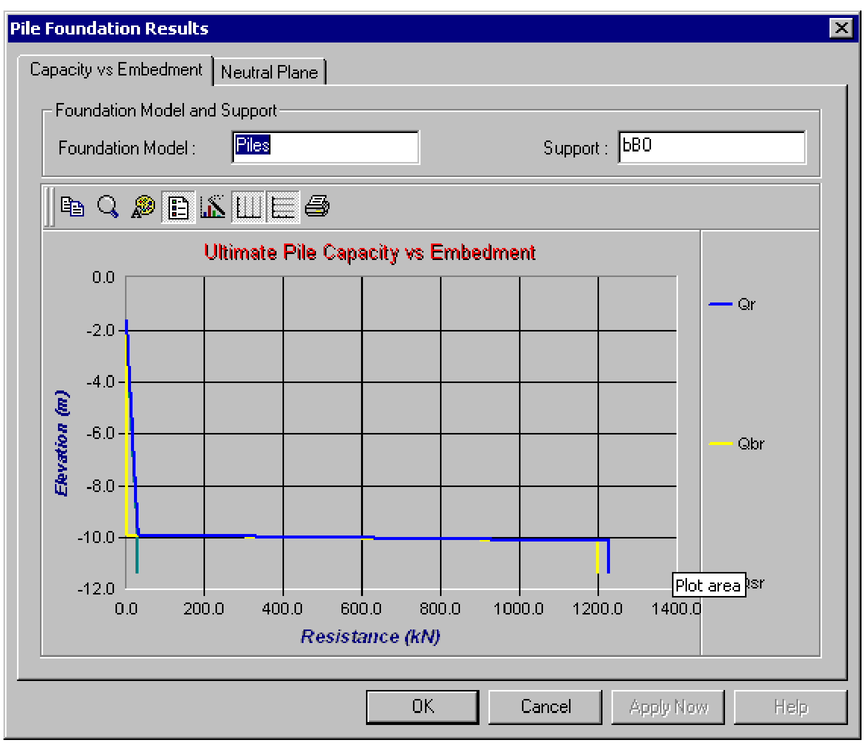
Geotechnical Results for Pile Groups Spreadsheet

18	Model	Support	Load Combination	Length m	Cr Geo. Group kN	Tr Geo. Group kN	Settlement dy m	Rfx kN	Rfy kN	Rfz kN	Mfx kN.m	Mfz kN.m	Geotech. Design Load Compression %	Result
1	Piles	aA0	DL1	10.000	4920.99	129.57	-0.001	0.74	551.98	6.22	17.36	-2.17	11.22	Ok
2	Piles	aA0	DL2	10.000	4920.99	129.57	-0.001	0.63	399.65	5.30	14.74	-1.84	8.12	Ok
3	Piles	bA0	DL1	10.000	4920.99	129.57	-0.003	-0.43	931.09	9.14	24.29	2.02	18.92	Ok
4	Piles	bA0	DL2	10.000	4920.99	129.57	-0.003	-0.38	723.60	7.81	20.70	1.71	14.70	Ok
5	Piles	cA0	DL1	10.000	4920.99	129.57	-0.001	-1.05	588.25	9.94	14.71	3.55	11.95	Ok
6	Piles	cA0	DL2	10.000	4920.99	129.57	-0.001	-0.92	429.85	8.49	12.56	3.04	8.74	Ok
7	Piles	aB0	DL1	10.000	4920.99	129.57	-0.003	3.83	887.96	0.05	0.05	-11.57	18.04	Ok
8	Piles	aB0	DL2	10.000	4920.99	129.57	-0.003	3.29	686.92	0.01	-0.07	-9.83	13.96	Ok
9	Piles	bB0	DL1	10.000	4920.99	129.57	-0.007	-0.39	1601.56	-0.65	-2.35	2.16	32.55	Ok
10	Piles	bB0	DL2	10.000	4920.99	129.57	-0.006	-0.37	1298.47	-0.57	-2.07	1.88	26.39	Ok
11	Piles	cB0	DL1	10.000	4920.99	129.57	-0.003	-3.59	835.90	-4.95	-7.19	11.72	16.99	Ok
12	Piles	cB0	DL2	10.000	4920.99	129.57	-0.002	-3.12	641.36	-4.12	-5.98	10.01	13.03	Ok
13	Piles	aC0	DL1	10.000	4920.99	129.57	-0.001	8.40	591.75	-3.14	-9.14	-13.14	12.03	Ok
14	Piles	aC0	DL2	10.000	4920.99	129.57	-0.001	7.19	432.95	-2.69	-7.83	-11.22	8.80	Ok
15	Piles	bC0	DL1	10.000	4920.99	129.57	-0.003	-7.55	869.75	-16.66	-40.63	11.88	17.67	Ok
16	Piles	bC0	DL2	10.000	4920.99	129.57	-0.003	-6.34	670.30	-14.26	-34.78	9.97	13.62	Ok
17	Piles	cC0	DL1	10.000	4920.99	129.57	-0.002	0.04	613.70	0.03	-0.09	0.03	12.47	Ok
18	Piles	cC0	DL2	10.000	4920.99	129.57	-0.001	0.02	452.42	0.02	-0.08	0.05	9.19	Ok
19														

Close

Graphic results are available through this spreadsheet. Select a line and click the button located at the bottom of dialog box.

Ultimate Pile Capacity vs. Embedment



This graph shows the point bearing capacity and compression/friction bearing capacity of piles.

Where:

Qr = Qbr + Qsr

Qbr = ϕcp * Qb

Qsr = ϕcs * Qs

Qb is the nominal point bearing capacity and ϕcp, the resistance factor for compression/point bearing;

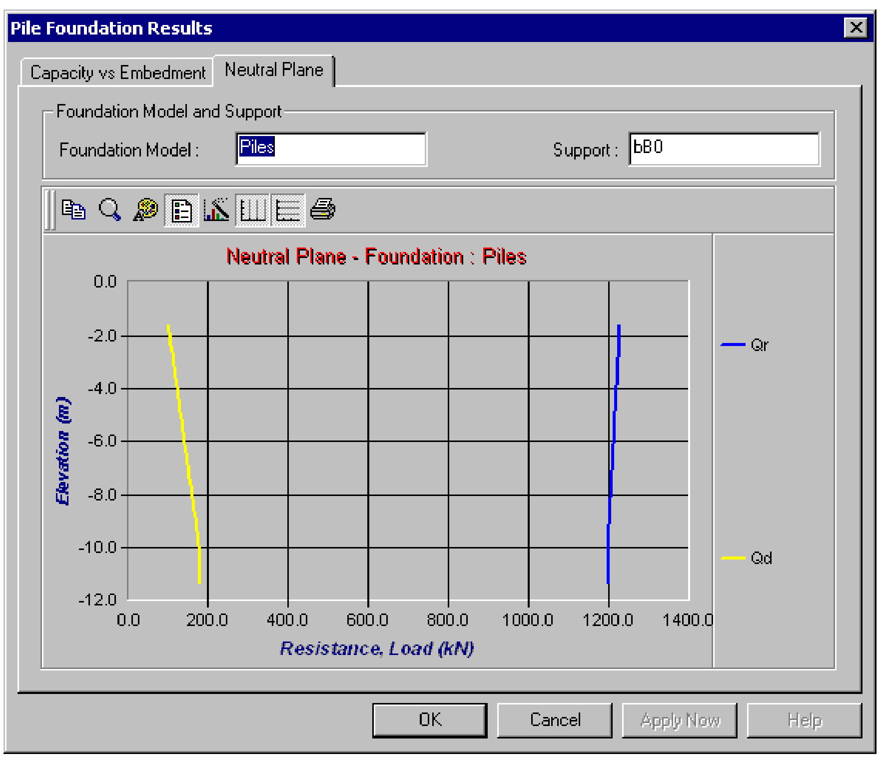
Qs is the nominal friction capacity and et ϕcs, the resistance factor for compression/friction bearing;

This graph indicates that the total ultimate capacity (Qr) of the pile is around 1200 kN because it is bearing on rock. (Place your cursor on the graph to obtain the numerical value at this point.)

Location of Neutral Plane

The neutral plane corresponds to the depth where negative lateral friction on pile becomes positive, i.e. where the two curves meet. In this example, piles are bearing on rocks, so, this graph is not relevant.

Qd represents the distribution of dead load according to depth of pile and Qr, the variation of total capacity of pile (Qbu + Qs) with its depth.



Forces and Reactions in Each Pile

- Go to **Results / Foundations / Piles Reactions**. This spreadsheet includes results for each pile within a pile group, for each load combination.

We selected the central foundation, which is the most critical.

This part of spreadsheet shows the minimum and maximum axial forces acting on each pile, for each load combinations. "x" is the lateral deflection of each pile, and "%x", the calculated ratio according to allowable lateral deflection.

Structural and Geotechnical Resistance for each Pile

72	Support	Load Combination	Pile Number	Nz Max. kN	Nz Min. kN	Mfy kN.m	Vfx kN	x mm	% x %	Mfx kN.m	Vfy kN
33	bB0	DL1	5	-399.39	-399.39	0.04	0.03	0.05	0.21	0.03	0.08
34	bB0	DL1	6	-400.33	-400.33	0.13	0.19	0.06	0.25	0.03	0.08
35	bB0	DL1	7	-400.45	-400.45	0.07	0.05	0.05	0.21	0.15	0.24
36	bB0	DL1	8	-401.39	-401.39	0.16	0.23	0.07	0.29	0.15	0.24
37	bB0	DL2	5	-323.73	-323.73	0.03	0.02	0.05	0.20	0.02	0.08
38	bB0	DL2	6	-324.56	-324.56	0.11	0.17	0.06	0.23	0.02	0.08
39	bB0	DL2	7	-324.67	-324.67	0.05	0.03	0.05	0.20	0.13	0.20
40	bB0	DL2	8	-325.50	-325.50	0.14	0.20	0.06	0.26	0.13	0.21

Then, further right, you will find results about the pressure acting on spring supports located along the pile, and the resistance of negative earth pressure, Kp.

Structural and Geotechnical Resistance for each Pile

72	y mm	% y %	Soil Pressure x-Dir. kPa	(Kp) Soil Dir. x kPa	P/(Kp) Dir. x %	Soil Pressure y-Dir. kPa	(Kp) Soil y-Dir. kPa	P/(Kp) Dir. y %	Nr Structure kN	Mrx Structure kN.m
33	0.09	0.35	0.18	66.67	0.27	0.13	72.54	0.18	969.45	14.11
34	0.09	0.35	0.07	79.89	0.08	0.13	72.54	0.18	969.45	14.11
35	0.09	0.37	0.33	61.97	0.53	0.09	79.89	0.11	969.45	14.11
36	0.09	0.37	0.08	79.89	0.10	0.09	79.89	0.11	969.45	14.11
37	0.08	0.31	0.13	66.67	0.19	0.11	72.54	0.15	969.45	14.11
38	0.08	0.31	0.06	79.89	0.07	0.11	72.54	0.15	969.45	14.11
39	0.08	0.32	0.22	66.67	0.34	0.08	79.89	0.10	969.45	14.11
40	0.08	0.32	0.07	79.89	0.09	0.08	79.89	0.10	969.45	14.11

Finally, you will find structural and geotechnical capacity of each pile.

Structural and Geotechnical Resistance for each Pile

72	Vry Structure kN	Mry Structure kN.m	Vrx Structure kN	Design Load Structural %	Cr Geotech. kN	Tr Geotech. kN	Geotech. Design Load Compression %	Geotech. Design Load Tension %
33	192.94	14.11	192.94	41.49	1230.25	32.39	32.46	0.00
34	192.94	14.11	192.94	42.21	1230.25	32.39	32.54	0.00
35	192.94	14.11	192.94	42.47	1230.25	32.39	32.55	0.00
36	192.94	14.11	192.94	42.98	1230.25	32.39	32.63	0.00
37	192.94	14.11	192.94	33.63	1230.25	32.39	26.31	0.00
38	192.94	14.11	192.94	34.27	1230.25	32.39	26.38	0.00
39	192.94	14.11	192.94	34.45	1230.25	32.39	26.39	0.00
40	192.94	14.11	192.94	34.91	1230.25	32.39	26.46	0.00

E X A M P L E S 1 7 & 1 8

2D & 3D Moving Load Analysis

Basic Principles

General

The **Moving Load** analysis module computes internal forces, influence lines, and envelopes 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 moving loads, as per codes, or they can create their own.

Pre-Defined Trucks

VisualDesign includes a list of predefined trucks in the **Moving Load Definition** spreadsheet, which is accessible through **Common/Trucks**. These trucks are described in several standards (CAN/CSA-S6-88, S6-00 and AASHTO LRFD-04). Each one of these trucks represents a different moving load case according to the position of axles and dynamic load allowance (DLA) factor. Refer to clause 3.8.4.5 of CAN/CSA-S6-00 standard.

Data cannot be modified in this spreadsheet. However, new "customized" trucks can be added at the end of the spreadsheet. For a quicker editing, duplicate an existing line and modify duplicated parameters.

To know the weight and the number of axles that are considered for each truck (load case) listed in the spreadsheet, select the **Axles/Wheels** tab.

Trucks' Nomenclature

VisualDesign is using the following nomenclature to describe trucks listed in the **Trucks** spreadsheet:

Examples:

[3D] – CL1-625 – 0.25: 3D structural model using a CL-625 truck, Level: Normal and 1, and DLA factor of 0.25.

[2D] – CL123-625 – 0.30a: 2D structural model using a CL-625 truck, Level 1, 2 & 3, and DLA factor of 0.3. The letter "a" means that an axle has been removed. This case is different from case "b" and "c".

Definition of Mobile Axis

Three moving load axes are available to study forces and displacements induced by trucks that are moving on different axes.

Axes can be located along members, plates (one side), and floors (one side). The axis number is selected in respective dialog boxes.

Road Surface

For a 3D model, floors or plates must be included in a structural model, creating a road surface where moving loads can be applied. Floors must not be modeled as rigid diaphragms.

Load Cases

It is recommended to include all required moving load cases that correspond to a given truck (Ex. CL-W). Each one has a different DLA factor so that all cases can be covered in the analysis (as required per clause 3.8.4.5 of CAN/CSA-S6-00 standard).

Moving load cases are entered in the **Moving Load Cases** spreadsheet, which is accessible through **Loads / Moving Load Cases/ Definition**.

Moving Load Cases Generator

A generator is available, which quickly generates all moving load cases, according to a chosen code or standard, and according to the user parameters. This generator is accessible through **Loads / Moving Load Cases / Generation Wizard**.

Axle/Wheel Factors for a 2D Model

2D axle factors must be defined in the **2D Axle Factors** spreadsheet, accessible through the **Loads** menu. These factors will be applied to forces that are transmitted to spans and supports.

