Volume

3

CIVILDESIGN INC.

VisualDesign Software – Version 5.9

April 2006

Complete User's Manual

Disclaimer

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

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

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

www.civild.com

Chapter

14

PRESTRESSED CONCRETE DESIGN

TABLE OF CONTENTS

Chapter 14 Prestressed Concrete Design

General

Prestressed Concrete Module

Basic Principles

This new module allows you to check your prestressed concrete design (with or without post-tensioning) for a 2D structure. The user must define and enter the exact position of cables in the elements. VisualDesign™ will check the design.

Project Configuration

You must complete the *Prestressing* tab of this dialog box if your structure has prestressed concrete elements (semi-continuous pre-tensioning or continuous post-tensioning). Call this dialog box by selecting **Project Configuration** under **File** menu.

Define construction stages by ticking off the appropriate boxes. Information will be used to calculate prestress losses in cables due to creep and shrinkage. Results, at each construction stages, will be added to the next stages.

Initial End Conditions

You must define the initial end conditions of the structure. Example: considering a pre-tensioning structure, you must not consider the slab in the initial stage.

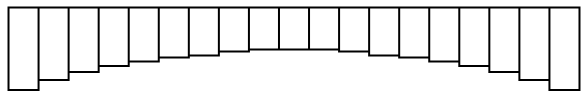
Make sure that superior fibres are aligned for each member. Otherwise, the continuous system will be erroneous.

Underbracing

If stage 3 is checked, the underbracing will be considered up to this stage. If stage 4 is checked, but stage 3 is not, the underbracing will be considered up to stage 4. Stage 5 may not exist if the chosen steel shape is already composed of a slab.

Example: Post-tensioning beam of variable inertia

All members are aligned at top fibres. To do so, choose the option "Above" for eccentricity *ey* in the **Connection** tab of **Member Characteristics** dialog box.



VisualDesign™ calculation method will modify end conditions for moving load analysis or for any load case, according to the stages specified in the **Load Combinations Definition** spreadsheet.

It is of prime importance to model the structure in an accurate way, as it is going to be built. Use more than one support instead of one because beams will not act the same. This will influence the calculation of forces and displacements in the supports and beams.

Design Specifications

Concrete specifications are required for design or verification. The construction code is specify in this spreadsheet and will be used for the calculation and also, parameters relative to the design of stirrups, concrete cover and rebars, among others.

For a pre-tensioning or post-tensioning structure, the user must choose the option "None" in column "Type of optimization" for main reinforcement.

IMPORTANT If you chose specification A23.3 for the design of a continuous system that is composed of prestressed concrete elements and concrete elements (without cables), you MUST use the General Method for Reinforced Concrete Design. The method is selected in the Concrete Design spreadsheet.

Position of cables

To accurately define cables characteristics, some basic information is required such as steel grades, strands and post-tensioning mechanisms. You will find these three spreadsheets in the **Common/Cables** menu.

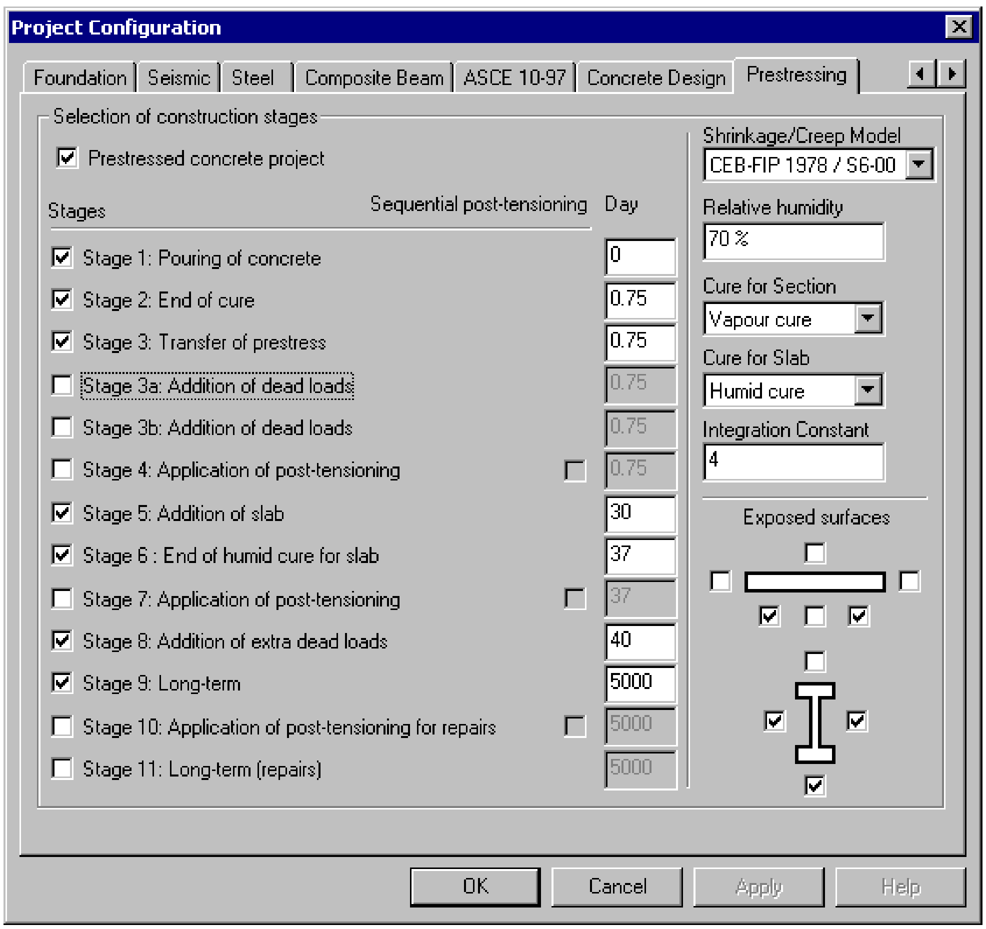
Two spreadsheets are used to define cable layouts: the **Group of Cables** and the **Cable Layouts** spreadsheets. The first one allows you to specify the number of cables in a group. It may be a single cable; it depends on the type of tensioning (with sheath or not). The second spreadsheet helps you to choose and define cable layouts. When defining new cable layouts, a dialog box will appear. It includes five pre-defined layouts.

Project Configuration

The Prestressing tab

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

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



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

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

Shrinkage and Creep Effects

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

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

See also

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

Calculation of Alpha for Prestressed Concrete

Ref.: Edward G. Nawy (2000), Prestressed Concrete –A Fundamental Approach, 3rd Edition, Prentice Hall, p.39. "Initial Compressive Strength and Modulus"

An appropriate value of f'ci is recommended for prestressed concrete project. Manufacturers use a concrete material that reaches a resistance of 40 MPa after 18 hours (for a 50 MPa concrete, after 28 days). Factor alpha allows accurately evaluating f'ci for a long-term period. This factor alpha is specified in the **Concrete Materials** spreadsheet.

Here are equations that will help you calculating this value:

f'ci can be evaluated from this equation:

$$f'_{ci}=\frac{t}{\alpha+\beta t}f'_c$$

Where:

f'ci = Concrete compressive strength at the prestressing stage

f'c = 28 days' compressive strength

t = Time in days

α = Factor depending on type of cement and curing conditions

= 4.0 for moist-cured type-I cement and 2.30 for moist-cured type-III cement

= 1.0 for steam-cured type-I cement and 0.70 for steam-cured type-III cement

β = Factor depending on the same parameters for α giving corresponding values of 0.85, 0.92, 0.95, and 0.98, respectively.

This equation can be modified knowing that f'ci = f'c after 28 days:

$$f'_{ci}=\frac{28}{\alpha+28\beta}f'_c$$

Then:

$$\alpha+28\beta=28$$

And:

$$\beta=\frac{28-\alpha}{28}$$

The initial equation becomes as follows:

$$f'_{ci}=\frac{t}{\alpha+(\frac{28-\alpha}{28})t}f'_c$$

Therefore, we can find a value for α when f'ci, f'c and t are known:

$$\alpha=\frac{t(f'_c-f'_{ci})}{f'_{ci}}\left(\frac{28}{28-t}\right)$$

See also

Concrete Materials Spreadsheet

Calculation of Strand Relaxation according to Steel Grade

Relaxation according to Steel Grade:

Type	fpy	Relaxation
Low-relaxation	0.90 Fu	Log10 (24.0*ft)/45.0 * (fsj/fpy – 0.55) * fsj
Smooth High-strength Bars	0.85 Fu	Log10 (24.0*ft)/10.0 * (fsj/fpy – 0.55) * fsj
Deformed High-strength Bars	0.80 Fu	Log10 (24.0*ft)/10.0 * (fsj/fpy – 0.55) * fsj
Normal Relaxation	0.85 Fu	Log10 (24.0*ft)/45.0 * (fsj/fpy – 0.55) * fsj

Relaxation according to Strand Layout

Layout	With Friction	Used Area	Relaxation
Internal without grout	Yes (1)	Sheath Area	Calculation of bw: reduction of width by 100% of sheath diameter, if necessary
Internal with grout	Yes (1)	Strands Area	Calculation of bw: reduction of width by 50% of sheath diameter, if necessary
External	Yes (2)	None	bw is not reduced

NOTE (1):

The calculation of friction loss is evaluated with usual equation fsj * (1 – exp (-K*L-μα)) taking into account the cone penetration Delta right and Delta left.

NOTE (2):

First, we calculate loss due to Delta right and Delta left. This loss is applied to initial tension, so its value is reduced. Then, losses in deviators are considered in a punctual manner with usual equation fsj * (1 – exp(-K*Ld-μα)), where "Ld" is the deviator length and α, the angular variation in the deviator, which is calculated according to the cable orientation. Therefore, the calculated losses will create graphic discontinuities at each deviator location within continuous system.

Theory on Creep and Shrinkage Effects

To know more about creep and shrinkage effects on prestressed concrete structures, go to our web site www.civild.com under **Learning Tools** / **Papers** and download the paper that is entitled *Shrinkage and Creep Effects on Prestressed Concrete Structures*.

Prestressing Elements

Cable Steel Grades Spreadsheet

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

Group: Shared Data: VDBase.mdb

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

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

For a low-relaxation 7 wire prestressing strand:

If $\varepsilon_p \leq 0,008$

We can say that:

$$f_p = \varepsilon_p \cdot E_p$$

If $\varepsilon_p > 0,008$

For a Grade 1760 Strand:

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

In VisualDesign:

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

So: σ_1 = 1749

k_1 = 0.433

ε_1 = 0.00614

α = 0.98

N.B. Be careful with units. MPa are used in formulas.

For a Grade 1860 Strand:

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

You will find the following values in the **Cable Steel Grades** spreadsheet:

σ_1 = 1848

k_1 = 0.517

ε_1 = 0.0065

α = 0.98

See also

Strands spreadsheet

Strands Spreadsheet

Define the type of strands that will be included in your prestressed concrete project. Select **Cables/Strands** under **Common** menu.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Type	Choose a type of strand among the drop-down list: Low relaxation, Smooth high strength bars, Deformed high strength bars, Normal relaxation	Double-click
Diameter	Enter the strand diameter. (This value is used for screen display only.)	Single click
Area	Enter the strand area	Single click
Material	Choose the strand steel grade among the drop-down list. To add or modify steel grades, select the **Steel Grades** spreadsheet under **Common/ Cables** menu.	Double-click

See also

Prestressed Concrete Module
Post-tensioning Mechanisms
Cable Groups Spreadsheet
Cable Layouts spreadsheet
Cable Steel Grades Spreadsheet

Post-tensioning Mechanisms

Define the post-tensioning mechanisms that will be used in your prestressed concrete project. Select **Cables/Post-tensioning Mechanisms** under **Common** menu. Two types are available: with sheath or through external deviator.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Distribution	Assign a "Public" or "Private" distribution to your personalized object. A private object will not be merged into another database at the opening of the file. The distribution of a pre-defined object is "Public" and is not editable.	Double-click
Type	Choose a type of post-tensioning mechanism: Sheath or External deviator.	Double-click
Wobble friction, K	Enter the wobble friction coefficient per meter of strand length.	Single click
Friction coefficient, Mu	Enter friction coefficient Mu.	Single click
Sheath diameter	Enter the sheath diameter (used for screen display only).	Single click
Length of deviator	Enter the length Ld of external deviator. This value is used to calculate prestress losses (friction between cable and deviator).	Single click
Deviator curvature	Enter the deviator curvature, ρd. (used for screen display only).	Single click

See also

Prestressed Concrete Module
Project Configuration (Prestressing tab)
Strands spreadsheet
Cable Group spreadsheet
Cable Layout spreadsheet

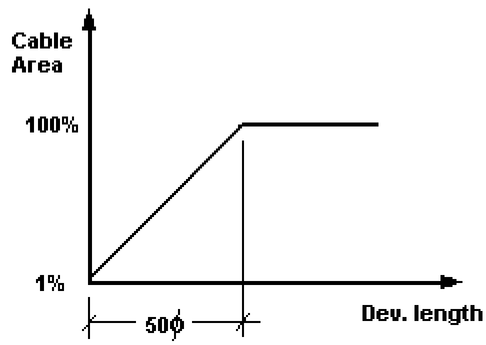
Cable Groups and Layouts

This spreadsheet is composed of two tabs: **Group of Cables** and **Cable Layout**. A cable layout must be defined for each group of cables.