Use the tables included in section 5.7.1 (Code S6-00) 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.

Moving Load Envelopes

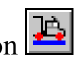
Moving load envelopes must be activated through the **Moving Load Envelopes** spreadsheet before launching a moving load analysis. This spreadsheet is located in the **Loads** menu.

A maximum of 10 envelopes can be generated in a single analysis. Concomitant values (Mx, Vy, Nz, etc.) can be obtained for critical forces, for each envelope. These concomitant values must be selected in the **Moving Load Analysis** dialog box.

Generation of Load Combinations

Use the **Load Combination Generation Wizard** to generate the required load combinations as per selected building code. Activate the option that includes envelopes *Lmi* in the generation.

Moving Load Analysis

Open the **Moving Load Analysis** dialog box by clicking this icon  or go to **Analysis** menu.

In the displayed dialog box, activate options such as envelopes, concomitant values, critical scenario (load case), etc.

Results

Results are in the form of envelopes (Lmi), which can be included in load combinations. Individual envelope results can be read through the Envelope activation mode. Use the **View Options** for graphic results and go to **Results / Envelope** to obtain numerical results.

Design

When a design is launched, the moving load analysis is automatically launched at each cycle of design because it is part of the design iterative process.

Before launching a design, open the **Moving Load Analysis** dialog box, activate some options and close it. Then, launch the design process.

3D Moving Load Analysis

Definition of the Project

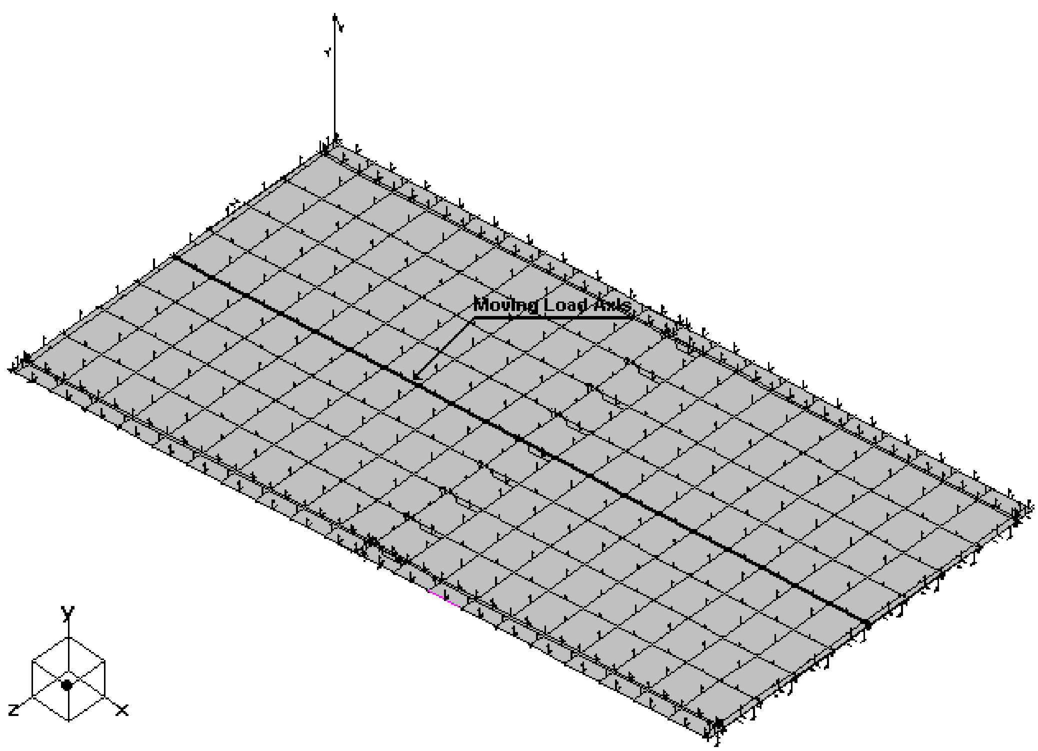
Length of bridge: +/- 50,000 m
Maximum width: 25,200 m
Number of spans: (2) spans
Length of spans: +/- 23,500 m

Bridge Deck:
Semi-continuous /Prefab Prestressed Concrete Beams NEBT 1200 and slab 220 mm with transverse post-tensioning

Moving Loads:
1 moving load axis and 12 moving load cases

Standard: CAN/CSA-S6-00.

The 3D model is as follows:

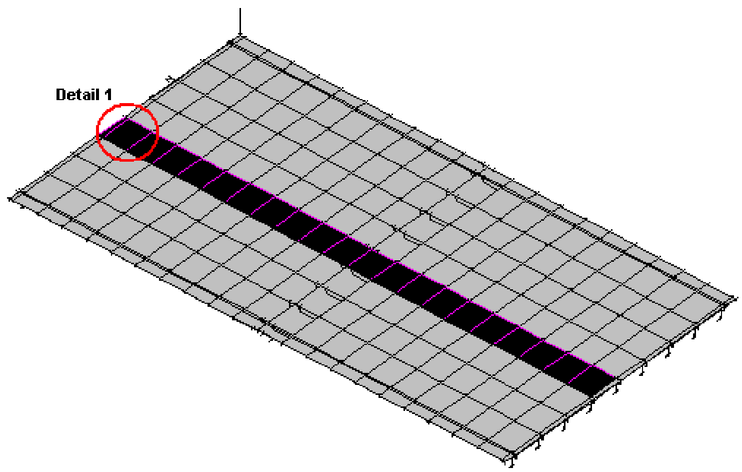


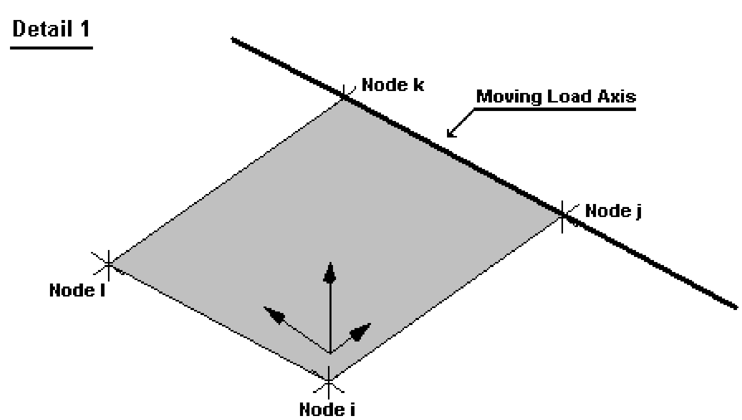
Definition of Moving Load Axis

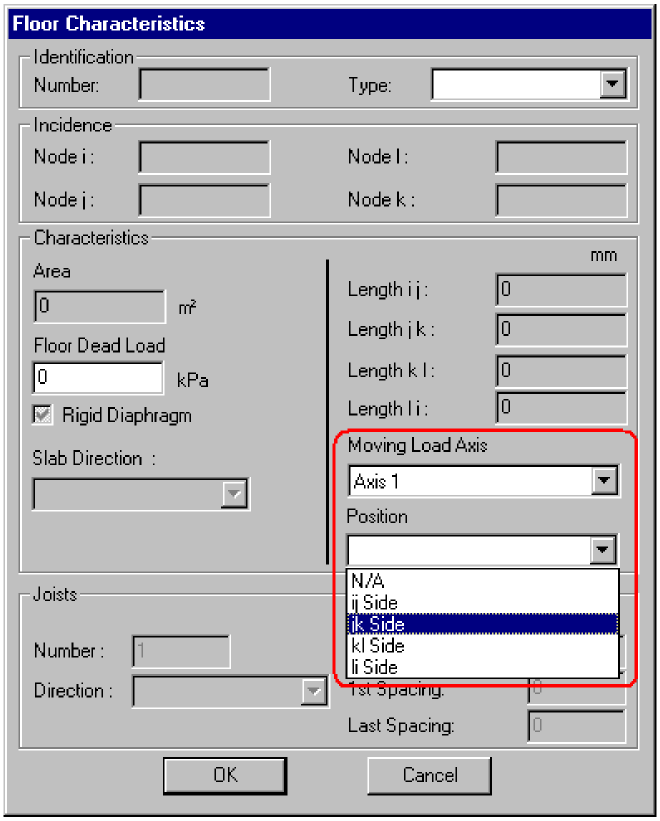
Floors

The moving load axis is located along the sides of floors that are located in the centre of bridge.

- Activate the Structure mode and the Floor icon.
- Select the strip of floors located at the center.
- Click the **Properties** icon to open the **Floor Characteristics** dialog box. Select the moving load axis number and specify the location of this axis through the side of floors. In our case, side jk. (To know on which side is passing the axis, display the floor local axis system through the **View Options**, as shown below.)







- Press OK to close the dialog box and save data.

Trucks and Load Cases

The CL-625 truck will be used for the design of this bridge (CAN/CSA-S6-00).

If you want to consult the list of pre-defined trucks, go to **Common** / **Trucks**. Sort data to group the [3D]-CL-625 load cases.

Definition of Moving Loads

Trucks | Axles/Wheels

64	Number	Total W kN	DLA (Truck)	DLA (Truck/Lane)	α %	Overload kN/m	DLA (W Lane)	Lane Width m	Remove Axle
32	[3D]-CL1-625-25	625.00	0.25	0.00	80.00	9.00	0.00	3.00	[x]
33	[3D]-CL1-625-30b	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[ ]
34	[3D]-CL1-625ON-25	625.00	0.25	0.00	80.00	9.00	0.00	3.00	[x]
35	[3D]-CL1-625ON-30b	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[ ]
36	[3D]-CL12-625-30c	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[ ]
37	[3D]-CL12-625-40	625.00	0.40	0.00	80.00	9.00	0.00	3.00	[ ]
38	[3D]-CL12-625ON-30c	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[ ]
39	[3D]-CL12-625ON-40	625.00	0.40	0.00	80.00	9.00	0.00	3.00	[ ]
40	[3D]-CL123-625-30a	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[x]
41	[3D]-CL123-625ON-30a	625.00	0.30	0.00	80.00	9.00	0.00	3.00	[x]
42	[3D]-CL2-625-25	625.00	0.25	0.00	80.00	9.00	0.00	3.00	[x]
43	[3D]-CL2-625ON-25	625.00	0.25	0.00	80.00	9.00	0.00	3.00	[x]
44	[3D]-CL3-625-40	625.00	0.40	0.00	80.00	9.00	0.00	3.00	[ ]
45	[3D]-CL3-625ON-40	625.00	0.40	0.00	80.00	9.00	0.00	3.00	[ ]

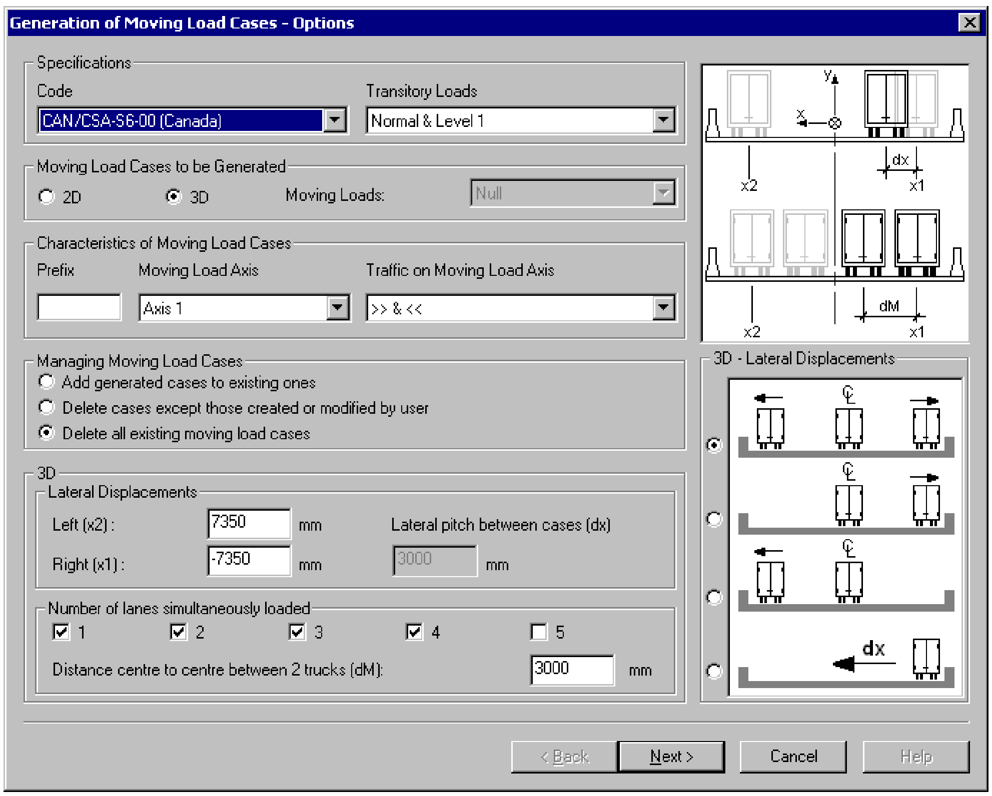
Moving Load Cases

Moving Load Case Generation Wizard

Moving load cases must be defined in the **Moving Load Cases** spreadsheet, which is accessible through **Loads / Moving Load Cases / Definition**. To quickly define them, we will use the **Generation Wizard**, available in the same submenu.

- In the **Options** page of the generator, select code CAN/CSA-S6-00 in the list box. The type of transitory loads will be *Normal and Level 1*.

- Just below the code, activate the "3D" radio button. Select moving load axis 1. The traffic on this axis is two-ways.



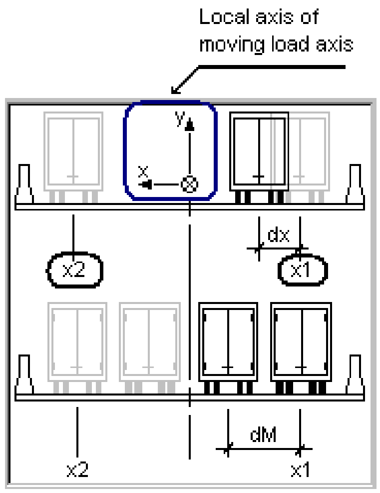
Lateral displacements of trucks must be specified for 3D project only (at the far left and right of moving load axis), according to the chosen layout.

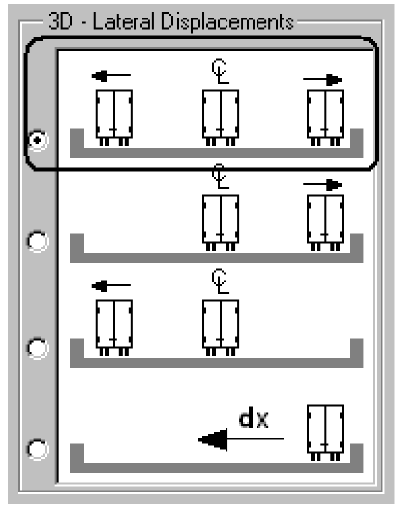
- Go to the *3D* section. Look at the four available layouts. Activate the first one, which will generate displacements of trucks at the left and right of moving load axis.