This spreadsheet can be accessed through the **Rebar Placement** menu or by double clicking on a group of cables on screen.

Group: Shared Data: VDBase.mdb

Column	Description	Editing
ID	Automatically calculated	No
Number	Enter a number for this group of cables (12 alphanumerical characters).	Single click
Post-tensioning mechanisms.	Choose a post-tensioning mechanism (sheath or external deviator). Choose "Nil" for pre-tensioning.	Double-click
Number widthwise	This value represents either, the number of strands widthwise for pre-tensioning or, the number of sheathes for post-tensioning.	Single click
X start	Position of first strand or first sheath at the far left.	Single click
X end	Position of first strand or first sheath at the far right.	Single click
Nos. of strand/sheath	Number of strands per sheath. If there is no sheath or deviator, the value equals 1.	Single click
Strand	Choose the strand number that you wish to use.	Single click
Layout	Choose a strand layout among the drop-down list box: Internal with grout, Internal without grout or External	Single click
Jacking %	Indicate the percentage of tension (%fpu) relative to maximum stress fpu that will be applied at jacking.	Single click
Cone Left	Post-tensioning only: Indicate the cone penetration located left of continuous system, after post-tensioning.	Single click
Cone Right	Post-tensioning only: Indicate the cone penetration located right of continuous system, after post-tensioning.	Single click

Column	Description	Editing
Stage	Choose the construction stage where pre-tensioning or post-tensioning will be applied to the structure: Initial prestressing, initial post-tensioning, composite post-tensioning, post-tensioning for repairs	Double-click
Cable development length	This factor multiplies the cable diameter to obtain the development length for cables. 	Single click
Deterioration	**For Bridge Evaluation module only**: Indicate the percentage of the original surface that will be lost for the group of strands (or one strand) due to corrosion or other type of deterioration..	Single click

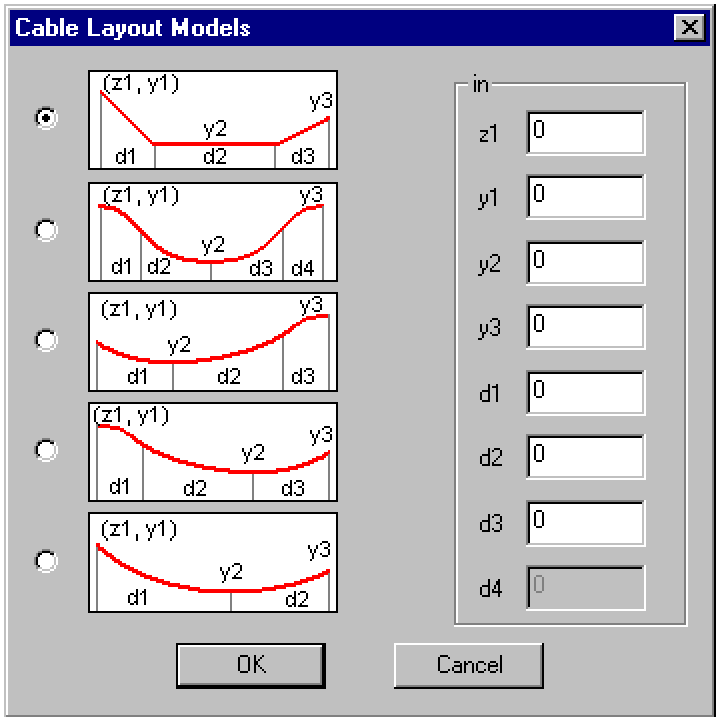
See also

Cable Layout Spreadsheet
The Bridge Evaluation module
Copying a Group of Cables along with Its Cable Layout

Cable Layouts Spreadsheet

Access this spreadsheet through the **Cable Groups and Layout** spreadsheet.

Use the following tool to define a cable layout. To open this dialog box, select any cell, right click, and select the **Automatic Generation** function in contextual menu.



When layouts are defined, the user can exit the dialog box. The **Layouts** spreadsheet will still be open on screen. Users can consult the spreadsheet and modify layouts as they wish.

Group: Structural data

Column	Description	Editing
ID	Automatically calculated	No
Number	Enter a number for this layout (12 alphanumerical characters).	Single click
Shape	Choose a layout shape among the drop-down list: Straight line, Parabolic or Circular.	Double-click
Beginning	Specify the cable fastening at the beginning of this sequence: Mobile, Fixed or Continuous.	Double-click
End	Specify the cable fastening at the end of this sequence: Mobile, Fixed or Continuous.	Double-click
z1	Position of the cable starting point, relative to continuous system.	Single click
y1	Vertical position of cable, in z, according to the continuous system axis. See the note below.	Single click

Column	Description	Editing
z2	Position of an intermediate point, relative to continuous system, in order to draw the parabolic part of the layout.	Single click
y2	Vertical position of cable, in z2, according to the continuous system axis. See the note below.	Single click
z3	End position of the layout parabolic curve or straight line, relative to continuous system.	Single click
y3	Vertical position of cable, in z3, according to the continuous system axis. See the note below.	Single click
Deterioration	**Bridge Evaluation Module only**: Enter the percentage of loss of the original area for the group of strands (or strand) that is due to corrosion or other deterioration. (0% means no deterioration.)	Single click

Note: To measure this distance, follow the convention used for rebar placement in the Reinforced Concrete Design module. See the topic Main Reinforcement spreadsheet

Copying a Group of Cables and Its Cable Layout

Procedure

- Open the **Cable Groups** spreadsheet in **Rebar Placement** menu.
- Select the line that corresponds to the group of cable you want to copy.
- Right click and choose function **Duplicate** available in the spreadsheet's contextual menu.

Or

- Double-click on a group of cables on elevation view or in an appropriate cross-section.
- The **Cable Groups** spreadsheet will include data on the select group only.
- Select the line, right click, and choose function **Duplicate** available in the spreadsheet's contextual menu.