- Then, enter the distance at the far left (x2) and far right (x1). VisualDesign will generate moving loads between these two ends.

Sign Convention

According to local axis of moving load axis, x2 will be positive and x1, negative.



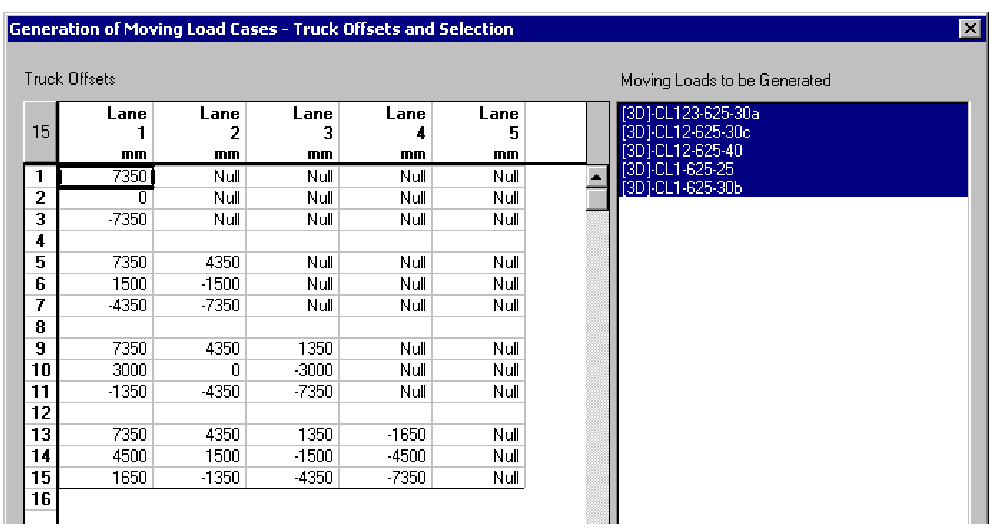


Four lanes can be loaded simultaneously. By default, the distance centre-to-centre of trucks (dM) is 3m. We keep this value.

- Activate boxes #1, #2, #3, and #4 at the section *Number of lanes simultaneously loaded.*

- Press the *Next* button.

The second page of generator shows the moving loads that are to be generated (at the right) according to the offsets listed in the left part of the dialog box. Moving loads are all selected by default, meaning that each one will be generated using the listed offsets. To withdraw a moving load, click on the name to cancel the selection.



Generation of Moving Load Cases - Truck Offsets and Selection

Truck Offsets

15	Lane 1 mm	Lane 2 mm	Lane 3 mm	Lane 4 mm	Lane 5 mm
1	7350	Null	Null	Null	Null
2	0	Null	Null	Null	Null
3	-7350	Null	Null	Null	Null
4					
5	7350	4350	Null	Null	Null
6	1500	-1500	Null	Null	Null
7	-4350	-7350	Null	Null	Null
8					
9	7350	4350	1350	Null	Null
10	3000	0	-3000	Null	Null
11	-1350	-4350	-7350	Null	Null
12					
13	7350	4350	1350	-1650	Null
14	4500	1500	-1500	-4500	Null
15	1650	-1350	-4350	-7350	Null
16					

Moving Loads to be Generated

[3D]-CL123-625-30a
[3D]-CL12-625-30c
[3D]-CL12-625-40
[3D]-CL1-625-25
[3D]-CL1-625-30b

VisualDesign is planning the generation of five trucks (listed at the right) using the 12 offsets (listed at the left), which correspond to 1, 2, 3, and 4 loaded lane2. Consequently, a minimum of 60 moving load cases will be generated. Fatigue moving load cases, which are not considering the lane overload, will be added to these by VisualDesign.

- Press the *Finish* button. The **Moving Load Cases** spreadsheet appears on screen.

We can see that 15 moving load cases were generated for *Fatigue* (without lane overload). The corresponding moving load envelope is named Lm02.

Moving Load Cases

Cases | Moving Load Cases Components

75	Number	Truck	Envelope	Moving Load Axis	Traffic on axis	DLA (Truck)	DLA (Truck/Lane)	Imbalance Factor	Add Overload
1	1F_1x1+000	[3D]-CL123-625-30a	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
2	1F_1x1+073	[3D]-CL123-625-30a	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
3	1F_1x1-073	[3D]-CL123-625-30a	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
4	1F_2x1+000	[3D]-CL12-625-30c	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
5	1F_2x1+073	[3D]-CL12-625-30c	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
6	1F_2x1-073	[3D]-CL12-625-30c	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
7	1F_3x1+000	[3D]-CL12-625-40	Truck : Lm02	Axis 1	>> & <<	0.40	0.00	0.00	[ ]
8	1F_3x1+073	[3D]-CL12-625-40	Truck : Lm02	Axis 1	>> & <<	0.40	0.00	0.00	[ ]
9	1F_3x1-073	[3D]-CL12-625-40	Truck : Lm02	Axis 1	>> & <<	0.40	0.00	0.00	[ ]
10	1F_4x1+000	[3D]-CL1-625-25	Truck : Lm02	Axis 1	>> & <<	0.25	0.00	0.00	[ ]
11	1F_4x1+073	[3D]-CL1-625-25	Truck : Lm02	Axis 1	>> & <<	0.25	0.00	0.00	[ ]
12	1F_4x1-073	[3D]-CL1-625-25	Truck : Lm02	Axis 1	>> & <<	0.25	0.00	0.00	[ ]
13	1F_5x1+000	[3D]-CL1-625-30b	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
14	1F_5x1+073	[3D]-CL1-625-30b	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
15	1F_5x1-073	[3D]-CL1-625-30b	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]

Below the list of fatigue moving load cases, we find 60 *Resistance* moving load cases, which are used for ultimate limits states design. The corresponding moving load envelope is named Lm01.

Moving Load Cases

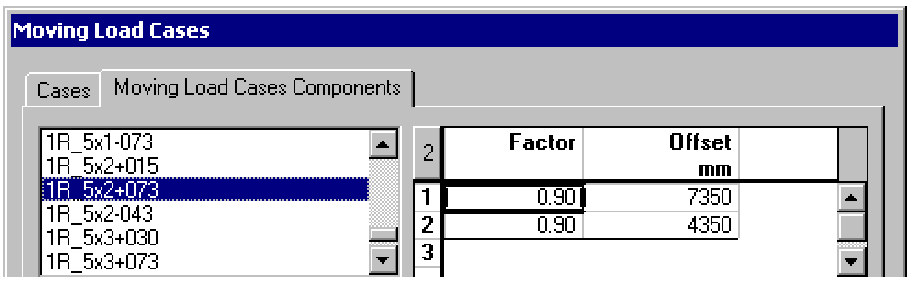
Cases | Moving Load Cases Components

75	Number	Truck	Envelope	Moving Load Axis	Traffic on axis	DLA (Truck)	DLA (Truck/Lane)	Imbalance Factor	Add Overload
16	1R_1x1+000	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
17	1R_1x1+073	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
18	1R_1x1-073	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
19	1R_1x2+015	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
20	1R_1x2+073	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
21	1R_1x2-043	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
22	1R_1x3+030	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
23	1R_1x3+073	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
24	1R_1x3-013	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
25	1R_1x4+017	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
26	1R_1x4+045	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
27	1R_1x4+073	[3D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
28	1R_2x1+000	[3D]-CL12-625-30c	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
29	1R_2x1+073	[3D]-CL12-625-30c	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
30	1R_2x1-073	[3D]-CL12-625-30c	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]

The nomenclature of generated numbers is as follows:

1F: Fatigue moving load case applied to moving load axis #1;
1R: Resistance moving load case applied to moving load axis #1;
1x1+043: First moving load applied when only one lane is loaded, with a 4.3m offset;
5x2+073: Fifth moving load applied when two lanes are loaded, with a 7.3m offset. (To know the second offset, go to the **Components** tab.)

- Select the line that corresponds to the moving load case 1R_5x2+073. In the spreadsheet, we can see that the second offset is equal to 4350mm.

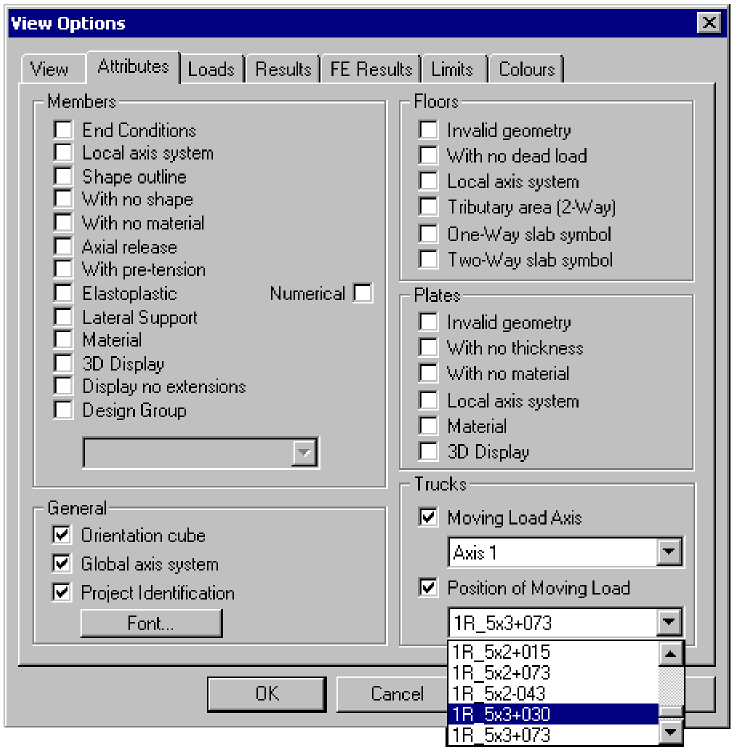


N. B. The "Factor" column: This factor is applied to concentrated loads and lane overload. It is used as *Modification Factor* when a 3D model, when many lanes are loaded.

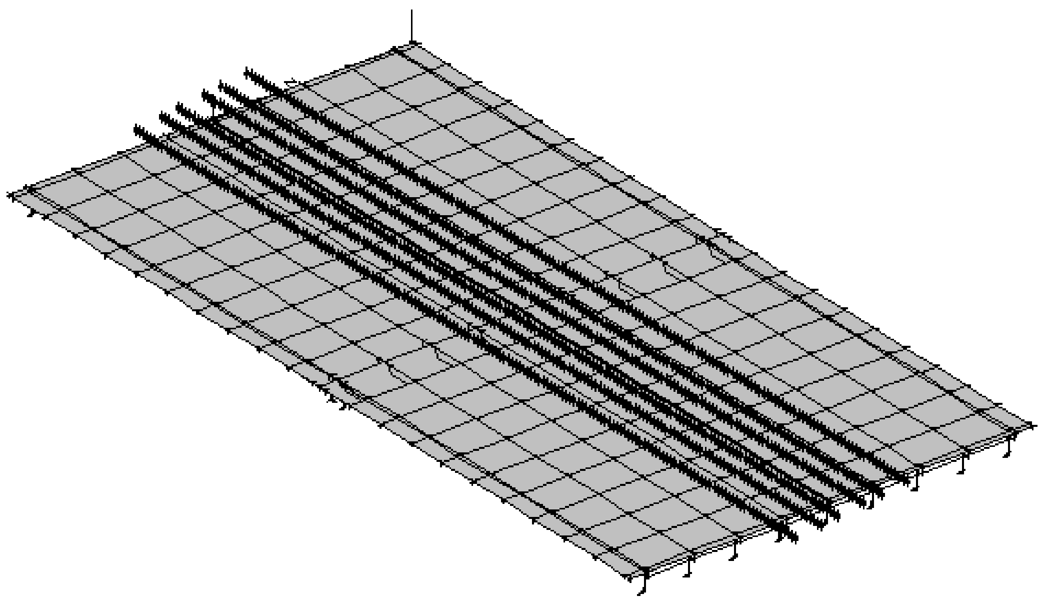
View Options

Display Moving Load axis and Moving Load Cases

Open the **View Options** dialog box and select the **Attributes** tab. Choose the moving load axis number and select a moving load case in the drop-down list box.



Moving Load Case #1R_5x3+030

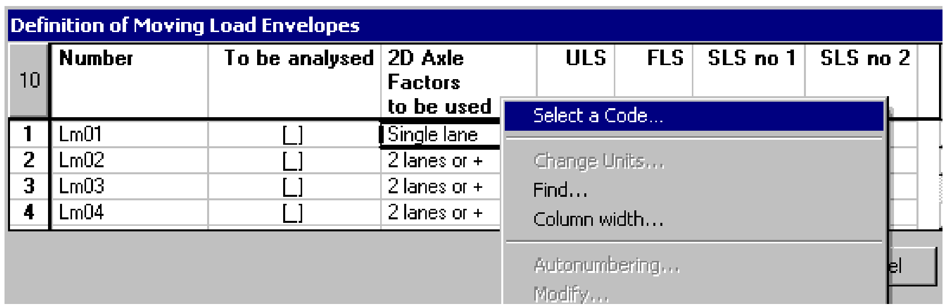


Definition of Moving Load Envelopes

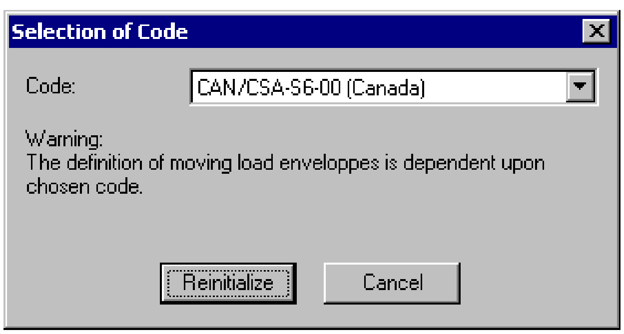
Moving load envelopes must be activated in this spreadsheet before launching a moving load analysis. When a code is selected in this spreadsheet, you obtain the required load combinations, and corresponding envelopes.

- Go to **Loads / Moving Load Envelopes**.

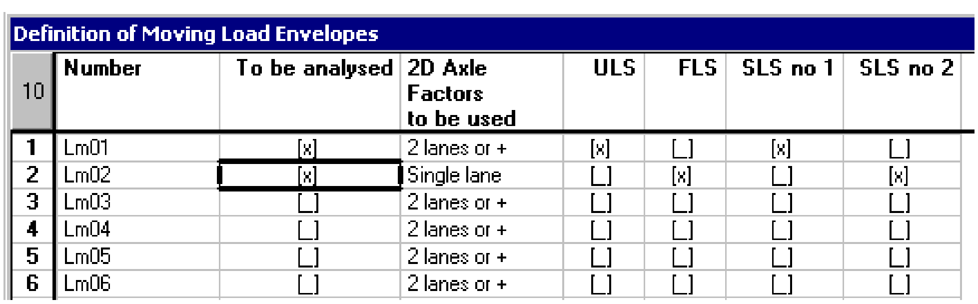
- Click in any cell and right click to open contextual menu. Choose the command **Select a code**.



- In the **Selection** dialog box, select code S6-00 and click the "Reinitialize" button.



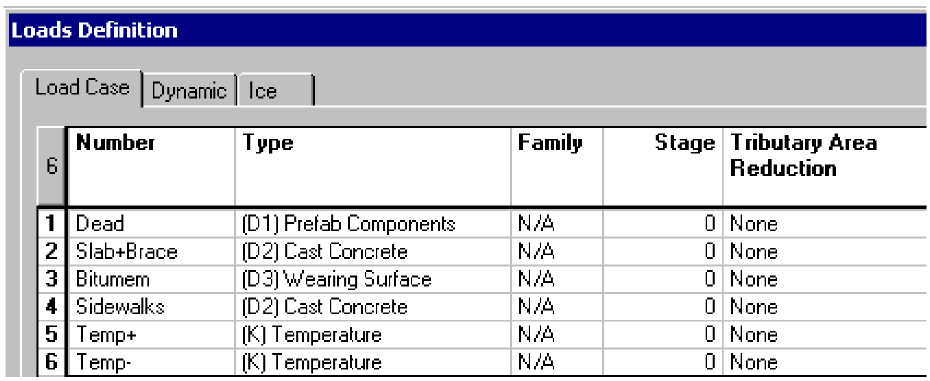
- Activate (double-click) moving load envelopes Lm01 and Lm02.



Load Cases

Loads Definition Spreadsheet

- Go to **Loads / Load Cases / Definition** and create load case titles and types according to code S6-00.


Loads Definition

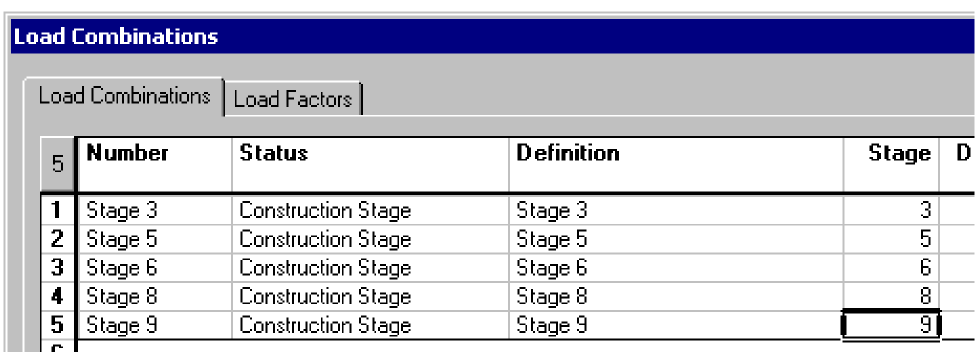
Load Case | Dynamic | Ice

6	Number	Type	Family	Stage	Tributary Area Reduction
1	Dead	(D1) Prefab Components	N/A	0	None
2	Slab+Brace	(D2) Cast Concrete	N/A	0	None
3	Bitumem	(D3) Wearing Surface	N/A	0	None
4	Sidewalks	(D2) Cast Concrete	N/A	0	None
5	Temp+	(K) Temperature	N/A	0	None
6	Temp-	(K) Temperature	N/A	0	None

Load Combinations

Construction Stages

- Go to **Loads / Load Combinations / Definition** and create construction stage load combinations according to the activated stages in the **Prestressing** tab (**Project Configuration**).


Load Combinations

Load Combinations | Load Factors

5	Number	Status	Definition	Stage	D
1	Stage 3	Construction Stage	Stage 3	3	
2	Stage 5	Construction Stage	Stage 5	5	
3	Stage 6	Construction Stage	Stage 6	6	
4	Stage 8	Construction Stage	Stage 8	8	
5	Stage 9	Construction Stage	Stage 9	9	

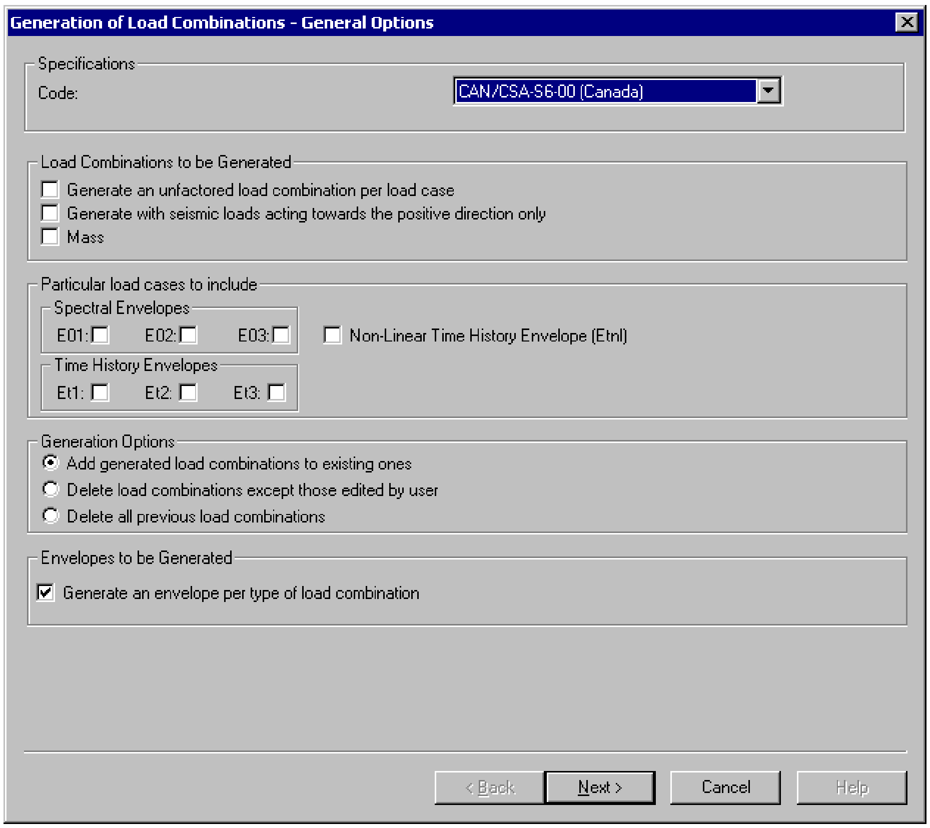
- Select the **Load Factors** tab. Select appropriate load cases and enter load factors.

- Close the spreadsheet.

Load Combination Generation Wizard

- Go to **Loads** / **Load Combinations** / **Generation Wizard**.

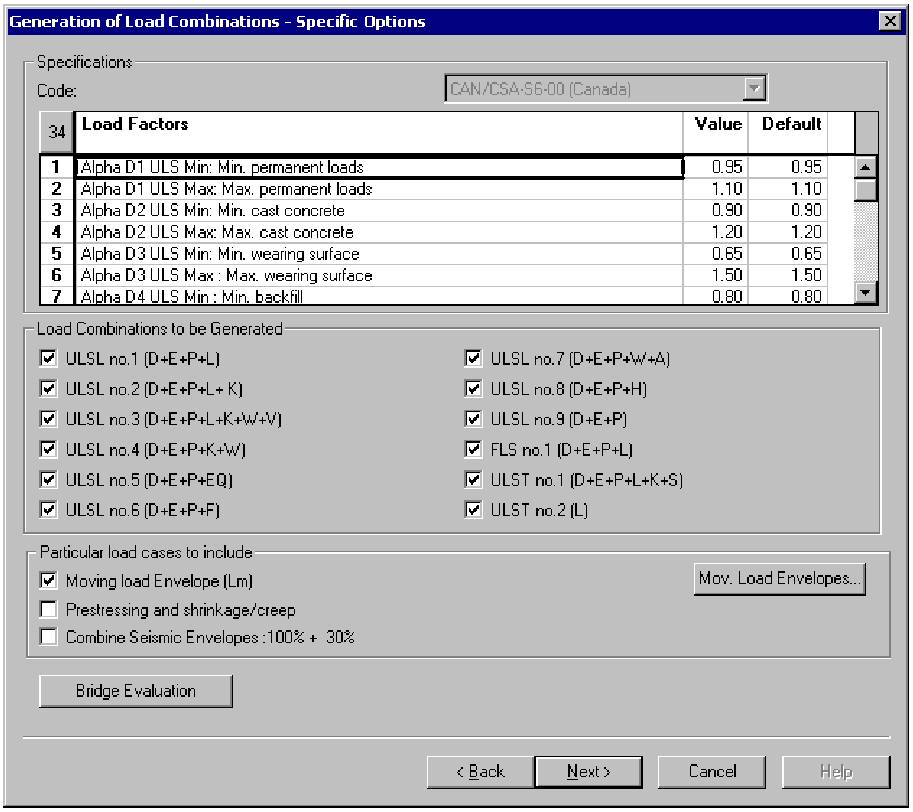
- In the **General Options** page, select code S6-00 among the drop-down list box. Activate option *Add generated load combinations to existing ones* to avoid the deletion of construction stages. Generate standard envelopes.



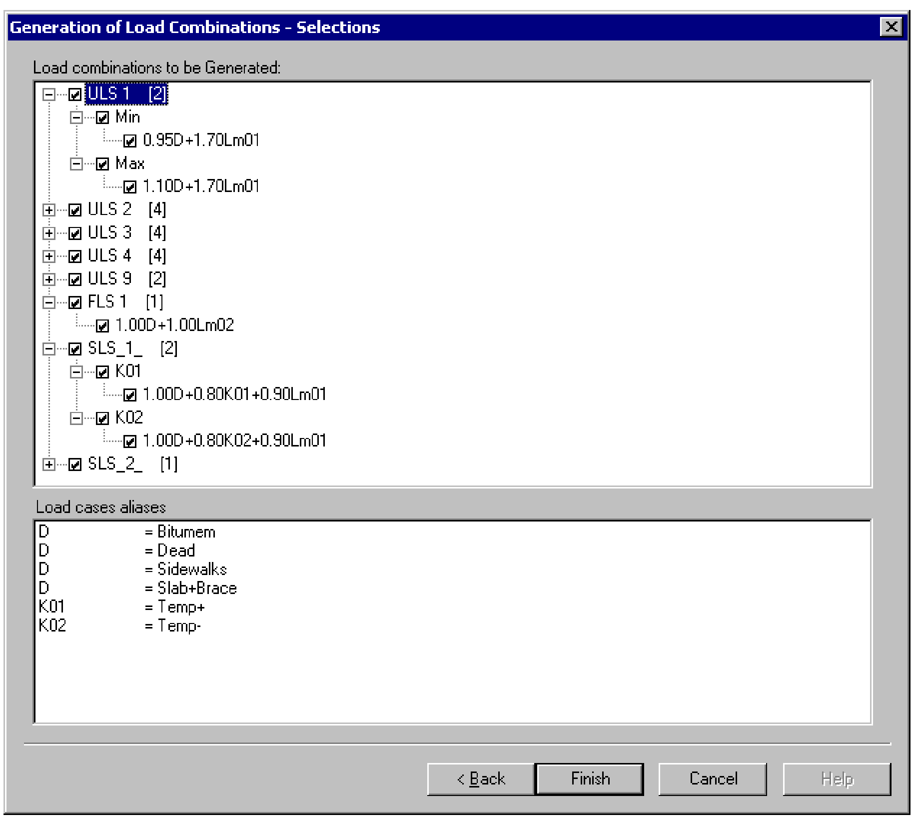
- Click *Next*.

- Activate the insertion of moving load envelopes in the **Specific Options** page of the generator.

N. B. The "Mov. Load Envelopes" button opens the **Moving Load Envelopes** spreadsheet. We have already activated envelopes (page 297).


Generation of Load Combinations - Specific Options
Specifications
Code:
CAN/CSA-S6-00 (Canada)
34
Load Factors
Value
Default
1 Alpha D1 ULS Min: Min. permanent loads 0.95 0.95
2 Alpha D1 ULS Max: Max. permanent loads 1.10 1.10
3 Alpha D2 ULS Min: Min. cast concrete 0.90 0.90
4 Alpha D2 ULS Max: Max. cast concrete 1.20 1.20
5 Alpha D3 ULS Min: Min. wearing surface 0.65 0.65
6 Alpha D3 ULS Max : Max. wearing surface 1.50 1.50
7 Alpha D4 ULS Min : Min. backfill 0.80 0.80
Load Combinations to be Generated
ULSL no.1 (D+E+P+L)
ULSL no.2 (D+E+P+L+ K)
ULSL no.3 (D+E+P+L+K+W+V)
ULSL no.4 (D+E+P+K+W)
ULSL no.5 (D+E+P+EQ)
ULSL no.6 (D+E+P+F)
ULSL no.7 (D+E+P+W+A)
ULSL no.8 (D+E+P+H)
ULSL no.9 (D+E+P)
FLS no.1 (D+E+P+L)
ULST no.1 (D+E+P+L+K+S)
ULST no.2 (L)
Particular load cases to include
Moving load Envelope (Lm)
Prestressing and shrinkage/creep
Combine Seismic Envelopes :100% + 30%
Mov. Load Envelopes...
Bridge Evaluation
< Back
Next >
Cancel
Help

- Click *Next*.


Generation of Load Combinations - Selections
Load combinations to be Generated:
ULS 1 [2]
Min
0.95D+1.70Lm01
Max
1.10D+1.70Lm01
ULS 2 [4]
ULS 3 [4]
ULS 4 [4]
ULS 9 [2]
FLS 1 [1]
1.00D+1.00Lm02
SLS_1_ [2]
K01
1.00D+0.80K01+0.90Lm01
K02
1.00D+0.80K02+0.90Lm01
SLS_2_ [1]
Load cases aliases
D = Bitumem
D = Dead
D = Sidewalks
D = Slab+Brace
K01 = Temp+
K02 = Temp-
< Back
Finish
Cancel
Help

- Press *Finish*.

The **Load Combination** spreadsheet will be displayed on screen.