See also

Spreadsheets' Contextual Menu
Duplicate function

Composite Members

Composite Slabs Spreadsheet

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

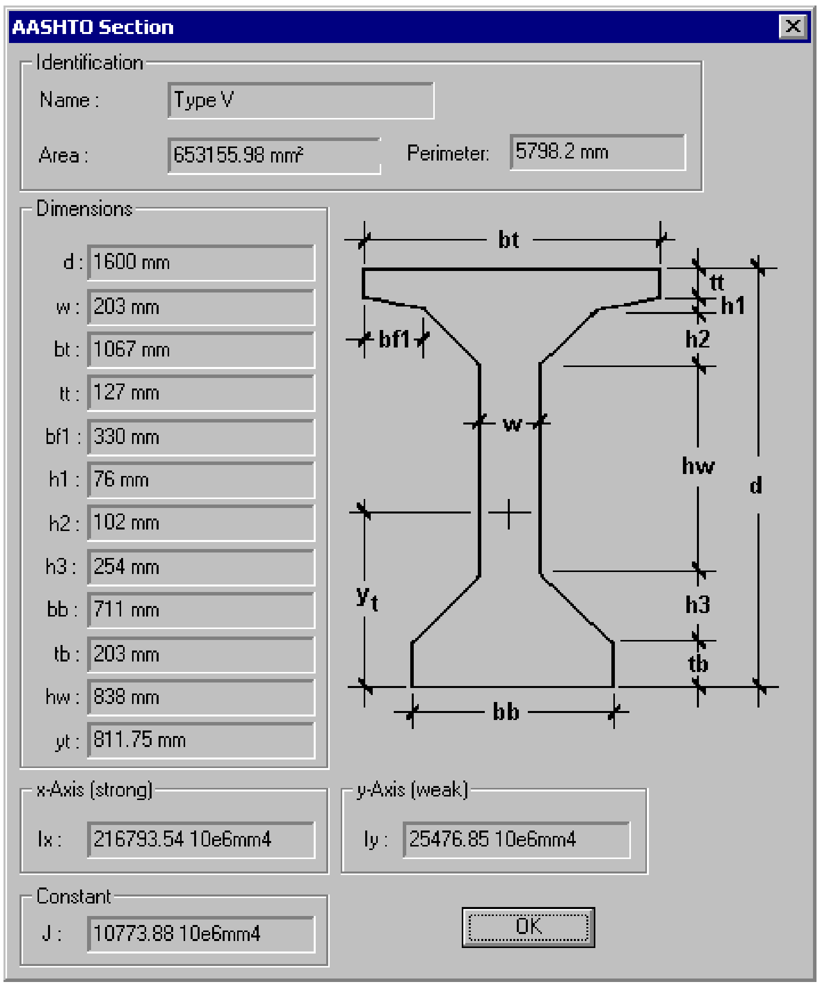
Group: Structural data

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

AASHTO Sections spreadsheet

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

A T section can be modeled from an AASHTO section, using appropriate dimensions in the AASHTO spreadsheet. To learn more, refer to the topic: Prestressed Concrete T-Section



Group: Shared Data: VDBase.mdb

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

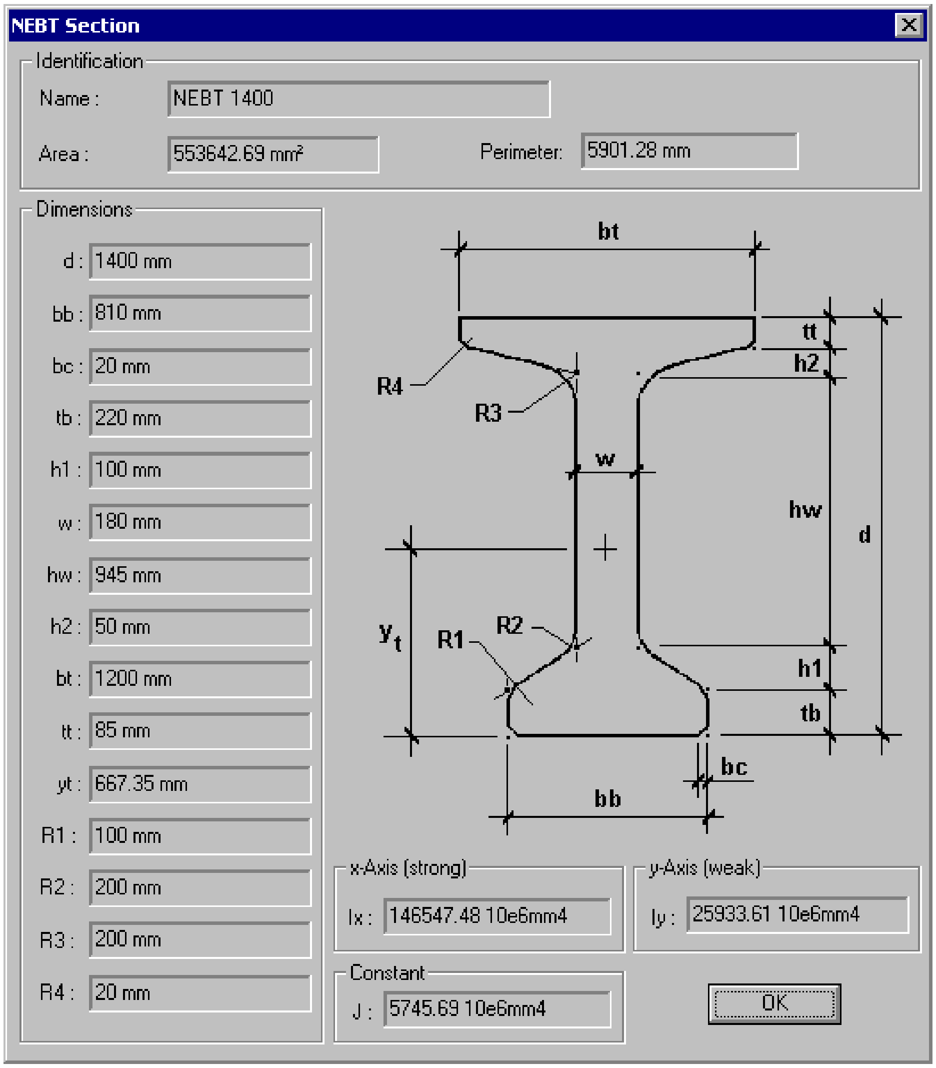
Column	Description	Editing
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Total height of the section	Single click
w	Thickness of the web	Double-click
bt	Total width of top flange	Single click
tt	Thickness of top flange	Single click
bf1	See the figure above.	Single click
h1	See the figure above.	Single click
h2	See the figure above.	Single click
h3	See the figure above.	Single click
bb	Total width of bottom flange	Single click
tb	Thickness of bottom flange	Single click
hw	Height of the web having a thickness w.	No
Area	Area of the section	No
yt	Distance from bottom to centre of gravity	No
Ix	Section inertia – strong axis	No
Iy	Section inertia – weak axis	No
J	Torsional constant	No
Perimeter	Perimeter of the section.	No