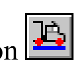
Load Combinations

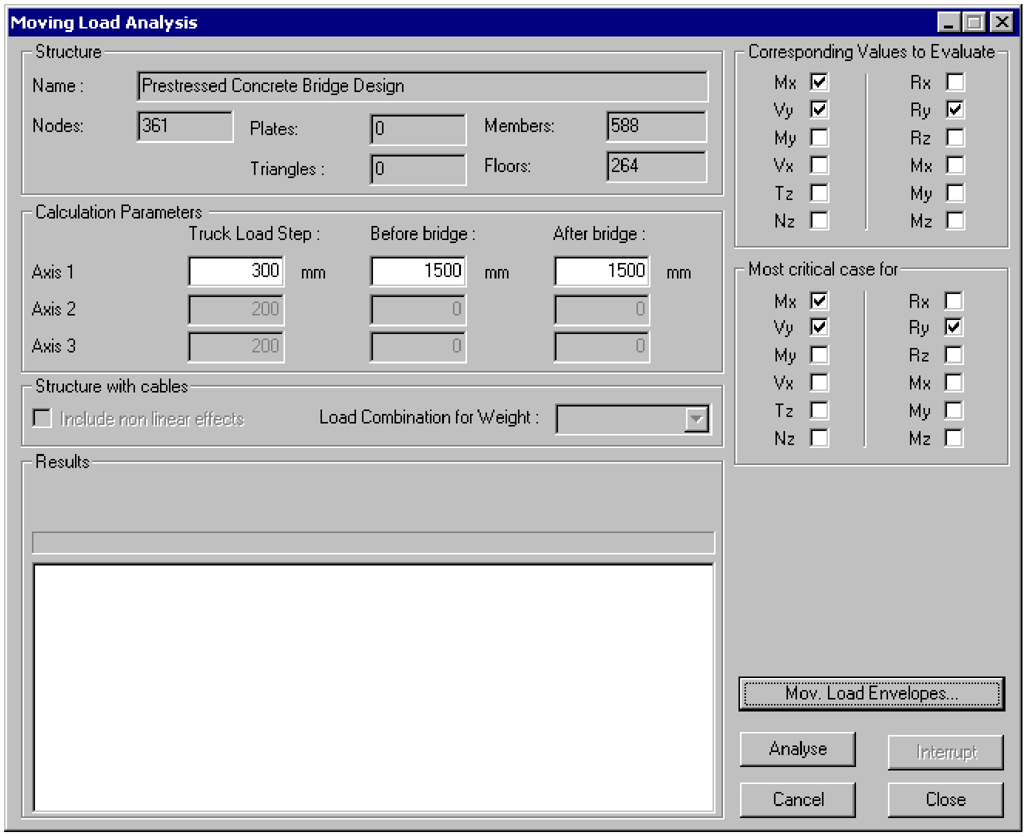
Load Combinations | Load Factors

25	Number	Status	Definition	Stage
1	Stage 3	Construction Stage	Stage 3	3
2	Stage 5	Construction Stage	Stage 5	5
3	Stage 6	Construction Stage	Stage 6	6
4	Stage 8	Construction Stage	Stage 8	8
5	Stage 9	Construction Stage	Stage 9	9
6	ULS 1:max02	ULS 1	1.10D+1.70Lm01	0
7	ULS 1:min01	ULS 1	0.95D+1.70Lm01	0
8	ULS 2:max05	ULS 2	1.10D+1.15K01+1.60Lm01	0
9	ULS 2:max06	ULS 2	1.10D+1.15K02+1.60Lm01	0
10	ULS 2:min03	ULS 2	0.95D+1.15K01+1.60Lm01	0
11	ULS 2:min04	ULS 2	0.95D+1.15K02+1.60Lm01	0
12	ULS 3:max09	ULS 3	1.10D+1.00K01+1.40Lm01	0
13	ULS 3:max10	ULS 3	1.10D+1.00K02+1.40Lm01	0
14	ULS 3:min07	ULS 3	0.95D+1.00K01+1.40Lm01	0
15	ULS 3:min08	ULS 3	0.95D+1.00K02+1.40Lm01	0
16	ULS 4:max13	ULS 4	1.10D+1.25K01	0
17	ULS 4:max14	ULS 4	1.10D+1.25K02	0
18	ULS 4:min11	ULS 4	0.95D+1.25K01	0
19	ULS 4:min12	ULS 4	0.95D+1.25K02	0
20	ULS 9:max16	ULS 9	1.35D	0
21	ULS 9:min15	ULS 9	1.35D	0
22	FLS 117	FLS 1	1.00D+1.00Lm02	0
23	SLS_1_18	SLS 1	1.00D+0.80K01+0.90Lm01	0
24	SLS_1_19	SLS 1	1.00D+0.80K02+0.90Lm01	0
25	SLS_2_20	SLS 2	0.90Lm02	0

- Close the spreadsheet.

Moving Load Analysis

- Open the **Moving Load Analysis** dialog box by pressing icon  on Tools toolbar and select analysis options.



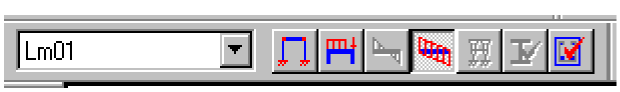
- Click the *Analyse* button to launch the analysis. Close the dialog box when analysis is completed.

Results – Moving Load Envelopes

Graphic Results

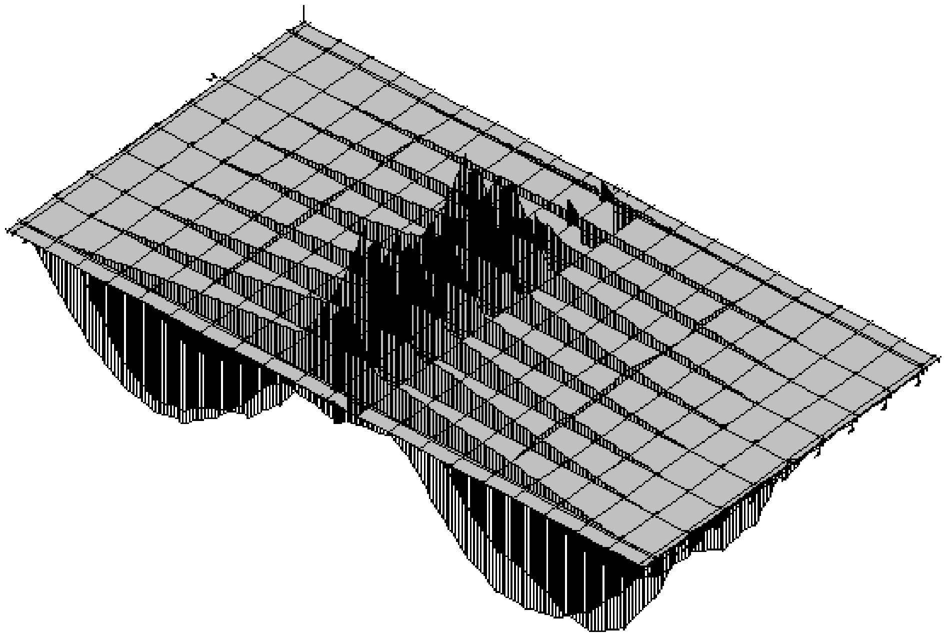
The Envelope mode is automatically activated when analysis is completed.

- Select envelope *Lm01* or *Lm02*.

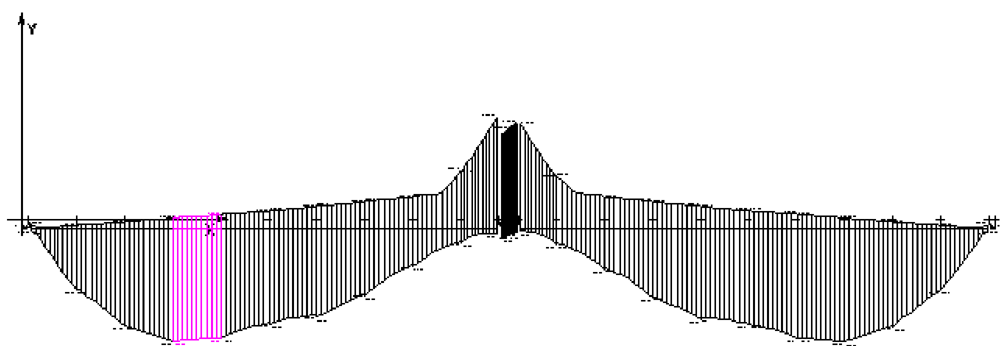


- Open the **View Options** dialog and go to the **Results** tab. Select a type of graph.

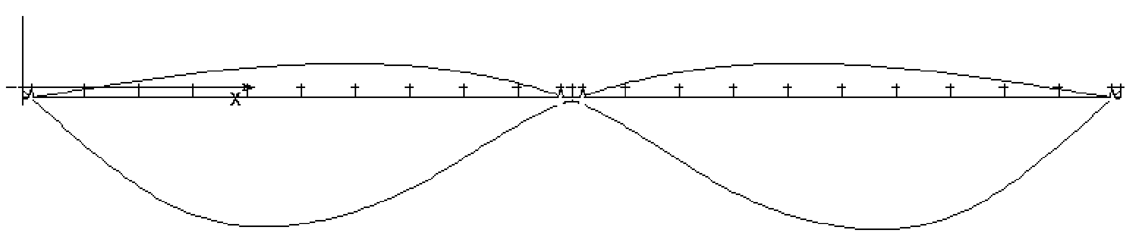
The image below is showing bending moments on strong axis (Mx), for moving load envelope *Lm01*.



- Select a bridge axis and use the **Mask** function.



- Go back to the **Results** tab of **View Options** dialog box and activate the **Deflection** diagram.



Member Forces and Concomitant Values

Double click on a member to open this spreadsheet. Minimum and maximum forces are listed and concomitant values also (if they were activated in the **Moving Load Analysis** dialog box).

Internal Forces and Deflections Spreadsheet (Lm01)

66	Number	Shape	Position mm	Value	Bending Mx kN.m	Mx Case	Shear Vy kN	Vy Case	Bending My kN.m	Shear Vx kN	Axial Nz kN	Torsion Tz kN.m	Strong axis v mm
1	m223	NEBT 1200	0	MxMax	859.65		61.14		104.30	68.69	-25.54	-0.59	-5.58
2	m223	NEBT 1200	236	MxMax	845.69		24.73		82.62	64.21	-25.14	-1.64	-5.56
3	m223	NEBT 1200	472	MxMax	837.72		54.72		91.49	56.37	-12.51	-3.01	-5.64
4	m223	NEBT 1200	709	MxMax	833.80		57.76		76.28	52.70	-12.07	-3.79	-5.61
5	m223	NEBT 1200	945	MxMax	829.25		11.61		65.87	53.44	-11.60	-3.63	-5.56
6	m223	NEBT 1200	1181	MxMax	818.87		13.01		55.31	56.65	-11.08	-2.97	-5.51
7	m223	NEBT 1200	1417	MxMax	808.70		63.66		41.93	56.65	-11.08	-2.97	-5.51
8	m223	NEBT 1200	1653	MxMax	800.60		61.37		29.87	50.84	-12.56	-4.04	-5.45
9	m223	NEBT 1200	1889	MxMax	795.56		56.45		21.00	45.15	-13.03	-5.27	-5.41
10	m223	NEBT 1200	2126	MxMax	793.06		-22.32		-2.84	45.00	-14.49	-6.21	-5.40
11	m223	NEBT 1200	2362	MxMax	796.61		-27.02		-9.32	42.57	-14.31	-6.98	-5.42
12	m223	NEBT 1200	0	MxMin	-159.12		17.46		-26.53	7.54	9.98	1.49	1.63
13	m223	NEBT 1200	236	MxMin	-163.24		17.46		-28.31	7.54	9.98	1.49	1.65
14	m223	NEBT 1200	472	MxMin	-167.37		17.46		-30.09	7.54	9.98	1.49	1.68
15	m223	NEBT 1200	709	MxMin	-171.49		17.46		-31.87	7.54	9.98	1.49	1.70
16	m223	NEBT 1200	945	MxMin	-175.61		17.46		-33.65	7.54	9.98	1.49	1.72
17	m223	NEBT 1200	1181	MxMin	-179.74		17.46		-35.43	7.54	9.98	1.49	1.74
18	m223	NEBT 1200	1417	MxMin	-183.86		17.46		-37.21	7.54	9.98	1.49	1.76
19	m223	NEBT 1200	1653	MxMin	-187.98		17.46		-38.99	7.54	9.98	1.49	1.77
20	m223	NEBT 1200	1889	MxMin	-192.11		17.46		-40.77	7.54	9.98	1.49	1.79
21	m223	NEBT 1200	2126	MxMin	-196.23		17.46		-42.55	7.54	9.98	1.49	1.80
22	m223	NEBT 1200	2362	MxMin	-200.35		17.46		-44.33	7.54	9.98	1.49	1.82

Internal Forces and Deflections Spreadsheet (Lm01)

66	Number	Shape	Position mm	Value	Bending Mx kN.m	Mx Case	Shear Vy kN	Vy Case	Bending My kN.m	Shear Vx kN	Axial Nz kN	Torsion Tz kN.m	Strong axis v mm
23	m223	NEBT 1200	0	VyMax	538.01		112.84		45.50	41.04	-17.06	3.75	-2.87
24	m223	NEBT 1200	236	VyMax	536.57		115.99		45.90	48.93	-16.42	5.05	-2.92
25	m223	NEBT 1200	472	VyMax	530.35		116.54		31.87	45.32	-16.57	3.83	-2.94
26	m223	NEBT 1200	709	VyMax	520.95		120.22		16.42	41.32	-21.21	3.03	-3.05
27	m223	NEBT 1200	945	VyMax	492.56		120.22		6.66	41.32	-21.21	3.03	-3.03
28	m223	NEBT 1200	1181	VyMax	493.80		125.40		-2.43	43.30	-21.70	3.27	-3.10
29	m223	NEBT 1200	1417	VyMax	491.08		127.62		-11.52	48.16	-21.25	3.98	-3.16
30	m223	NEBT 1200	1653	VyMax	460.94		127.62		-22.90	48.16	-21.25	3.98	-3.13
31	m223	NEBT 1200	1889	VyMax	430.80		127.62		-34.27	48.16	-21.25	3.98	-3.09
32	m223	NEBT 1200	2126	VyMax	400.65		127.62		-45.65	48.16	-21.25	3.98	-3.06
33	m223	NEBT 1200	2362	VyMax	370.51		127.62		-57.02	48.16	-21.25	3.98	-3.01
34	m223	NEBT 1200	0	VyMin	631.45		-66.70		66.12	35.75	-23.04	-6.85	-4.47
35	m223	NEBT 1200	236	VyMin	647.21		-66.70		57.68	35.75	-23.04	-6.85	-4.51
36	m223	NEBT 1200	472	VyMin	662.96		-66.70		49.24	35.75	-23.04	-6.85	-4.56
37	m223	NEBT 1200	709	VyMin	678.71		-66.70		40.79	35.75	-23.04	-6.85	-4.60
38	m223	NEBT 1200	945	VyMin	694.47		-66.70		32.35	35.75	-23.04	-6.85	-4.63
39	m223	NEBT 1200	1181	VyMin	680.15		-65.78		22.40	33.02	-23.25	-7.29	-4.56
40	m223	NEBT 1200	1417	VyMin	664.63		-63.19		14.96	30.87	-22.85	-7.38	-4.48
41	m223	NEBT 1200	1653	VyMin	627.00		-63.15		10.01	25.57	-22.50	-8.04	-4.30
42	m223	NEBT 1200	1889	VyMin	641.92		-63.15		3.98	25.57	-22.50	-8.04	-4.31
43	m223	NEBT 1200	2126	VyMin	656.83		-63.15		-2.06	25.57	-22.50	-8.04	-4.33
44	m223	NEBT 1200	2362	VyMin	671.63		-62.31		-8.10	25.57	-22.50	-8.04	-4.34