See also

About Shapes
Prestressed Concrete Composite Beams

NEBT Sections spreadsheet

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



Group: Shared Data: VDBase.mdb

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

Column	Description	Editing
Imperial Designation	The imperial designation for this section (12 alphanumeric characters)	Single click
Material	Choose the shape material among the list box.	Double-click
Distribution	Assign a "Public" or "Private" distribution to your personalized shape. A private shape will not be merged into another database at the opening of the file. The distribution of pre-defined shapes is not editable.	Double-click
d	Total height of section	Single click
bb	Total width of bottom flange	Single click
bc	Width of truncated part (45 deg.) of bottom flange.	Single click
tb	See the figure above.	Single click
R1	Radius of curvature of bottom flange. (See the figure above)	Single click
R2	Radius of curvature of bottom flange. (See the figure above)	Single click
h1	See the figure above.	Single click
w	Web thickness of the section	Single click
hw	Height of web having a thickness w.	Single click
R3	Radius of curvature of top flange. (See the figure above)	Single click
R4	Radius of curvature of top flange. (See the figure above)	Single click
h2	See the figure above.	Single click
bt	Total width of top flange.	Single click
tt	See the figure above.	Single click
Area	Area of the section	No
yt	Distance from bottom fibre to centre of gravity	No
Ix	Section inertia – strong axis	No
Iy	Section inertia – weak axis	No

Column	Description	Editing
J	Torsional constant	No
Perimeter	Perimeter of the section.	No

See also

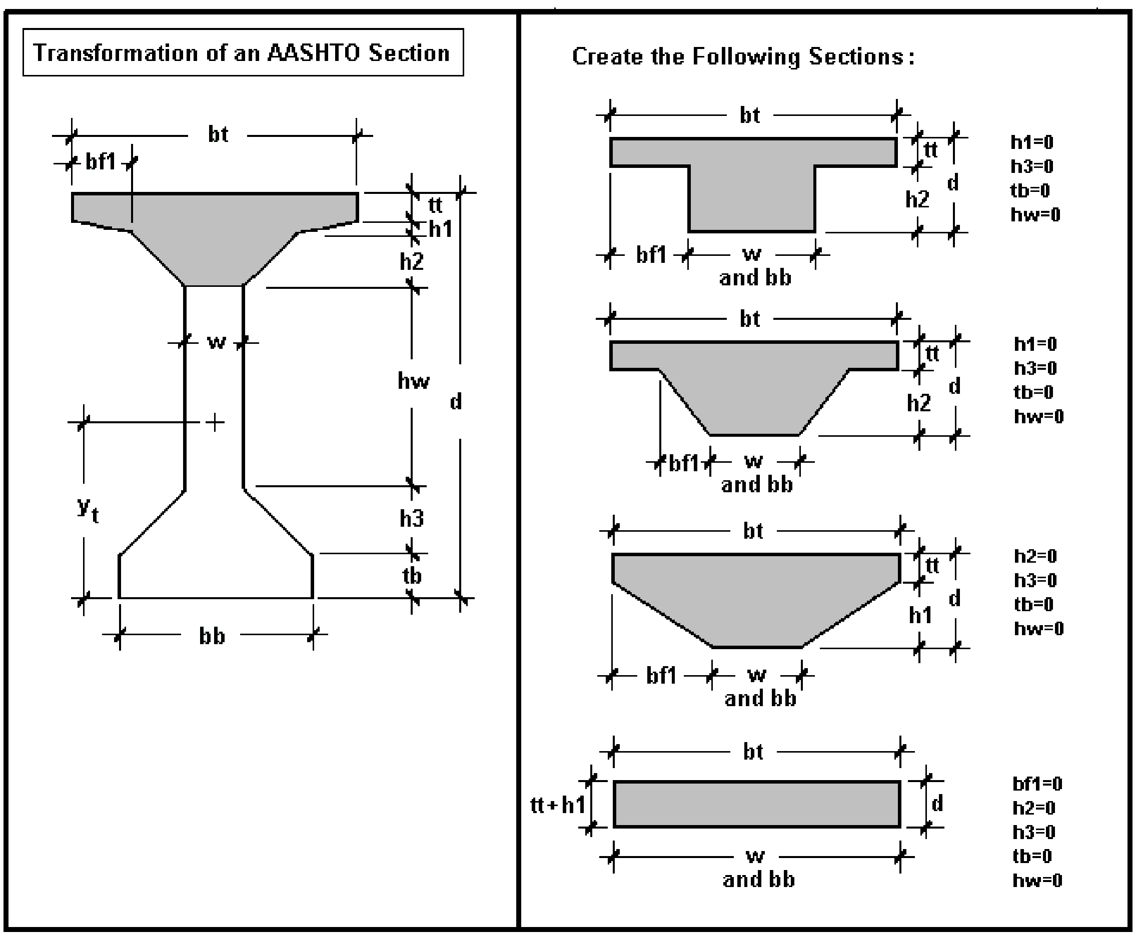
About Shapes
Prestressed Concrete Module
Prestressed Concrete Composite Beams

Prestressed Concrete T Sections

Composite T-Section

T-Sections are mostly used with post-tensioning cables.

Model a Composite T-Section using an AASHTO Section and Transform it into different shapes, as shown below:



Procedure:

- Select the AASHTO Sections spreadsheet in the **Common** menu / **Shapes.**
- Insert a line at the end of the spreadsheet. Give a name to this section and enter the required parameters to create a new section.
- Define a slab in the **Composite Slabs** spreadsheet (**Structure** menu).
- Define a member as *Composite Beam* in the "Composition" field in the **Member Characteristics** dialog box.

Standard T Shape

Use the T-Shapes spreadsheet for a T-section that is not composite.

Procedure:

- Select the **T-Shapes** spreadsheet in the **Common** menu / **Shapes.**
- Insert a line at the end of the spreadsheet. Give a name to this section and enter the required parameters (b, d, w, and t). VisualDesign will automatically calculate other properties.
- Define a member as *Standard* in the "Composition" field in the **Member Characteristics** dialog box.

See also
AASHTO Sections spreadsheet
Composite Prestressed Concrete Beam

Composite Prestressed Concrete Beam

Follow the procedure to properly define a composite prestressed concrete member.

- Define a slab in the **Composite Slabs** spreadsheet (**Structure** menu).
- In the **Member** tab of **Member Characteristics** dialog box:
  - Choose an AASHTO or NEBT section and specify the material as concrete.
  - Activate the design criteria and choose a "Composite beam" composition.
  - Select the construction stage number where the member will be effective.
  - In the zone "For moving load analysis", check the "Mobile axis" box if the element is part of the moving load axis.
- In the **Connection** tab:
  - Align the section according to eccentricity "ey".