Internal Forces and Deflections Spreadsheet (Lm01)

66	Number	Shape	Position mm	Value	Bending Mx kN.m	Mx Case	Shear Vy kN	Vy Case	Bending My kN.m	Shear Vx kN	Axial Nz kN	Torsion Tz kN.m	Strong axis v mm
45	m223	NEBT 1200	0	Max	859.65	1R_4x2-043	112.84	1R_1x1-073	124.37	68.99	25.92	5.61	1.63
46	m223	NEBT 1200	236	Max	845.69	1R_4x2-043	115.99	1R_1x1-073	111.04	68.99	25.92	5.61	1.65
47	m223	NEBT 1200	472	Max	837.72	1R_4x3-013	116.54	1R_1x1-073	100.27	68.99	25.92	5.61	1.68
48	m223	NEBT 1200	709	Max	833.80	1R_4x3-013	120.22	1R_4x1-073	89.51	68.99	25.92	5.61	1.70
49	m223	NEBT 1200	945	Max	829.25	1R_4x3-013	120.22	1R_4x1-073	86.39	68.99	25.92	5.61	1.72
50	m223	NEBT 1200	1181	Max	818.87	1R_4x3-013	125.40	1R_4x1-073	85.11	68.99	25.92	5.61	1.74
51	m223	NEBT 1200	1417	Max	808.70	1R_4x3-013	127.62	1R_4x1-073	84.70	68.99	25.92	5.61	1.76
52	m223	NEBT 1200	1653	Max	800.60	1R_4x3-013	127.62	1R_4x1-073	86.24	68.99	25.92	5.61	1.77
53	m223	NEBT 1200	1889	Max	795.56	1R_4x3-013	127.62	1R_4x1-073	88.24	68.99	25.92	5.61	1.79
54	m223	NEBT 1200	2126	Max	793.06	1R_4x3-013	127.62	1R_4x1-073	90.24	68.99	25.92	5.61	1.80
55	m223	NEBT 1200	2362	Max	796.61	1R_4x3-013	127.62	1R_4x1-073	92.24	68.99	25.92	5.61	1.82
56	m223	NEBT 1200	0	Min	-159.12	1R_4x3-013	-66.70	1R_4x2-043	-38.71	-22.42	-36.92	-8.45	-5.71
57	m223	NEBT 1200	236	Min	-163.24	1R_4x3-013	-66.70	1R_4x2-043	-41.76	-22.42	-36.92	-8.45	-5.74
58	m223	NEBT 1200	472	Min	-167.37	1R_4x3-013	-66.70	1R_4x2-043	-44.81	-22.42	-36.92	-8.45	-5.76
59	m223	NEBT 1200	709	Min	-171.49	1R_4x3-013	-66.70	1R_4x2-043	-47.86	-22.42	-36.92	-8.45	-5.78
60	m223	NEBT 1200	945	Min	-175.61	1R_4x3-013	-66.70	1R_4x2-043	-50.92	-22.42	-36.92	-8.45	-5.79
61	m223	NEBT 1200	1181	Min	-179.74	1R_4x3-013	-65.78	1R_4x2-043	-53.97	-22.42	-36.92	-8.45	-5.80
62	m223	NEBT 1200	1417	Min	-183.86	1R_4x3-013	-63.19	1R_4x2-043	-57.02	-22.42	-36.92	-8.45	-5.80
63	m223	NEBT 1200	1653	Min	-187.98	1R_4x3-013	-63.15	1R_4x2-043	-60.07	-22.42	-36.92	-8.45	-5.79
64	m223	NEBT 1200	1889	Min	-192.11	1R_4x3-013	-63.15	1R_4x2-043	-63.12	-22.42	-36.92	-8.45	-5.78
65	m223	NEBT 1200	2126	Min	-196.23	1R_4x3-013	-63.15	1R_4x2-043	-66.17	-22.42	-36.92	-8.45	-5.76
66	m223	NEBT 1200	2362	Min	-200.35	1R_4x3-013	-62.31	1R_4x2-043	-69.23	-22.42	-36.92	-8.45	-5.74

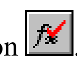
Support Reactions (min/max)

Double-click on a support to open this spreadsheet or select many supports and go to **Results / Envelope / Support Reactions (min / max).**

Support Reactions Spreadsheet (min/max)

89	Number	Value	Rx kN	Ry kN	Ry Case	Rz kN	Mx kN.m	My kN.m	Mz kN.m
1	n113%1	RyMax	0.00	59.05		40.46	0.00	0.00	0.00
2	n113%1	RyMin	0.00	-21.40		-10.66	0.00	0.00	0.00
3									
4	n113%1	Max	0.00	59.05	1R_4x2+073	48.81	0.00	0.00	0.00
5	n113%1	Min	0.00	-21.40	1R_4x3+073	-15.38	0.00	0.00	0.00
6									
7	n114%1	RyMax	0.00	247.19		44.85	0.00	0.00	0.00
8	n114%1	RyMin	0.00	-14.30		-16.76	0.00	0.00	0.00
9									
10	n114%1	Max	0.00	247.19	1R_4x2+073	61.50	0.00	0.00	0.00
11	n114%1	Min	0.00	-14.30	1R_4x4+073	-22.39	0.00	0.00	0.00
12									
13	n115%1	RyMax	0.00	349.03		-16.33	0.00	0.00	0.00
14	n115%1	RyMin	0.00	-15.35		-7.78	0.00	0.00	0.00
15									
16	n115%1	Max	0.00	349.03	1R_4x2+073	60.89	0.00	0.00	0.00
17	n115%1	Min	0.00	-15.35	1R_4x4+073	-53.05	0.00	0.00	0.00
18									
19	n116%1	RyMax	0.00	330.33		-13.98	0.00	0.00	0.00
20	n116%1	RyMin	0.00	-15.16		3.13	0.00	0.00	0.00

Load Combinations and Design Results

Launch a bridge design by clicking the Design icon .

The moving load analysis will be part of the design iterative process. It will automatically be launched at each cycle of design (steel, concrete, or prestressed concrete design, etc.). Therefore, we suggest selecting options in the **Moving Load Analysis** dialog box before launching the design.

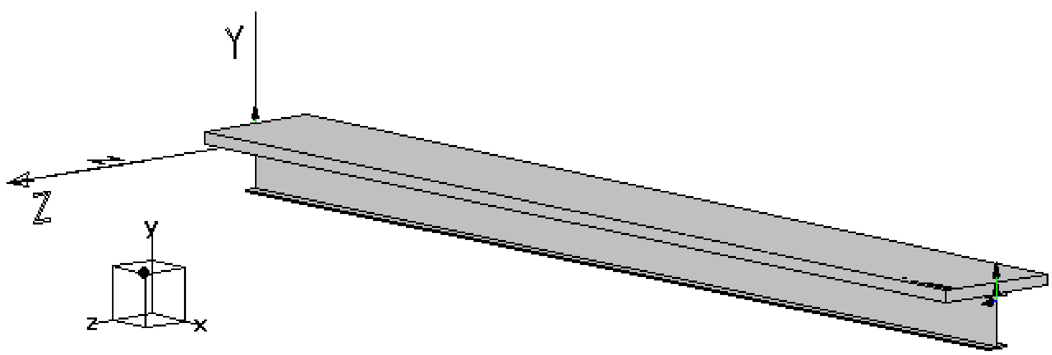
2D Moving Load Analysis

Definition of the Project

Composite beam of 20m long on a single span;
Stud: Neilson 22mm;
Concrete slab of 200mm.

Moving Loads: According to CAN/CSA-S6-00;
One moving load axis.

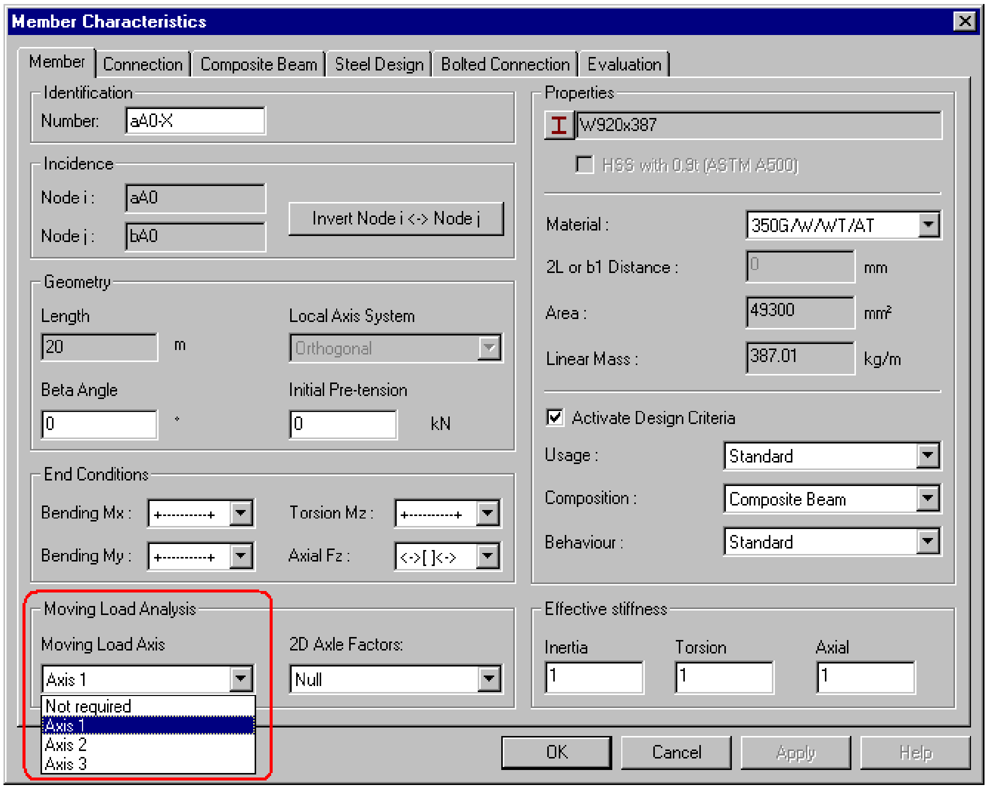
The 2D model is as follows:



Definition of Moving Load Axis

The moving load axis must be located on the beam (there is only one beam).

- Activate the Structure mode and double click on the beam.
- In the **Member Characteristics** dialog box, go to section "Moving Load Axis", click the arrow to open the drop-down list box and select *Axis 1*.



2D Axle Factors

Axles factors are required for a 2D project. These factors will be applied to calculated forces that are transmitted to supports and spans.

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.

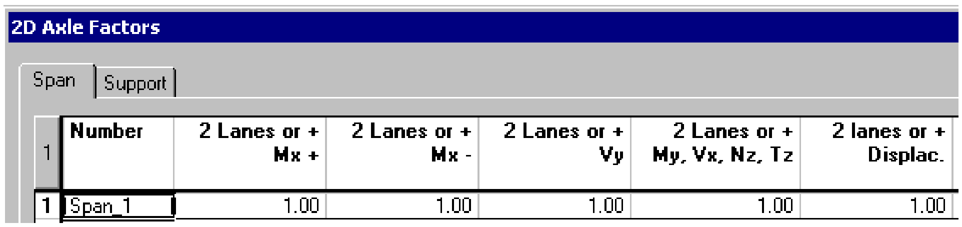
Mg = Fa * Mt

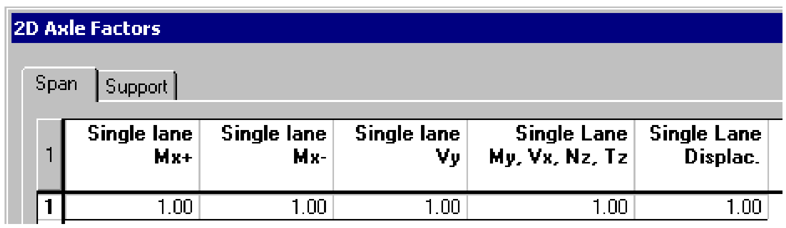
Where Mt is the maximum bending moment induced by one truck on a section of the studied span.

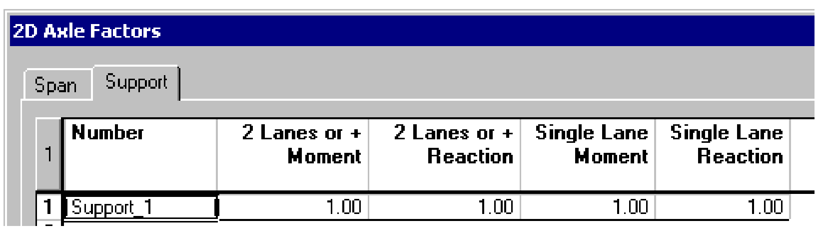
2D Axles Factors Spreadsheet

When axle factors are calculated:

- Open the **2D Axles Factors** spreadsheet (**Loads** menu)
- Insert a line and give a name to axles factors (spans and supports).
- Click in each cell and enter factors.



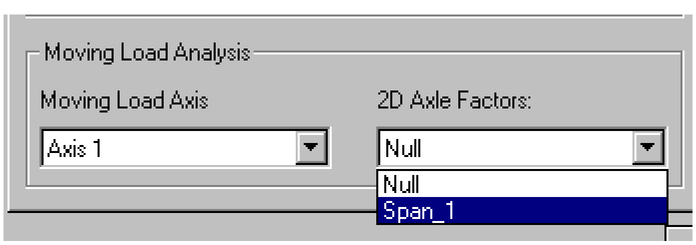




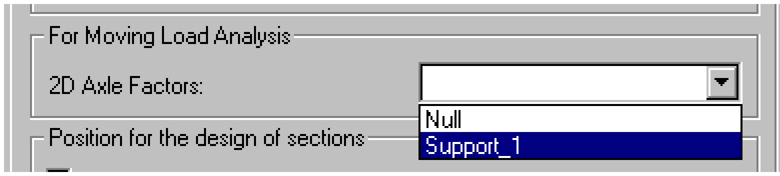
- Close the spreadsheet.