- Define "ez" rigid extensions.

- In the **Composite Beam** tab:

  - Choose the type of slab that you defined in the Slabs spreadsheet. Enter an effective and real b. Check the box "Use composite beam inertia Ix". ***Never tick off the box "Add dead load of slab" for a prestressed concrete project because it will be installed after construction***.

  - In the zone "Composite beams", choose the statically determinate beams end conditions, according to strong axis, for construction stages 1 to 5. VisualDesign™ will apply these end conditions when transferring to a statically indeterminate structure.

Calculation of inertia

Values of inertia for composite (concrete-concrete) sections are now represented with the usual transformed inertia. For Canadian and American codes, the calculation of inertia with interfacial slip is not considered for this type of composite section.

Rigid Floors and Bridge Design...Warning!

Generally, a diaphragm action is never used for bridges. If the floors axis is different from beam axis, it creates an inappropriate composite effect between these two types of elements. Also, the diaphragm action induces a resistance against temperature variation and finally, the diaphragm absorbs prestressing effects, which is not good.

See also

AASHTO Sections
NEBT Sections
Procedure – Prestressed Concrete Analysis
Statically determinate and indeterminate structures
Cable Groups
Cable Layouts
Load Cases
Load Combinations
Copying a Group of Cables along with Its Cable Layout

Load Cases and Load Combinations

Prestressed Concrete Load Cases

It is of prime importance to properly define all load types that will be applied to the structure according to the construction stages that have been specified in the **Project Configuration** (**Prestressing** tab).

Select the **Loads Definition** spreadsheet in the **Loads/Load Cases / Definition** menu. Insert lines and define load cases that will be superimposed according to construction stages. For example, create an additional dead load case to account for the beam and a second one, for the composite slab. You can also add temperature loads.

Prestressing loads (jacking) and the pre-tensioning or post-tensioning stages are defined in the **Cable Groups** spreadsheet.

NOTES:

The "Prestressing" type of load is a virtual load that does not need to be defined in the **Loads Definition** spreadsheet. It corresponds to the stresses added at all construction stages. These stresses will be multiplied according to the load factor that you specify in the **Load Factor** tab.

"Creep/Shrinkage" is also a virtual load. VisualDesign™ recognizes this type of load and recovers the secondary prestress due to creep and shrinkage.

See also

Project Configuration
Prestressed Concrete Load Combinations
Statically Determinate and Indeterminate Structures
Groups of Cables spreadsheet
Loads Definition spreadsheet

Construction Stage Load Combinations

Load combinations must be compatible with user-defined construction stages in the project configuration otherwise warning messages will be displayed on screen.

Load Combinations with Construction Stages:

Define load combinations that will be applied to each construction stages. For each load combination, double-click in the "Stage" cell and choose the stage where it will be applied. Do not cumulate load cases at each construction stage. The software automatically does it. Ex. The dead load at stage 1 will be automatically added to the dead load specified at stage 2, etc.

The load combinations' statuses for construction stages must be set to "Construction stage". Please refer to topic Load Combination Status.

Load Factors and Selected Load Cases:

Select the **Load Factors** tab and include all load cases that are part of a load combination, in the right part of the dialog. Double-click in the "Load Case" cell and choose the type of load that will be applied to this load case: Dead, Creep/Shrinkage, Prestressing (see the notes below), Formwork, Additional Dead Load, etc.

Enter the ULS load factors for each load case that is composing load combinations. ULS load factors for load cases other than the ***Prestressing*** and ***Creep/Shrinkage*** must be identical for construction stages. If they are different, the stresses for serviceability load combinations will not be available in the Graphic Results. Refer to the topic: Construction Stages and Serviceability Load Combinations.

The Prestressing and Creep/Shrinkage "Virtual" Load Cases:

All load combinations representing construction stages must include at least these two virtual load cases: "Prestressing" and "Shrinkage/Creep".

The ***Prestressing*** type of load is a virtual load that needs not to be defined in the **Loads Definition** spreadsheet. It corresponds to the stresses added at all construction stages. These stresses will be multiplied according to the load factor entered in the **Load Factor** tab.

Creep/Shrinkage is also a virtual load. VisualDesign™ recognizes this type of load and recovers the secondary prestress results due to creep and shrinkage.

See also
Usual Load Combinations
Project Configuration
Prestressed Concrete Load Cases
Groups of Cables spreadsheet
Statically determinate and indeterminate structures
Definition of Load Combinations spreadsheet
Copying a Load Combination along with Load Factors

Construction Stages and Serviceability Load Combinations

Construction stage load combinations must have the same load factors as those included in serviceability load combinations because the stresses calculated during the analysis of construction stages are used for calculating stresses for serviceability load combinations.

For example, these load combinations are adequate:

Stage 3: 0.8 (Shrinkage+Creep) + 1.0P + 1.0 DL_beam
Stage 5: 0.8 (Shrinkage+Creep) + 1.0P + 1.0 DL_slab
Stage 6: 0.8 (Shrinkage+Creep) + 1.0P
Stage 8: 0.8 (Shrinkage+Creep) + 1.0P + 1.0 Ds + 1.0DL_barrier
Stage 9: 0.8 (Shrinkage+Creep) + 1.0P

Temperature loads and moving load envelopes must not be part of any construction stage load combinations.

For serviceability load combinations, load cases must be AT LEAST the same as for construction stages and generally, other effects are included as required by the building code, such as temperature and moving load envelope.

SERVICE SLS NO1, A:

0.8 (Shrinkage+Creep)+1.0P+1.0(DL_beam+DL_slab+Ds+DL_barrier)+0.8Tmax+ 0.9Lm

SERVICE SLS NO1, B:

0.8 (Shrinkage+Creep)+1.0P+1.0(DL_beam+DL_slab+Ds+DL_barrier)+0.8Tmin+ 0.9Lm

Other load combinations may be defined but load factors must be the same as construction stage load combinations, otherwise they will be available for displaying forces in VisualDesign main window, but not stresses (in VisualDesign *Rebar Placement* window).

Stresses will not be available in the *Rebar Placement* window because the load factors will not be compatible with those included in construction stages.

Explanation:

In VisualDesign, the calculation of stresses is done at each construction stage and these stresses include effects of all load cases include in a stage. Therefore, stresses for shrinkage only, or creep, or dead load only, are not available. When the analysis is completed, we cannot supply the stresses individually. However, forces are saved individually and diagrams can be displayed in VisualDesign main window for any load combination.