Assign 2D Axles Factors to Span and Supports

- Activate the Member element and double click on the beam. In the **Member Characteristics** dialog box, select the name *Span_1* as 2D axles factors.



- Activate the Support element and select both supports. Click the **Properties** icon. In the **Support** tab **of Node Characteristics** dialog box, select the name Support_1 as 2D axles factors.



Trucks

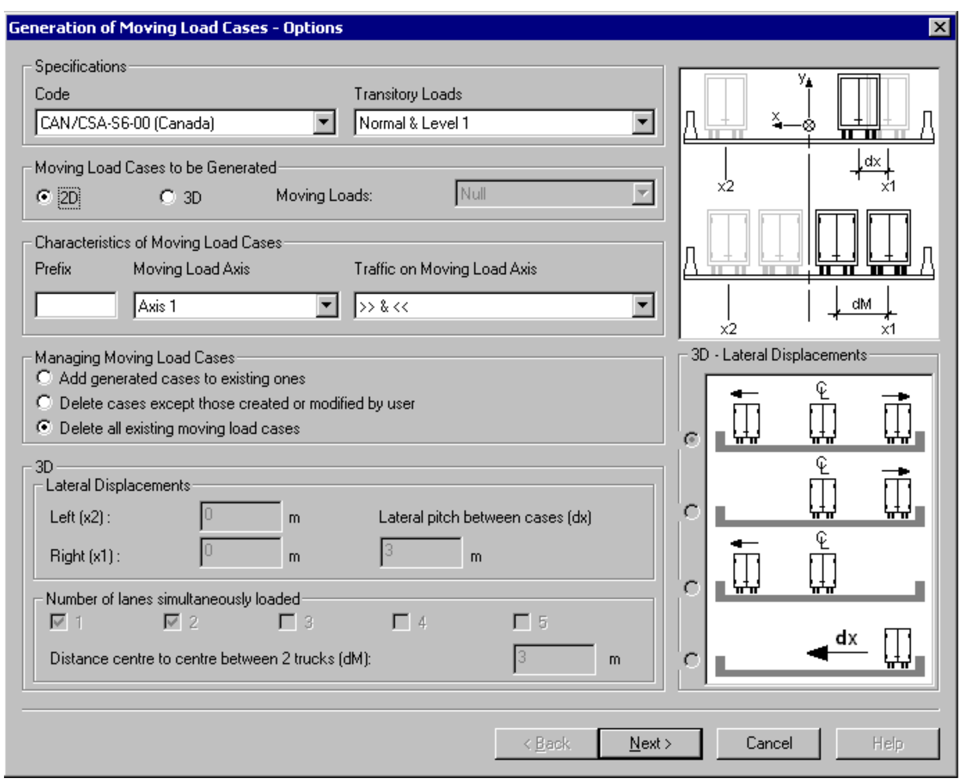
CL-625 trucks will be used, according to CAN/CSA-S6-00 code. To consult the list of available pre-defined [2D]-CL-625 trucks, go to menu **Common / Trucks**.

Moving Load Cases

Generation Wizard

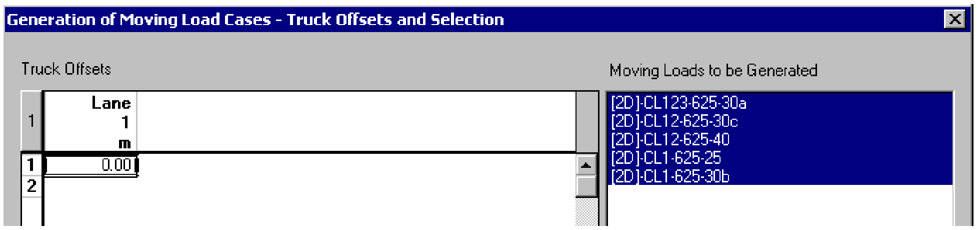
- Use the **Moving Load Case Generation Wizard**, located under **Loads / Moving Load Cases** / **Generation Wizard** to quickly generate required cases according to code S6-00.

- In the first page, select code S6-00 and the type of transitory loads. Activate the "2D" radio button, select the moving load axis, and specify the direction of traffic on the bridge.



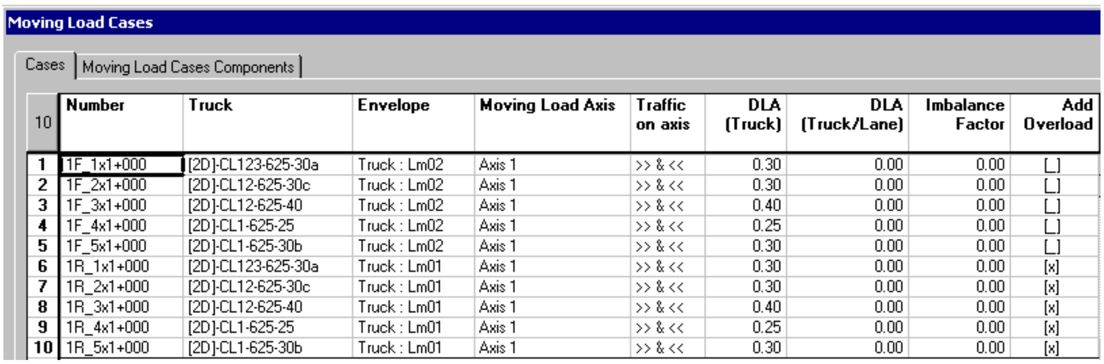
- Click the *Next* button.

In the next page, we can see that five moving loads will be applied at an offset of 0.0m from moving load axis because it is a 2D project. To withdraw a moving load from the list, click on the name to cancel the selection.



- Press the *Finish* button.

The following moving load cases were generated. Please refer to the previous example to know the nomenclature that is used for the numbering of cases.


Moving Load Cases

Cases | Moving Load Cases Components

10	Number	Truck	Envelope	Moving Load Axis	Traffic on axis	DLA (Truck)	DLA (Truck/Lane)	Imbalance Factor	Add Overload
1	1F_1x1+000	[2D]-CL123-625-30a	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
2	1F_2x1+000	[2D]-CL12-625-30c	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
3	1F_3x1+000	[2D]-CL12-625-40	Truck : Lm02	Axis 1	>> & <<	0.40	0.00	0.00	[ ]
4	1F_4x1+000	[2D]-CL1-625-25	Truck : Lm02	Axis 1	>> & <<	0.25	0.00	0.00	[ ]
5	1F_5x1+000	[2D]-CL1-625-30b	Truck : Lm02	Axis 1	>> & <<	0.30	0.00	0.00	[ ]
6	1R_1x1+000	[2D]-CL123-625-30a	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
7	1R_2x1+000	[2D]-CL12-625-30c	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]
8	1R_3x1+000	[2D]-CL12-625-40	Truck : Lm01	Axis 1	>> & <<	0.40	0.00	0.00	[x]
9	1R_4x1+000	[2D]-CL1-625-25	Truck : Lm01	Axis 1	>> & <<	0.25	0.00	0.00	[x]
10	1R_5x1+000	[2D]-CL1-625-30b	Truck : Lm01	Axis 1	>> & <<	0.30	0.00	0.00	[x]

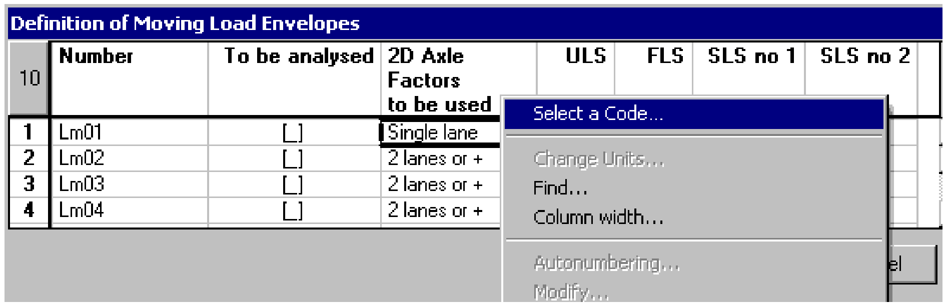
- Close the spreadsheet.

Moving Load Envelopes

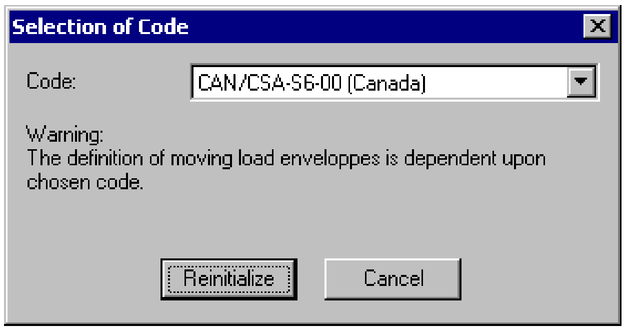
Moving load envelopes must be activated in this spreadsheet before launching a moving load analysis. When a code is selected in this spreadsheet, you obtain the required load combinations, and corresponding envelopes.

- Go to **Loads / Moving Load Envelopes**.

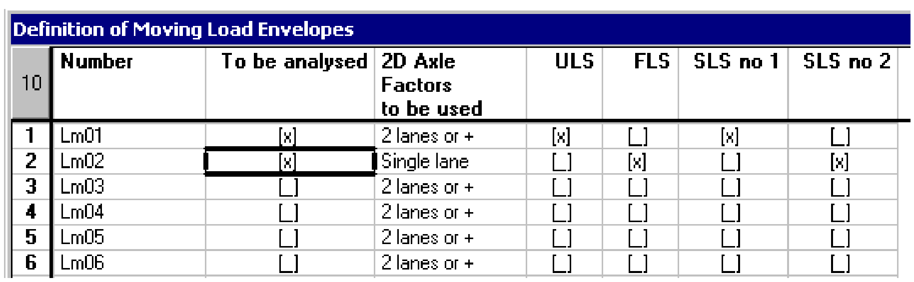
- Click in any cell and right click to open contextual menu. Choose the command **Select a code**.



- In the **Selection** dialog box, select code S6-00 and click the "Reinitialize" button.



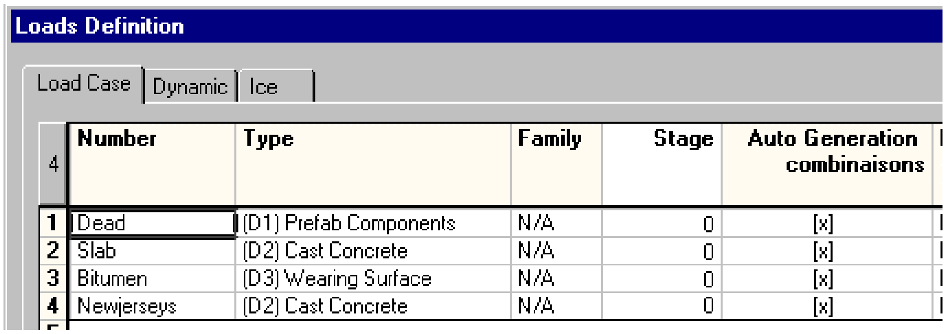
- Activate (double click) moving load envelopes Lm01 and Lm02.



Load Cases and Load Combinations

Load Cases

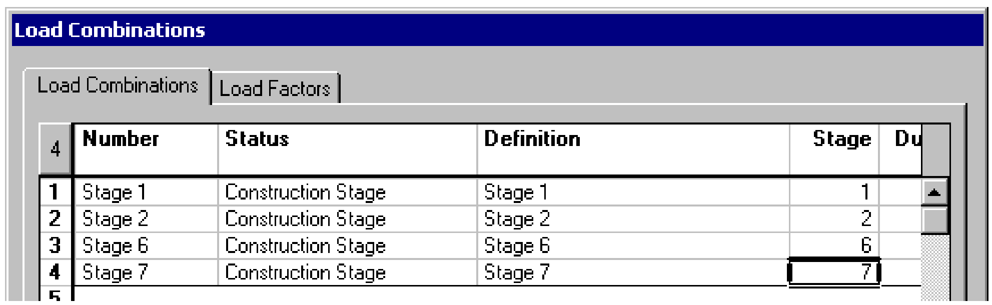
The following load cases have been defined in the **Loads Definition** spreadsheet (**Loads / Load Cases / Definition**) for the composite beam. They correspond to construction stages that we specified in the **Composite Beam** tab of **Project Configuration** dialog box.



4	Number	Type	Family	Stage	Auto Generation combinaisons
1	Dead	(D1) Prefab Components	N/A	0	[x]
2	Slab	(D2) Cast Concrete	N/A	0	[x]
3	Bitumen	(D3) Wearing Surface	N/A	0	[x]
4	Newjerseys	(D2) Cast Concrete	N/A	0	[x]

Load Combinations

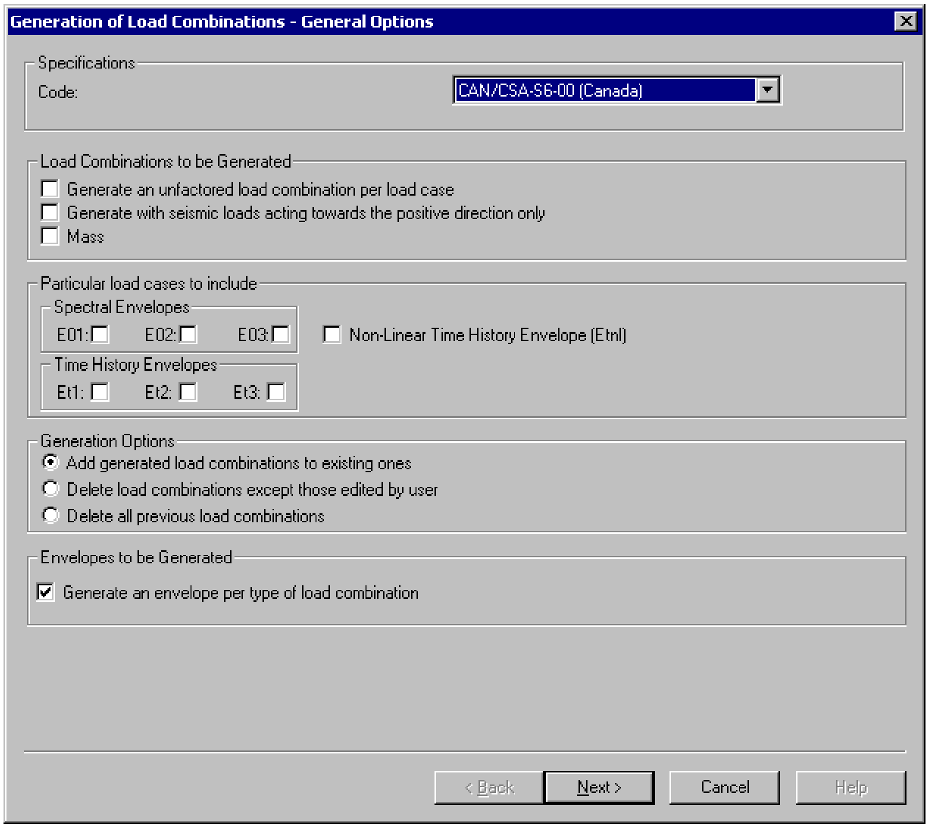
Construction stage load combinations must be defined in the **Load Combination** spreadsheet before using the **Load Combination Generation Wizard**.



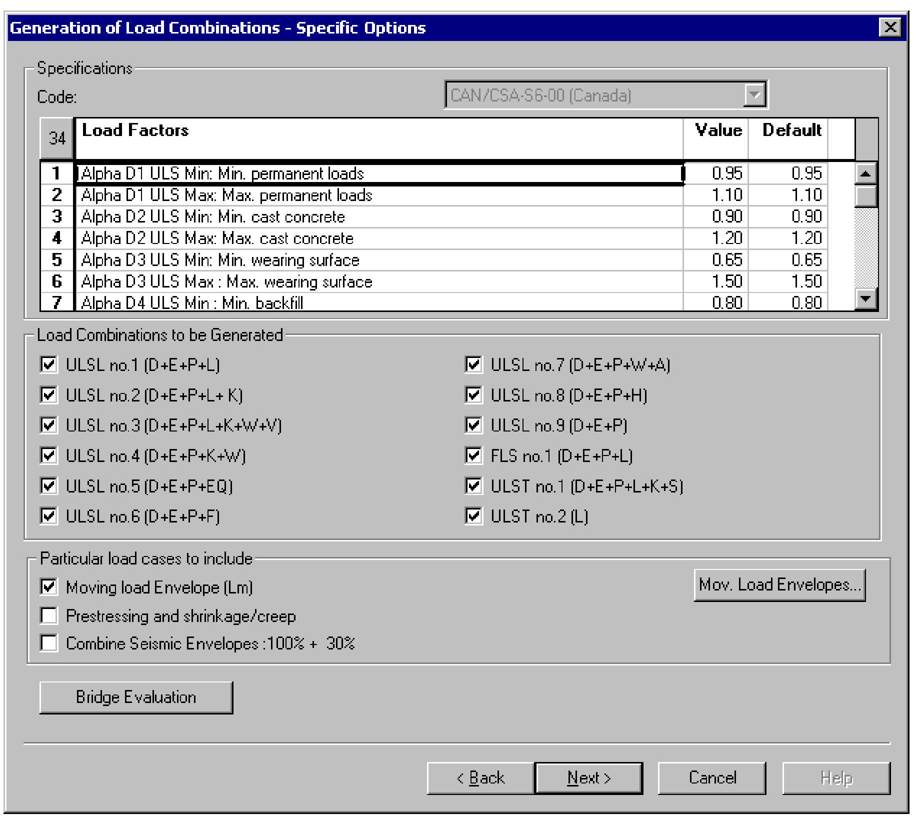
4	Number	Status	Definition	Stage	Du
1	Stage 1	Construction Stage	Stage 1	1	
2	Stage 2	Construction Stage	Stage 2	2	
3	Stage 6	Construction Stage	Stage 6	6	
4	Stage 7	Construction Stage	Stage 7	7	

When it is done, go to **Loads / Load Combinations / Generation Wizard**. Activate generation options.

- In the **General Options** page, select code S6-00 among the drop-down list box. Activate option *Add generated load combinations to existing ones* to avoid the deletion of construction stages. Generate standard envelopes.

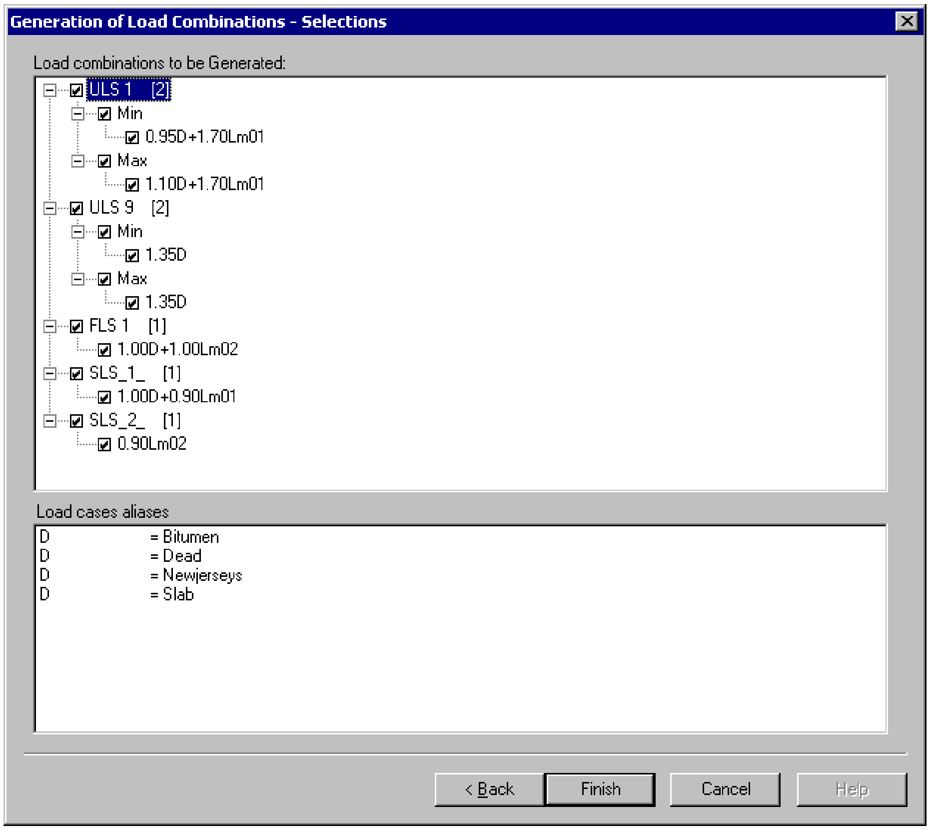

Generation of Load Combinations - General Options
Specifications
Code:
CAN/CSA-S6-00 (Canada)
Load Combinations to be Generated
Generate an unfactored load combination per load case
Generate with seismic loads acting towards the positive direction only
Mass
Particular load cases to include
Spectral Envelopes
E01: E02: E03:
Non-Linear Time History Envelope (Etnl)
Time History Envelopes
Et1: Et2: Et3:
Generation Options
Add generated load combinations to existing ones
Delete load combinations except those edited by user
Delete all previous load combinations
Envelopes to be Generated
Generate an envelope per type of load combination
< Back
Next >
Cancel
Help

- Click *Next*.


Generation of Load Combinations - Specific Options
Specifications
Code:
CAN/CSA-S6-00 (Canada)
34 Load Factors Value Default
1 Alpha D1 ULS Min: Min. permanent loads 0.95 0.95
2 Alpha D1 ULS Max: Max. permanent loads 1.10 1.10
3 Alpha D2 ULS Min: Min. cast concrete 0.90 0.90
4 Alpha D2 ULS Max: Max. cast concrete 1.20 1.20
5 Alpha D3 ULS Min: Min. wearing surface 0.65 0.65
6 Alpha D3 ULS Max : Max. wearing surface 1.50 1.50
7 Alpha D4 ULS Min : Min. backfill 0.80 0.80
Load Combinations to be Generated
ULSL no.1 (D+E+P+L)
ULSL no.2 (D+E+P+L+ K)
ULSL no.3 (D+E+P+L+K+W+V)
ULSL no.4 (D+E+P+K+W)
ULSL no.5 (D+E+P+EQ)
ULSL no.6 (D+E+P+F)
ULSL no.7 (D+E+P+W+A)
ULSL no.8 (D+E+P+H)
ULSL no.9 (D+E+P)
FLS no.1 (D+E+P+L)
ULST no.1 (D+E+P+L+K+S)
ULST no.2 (L)
Particular load cases to include
Moving load Envelope (Lm)
Prestressing and shrinkage/creep
Combine Seismic Envelopes :100% + 30%
Mov. Load Envelopes...
Bridge Evaluation
< Back
Next >
Cancel
Help

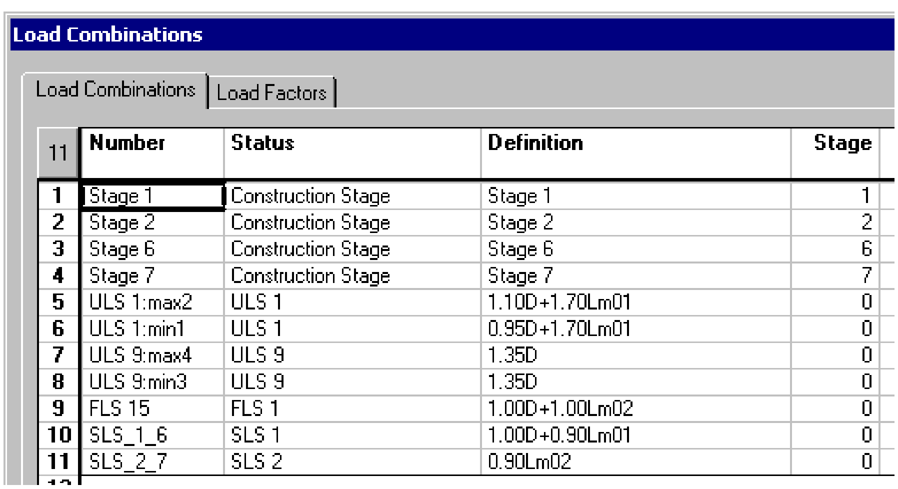
- Activate the insertion of moving load envelopes in the **Specific Options** page of the generator and click *Next*.

N. B. The "Mov. Load Envelopes" button opens the **Moving Load Envelopes** spreadsheet. We have already activated envelopes (page 297).



- Press *Finish*.

The **Load Combination** spreadsheet will be displayed on screen.



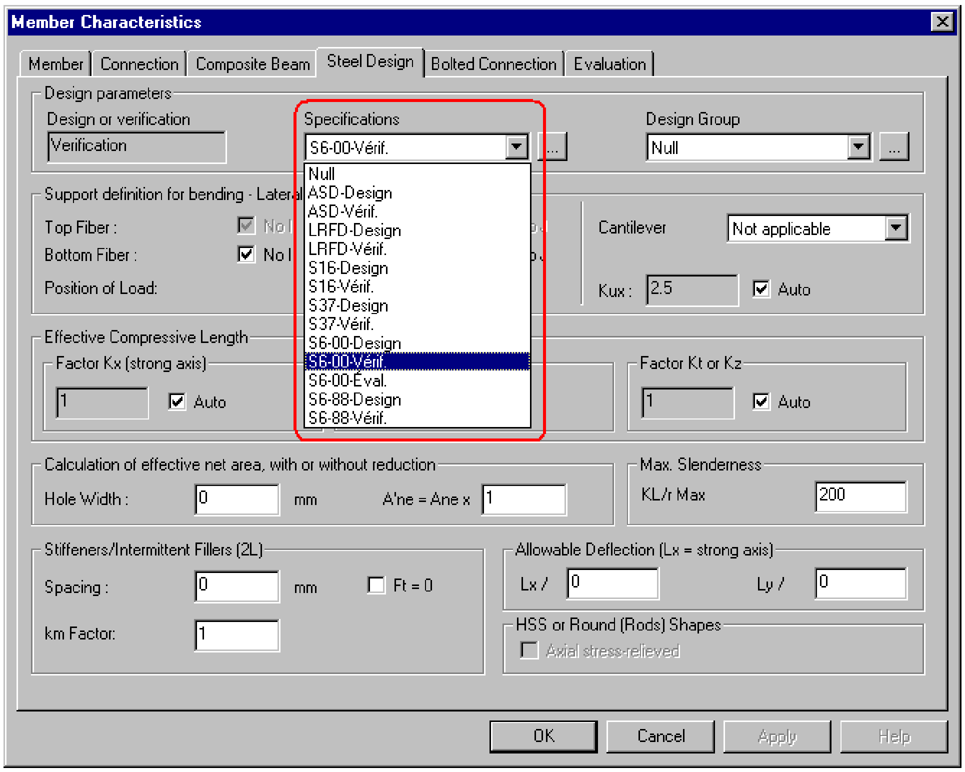
11	Number	Status	Definition	Stage
1	Stage 1	Construction Stage	Stage 1	1
2	Stage 2	Construction Stage	Stage 2	2
3	Stage 6	Construction Stage	Stage 6	6
4	Stage 7	Construction Stage	Stage 7	7
5	ULS 1:max2	ULS 1	1.10D+1.70Lm01	0
6	ULS 1:min1	ULS 1	0.95D+1.70Lm01	0
7	ULS 9:max4	ULS 9	1.35D	0
8	ULS 9:min3	ULS 9	1.35D	0
9	FLS 15	FLS 1	1.00D+1.00Lm02	0
10	SLS_1_6	SLS 1	1.00D+0.90Lm01	0
11	SLS_2_7	SLS 2	0.90Lm02	0

- Close the spreadsheet.

Steel Specification

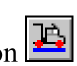
- Open the **Steel Specifications** spreadsheet through the **Structure** / **Specifications** menu. Consult the default values associated to specification *S6-00 Verification*. Modify parameters if necessary.

- After having activated design criteria in the **Member Characteristics** dialog box, go to the **Steel Design** tab and select specification *S6-00 Verification* in the drop-down list box.



- Close the dialog box.

Moving Load Analysis

- Open the **Moving Load Analysis** dialog box by pressing icon  on Tools toolbar or go to **Analysis** menu. Select options and Press the *Analyse* button to launch the analysis.

For more details, refer to previous example on 3D moving load analysis.

- Close the dialog box when analysis is completed.

Results

Envelopes

The Envelope mode is automatically activated when the analysis is completed and the dialog box closed.

- Select envelope *Lm01* or *Lm02* on Activation toolbar.
- Open the **View Options** dialog box and display a diagram;
- Access to numerical results through **Results / Envelopes**.

Design

The moving load analysis is part of any design (steel, reinforced concrete, and prestressed concrete) iterative process and will be automatically launched at each cycle of design. Therefore, we recommend that you select options (concomitant values) in the **Moving Load Analysis** dialog box before launching a design (steel, reinforced concrete or prestressed concrete design).

When the design is done, design results and load combinations results will be available, including envelope Lm01 and Lm02.