For example, the results for these load combinations will be available in VisualDesign main window but will not in the *Rebar Placement* window because load factors are not compatible with the ones defined in construction stages:

Combin 101: 1.0 (Shrinkage+Creep)

Combin 102: 1.0P + 1.0 (Shrinkage+Creep)

Combin 103: 1.0 (DL_beam + DL_slab + Ds + DL_barrier)

Usual Load Combinations

When the construction stage load combinations are defined in the **Load Combination Definition** spreadsheet, use the **Load Combination Generator** to usual load combinations as required by the building code.

All load combinations having a *Construction Stage* status will be available in the "Stage" list box of the **Graphic Results** dialog box. The moving load envelope *Lmi* and temperature loads will be included in these load combinations.

Load combinations that have a *Fatigue* status are applied to the statically indeterminate structure and VisualDesign™ automatically includes all the stresses that are cumulated at each construction stage.

See also
Construction Stage Load Combinations
Load Combination Generation Wizard
Project Configuration

Load Combination Statuses (General)

Load combination statuses must be specified in the **Load Combinations Definition** spreadsheet, through a selection tree when double clicking in the "Status" cell. The roots that you will find in this selection tree correspond to a code or standard and they are composed of a list off available statuses. Available Codes and Standards are: CNBC-95-ULS, CNBC-95-WSD, CNBC-05-ULS, CAN/CSA-S6-00, CAN/CSA-S37-01, AASHTO-LRFD-04, ASCE-02-SD, and ASCE-02-ASD.

Statuses are automatically initialized when the **Load Combination Generator** is used to generate required load combinations per selected code.

If a design is not required, use the general statuses that are listed under the **No code** root, namely *Analysis only*, *Ultimate*, *Service*, *Fatigue*, *Instantaneous deflection*, *Long-term deflection*, and *Mass*.

The table below describes the statuses that are common for all codes and standards:

Status	Description
Analysis only	Management of analyses. This load combination will be analysed and included in envelopes but will not be considered for a design.
Instantaneous Deflection	Instantaneous deflection usually includes live loads only.

Status	Description
Long-Term Deflection	Long-term deflection usually includes permanent loads only.
Mass	This load combination is required for modal analysis and usually includes the structure dead loads and some percentage of snow loads, among others.
Stage	This status is specific to load combinations that correspond to a construction stage. The Steel Design module is required for analyzing composite beams with construction stages and the Prestressed Concrete module is required for verifying prestressed composite beams with construction stages.

Specific Statuses

Please refer to the following topics (Chapter 4):

CAN/CSA-S6-00
AASHTO-LRFD-04
ASCE-7-02 SD
ASCE-7-02 ASD

See also

Load Combinations spreadsheet
Load Combination Generator
Procedure: Non-linear Time History Analysis
Load combinations for Composite Beams with Construction stages
Load combinations for Prestressed Concrete Beams
General Results spreadsheet – Reinforced Concrete Design
Generation of Resistance, Deflection, and Fatigue Envelopes

Analysis and Procedure

Analysis - Prestressed Concrete

To analyze prestressed concrete elements (verification only), press the **Analysis and Design** icon of Tools toolbar.

VisualDesign™ will design concrete elements having no prestressing cables and will analyze elements with cables in an iterative way,. The program considers strains and elastic losses in the cables when transferring prestress. It calculates and iterates until it reaches convergence (compatibility) between prestress loss in cables and strains.

Furthermore, VisualDesign™ uses the appropriate stiffness matrix for statically determinate and indeterminate structures.

IMPORTANT If you chose specification A23.3-94 for the design of a continuous system with prestressing cables having some elements without cables, you MUST select the *General* method in the **Concrete Specifications.** If you select the *Simplified* method, results will be incorrect.

Date and Time

VisualDesign now displays the date of analysis and design in the **Design** dialog box. You will also find the time it was launched and the time it ended in the upper part and lower part of the dialog box.

See also

Prestressed Concrete Module
Procedure – Prestressed Concrete analysis
Statically determinate and indeterminate structures
Load Cases
Load Combinations

Statically Determinate and Indeterminate Structures

When analyzing prestressed concrete elements, some load cases are applied to a statically determinate structure (construction stages up to composite effects) and some others are applied to a statically indeterminate structure. VisualDesign™ recognizes types of loads that have to be applied to such systems: the program will use the appropriate stiffness matrix for the calculation. After the calculation, it will combine the results obtained from the statically determinate and indeterminate structure.

Actually, when calculating the load combinations, VisualDesign™ first analyses the statically determinate structure using the dead loads, additional dead loads and prestressing loads defined according to construction stages, before considering this system as statically indeterminate structure (composite effect of beam-slab).

Then, the program considers elements as continuous and fixes the beams end conditions to "fix-fix". The system then becomes statically indeterminate. VisualDesign™ does another analysis with appropriate loads such as "Prestressing" with additional loads due to composite effect and moving loads, as you defined them in the load combinations. When a "Prestressing" type of load is selected in the **Load Factor** tab, it represents the cumulative stresses of all construction stages. This load case is applied to a statically indeterminate structure.

Load combinations that have a "Fatigue" status are also applied to a statically indeterminate structure and VisualDesign™ automatically includes all stresses generated during all construction stages.

See also
Load Cases
Load Combinations
Project Configuration

Member with a Linear Behaviour

It is possible to define a member having a linear behaviour at all times even in a non-linear analysis. To do so, select option "Linear only" among the *Behaviour* drop-down list box of the Member tab.

This functionality is useful to model the small members that are located between bridge supports and pier supports. With a linear behaviour, these members will not create horizontal components (usually created in a non-linear analysis). Consequently, only axial forces will be transferred and the convergence will be faster than before.

See also
Member Characteristics Dialog Box
The Member tab

Procedure - Prestressed Concrete Analysis

1. Project Configuration

- Select the **Prestressing** tab of **Project Configuration** dialog box. Check the "Prestressed Concrete Project" box to activate the dialog box. Check the appropriate construction stages corresponding to your project.

- Select the **Analysis** tab of **Project Configuration** dialog box. In the zone "Parameters for non-linear analysis", enter a number of iterations greater than 5 (analysis of construction stage 3 sometimes takes more than 5 iterations).

- Select the **Concrete** tab of **Project Configuration** dialog box and modify parameters if required.

2. Concrete Specification and Selection of reinforcement

- Go to **Structure** / **Specifications / Concrete.** Select the code or standard that will be used and modify default parameters, if needed. Select the *General Method* for the design of concrete elements. Choose transverse and longitudinal reinforcement that will be used for the design of concrete elements, if any.

3. Slab

- If beams are composite, define a slab in the **Composite Slabs** spreadsheet of **Structure** menu.

4. Member Characteristics Dialog box

- Activate design criteria in the **Member** tab of **Member Characteristics** dialog box. Choose a concrete section. If the section is composite (AASHTO or NEBT), select "Composite beam" as composition and specify the construction stage number where the member should be effective. (Usually, stage 2.)

- Then, go to the **Composite Beam** tab. Select the slab that you created before and choose the prestressed concrete elements end conditions according to strong axis, for construction stages 1 to 5.

- Define member rigid extensions in the **Connection** tab. N.B. The *ez* rigid extension at node i will be positive and the one at node j will be negative. Make sure that elements are aligned. Example: if gravity axis is y, align elements by setting eccentricity *ey* "Above" the axis passing through nodes i and j.

5. Continuous Systems Spreadsheet

- Go to **Structure / Continuous Systems**. In the "Specification" column, choose the concrete specification that will be used for design or verification. Define the type of exposure and concrete cover. Enter crack control parameters.

6. Strands and Post-tensioning Mechanisms

- Go to **Cables** heading of **Common** menu land define steel grades, strands and post-tensioning mechanisms.

7. Definition of Cable Groups and Cable Layouts

- Activate the "Rebar Placement" mode by double-clicking on any continuous system in order to open the *Rebar Placement* window.

- Go to **Rebar Placement** / **Cable Groups and Layouts.** Insert the appropriate number of lines to define each cable group and complete the information requested. Specify the jacking and the pre- or post-tensioning stage where it will be applied.

- Go to the **Cable Layout** tab. For each group of cables, define the cable layout using the **Cable Layout Models** dialog box. To open it, click any cell in the spreadsheet, right click, and select **Automatic Generation** in contextual menu.

8. Load Cases, Load Combinations and Envelopes

- Go to **Loads** / **Load Cases** /**Definition**. Enter load case titles and types according to appropriate code.

- Go to **Loads / Load Combinations / Definition**. Define load combinations that will be applied to each construction stage. For each one, select a *Construction Stage* status and specify the stage number.

- Select the **Load Factors** tab. For each load combination, double-click in the "Load Case" cell and select the right type of load in the list box. Enter load factors.

- Use the **Load Combination Generator** to generate other required load combinations as per selected code.

9. Moving Load Analysis

- Launch the moving load analysis. (See the Procedure for Analysing Moving Loads)

10. Analysis and Design

- Launch the analysis and design. VisualDesign™ will verify prestressed concrete elements (elements having cables) and will design concrete elements that have no cables

11. Interpreting Results:

- Activate the "Rebar Placement" mode on Activation toolbar and double-click on a continuous system to open the *Rebar Placement* window.

Forces and Resistances Diagrams:

- In the *Rebar Placement* window, select the **View Options** dialog box. In the *Rebar Placement* tab, check the boxes that correspond to the diagrams that you want to look at. Check the *Dimension* and *General* roots.

Results: Stresses in Cables and Concrete:

- Select cables results in the **Results** menu of *Rebar Placement* window. You will find the following results: Prestress losses in cables, Intermediate Results and Stresses in cables and in concrete (spreadsheets).
- In the **Results** menu also, look at results in the form of graphs, for "Service" and "Construction Stage" load combination, by selecting **Graphs**.

Various Numerical Results:

- Select the **General Results** spreadsheet under **Results** menu. This spreadsheet includes many calculated parameters for Reinforced Concrete Design. Yellow lines mean that there are parameters exceeding the code requirements.

See also

Prestressed Concrete Module
Load Cases
Load Combinations

Rebar Placement Window

Rebar Placement Window

This window includes all functions useful to check, modify and design the required reinforcement for concrete members, including prestressed concrete, that are part of any continuous systems. The name of the project file is written in the upper part of the screen.

Activate the *Rebar Placement* mode and double-click on any continuous system to open this window. Then, you will be allowed to look at the design or to create your own design by adding and placing main reinforcement and stirrups.

Once that the Rebar Placement mode is activated, access the *Rebar Placement* window by doing one of the following procedure:

- Click on any continuous system and select the **Properties** function;
- Double-click on any continuous system.

The *Rebar Placement* window is composed of the following menus:

FILE

Close
Save Rebar Placement
Save Rebar Placement As
DXF Out
Project Configuration
Print
Print Preview
Printer Configuration

EDIT

Undo
Redo
Properties
Add a Longitudinal Rebar
Translation
Delete
Activate Cursor Mode
Move
Stretch

VIEW

View Options
Zoom Window
Global Zoom

Zoom +
Zoom -
Dynamic Pan
Toolbars
Status Bar

COMMON

Concrete Materials
Rebar Steel Grades
Standard Reinforcing Bars
FRP Reinforcing Bars
Meshes
Bending Shapes
Cable Steel Grades
Strands
Post-tensioning Mechanisms

REBAR PLACEMENT

Continuous System
Concrete Specifications
Main Reinforcement spreadsheet
Transverse Reinforcement spreadsheet
Cross sections
Cable Groups
Cable Layout
Bar List - Continuous System
Automatic Generation of Cross-sections

RESULTS

Loss of Prestress in Cables
Stresses at each construction stage
Graphs
Intermediate Results
General Results
3D Interaction Curves

HELP

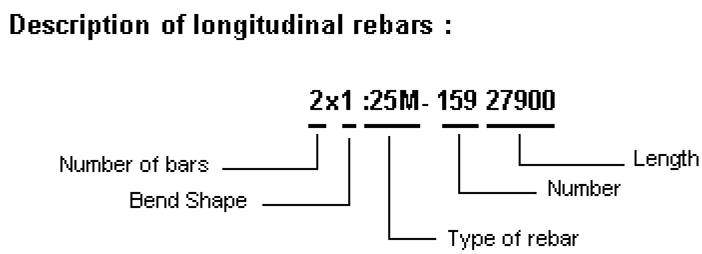
Using Help
Reminder - Editing Keys
About VisualDesign™

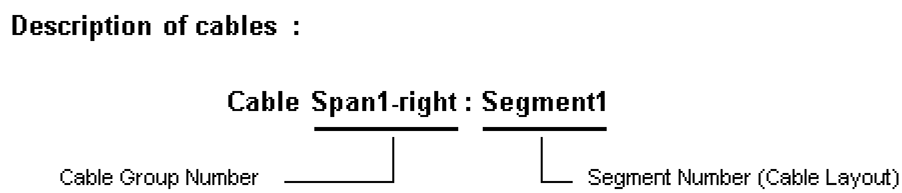
See also

Rebar Placement window Legend

Rebar Placement Drawing

When activating the *Rebar Placement* window, display rebars information through the **Rebar Placement** tab of **View Options**. The description of main reinforcement and prestressed cables is as follows:

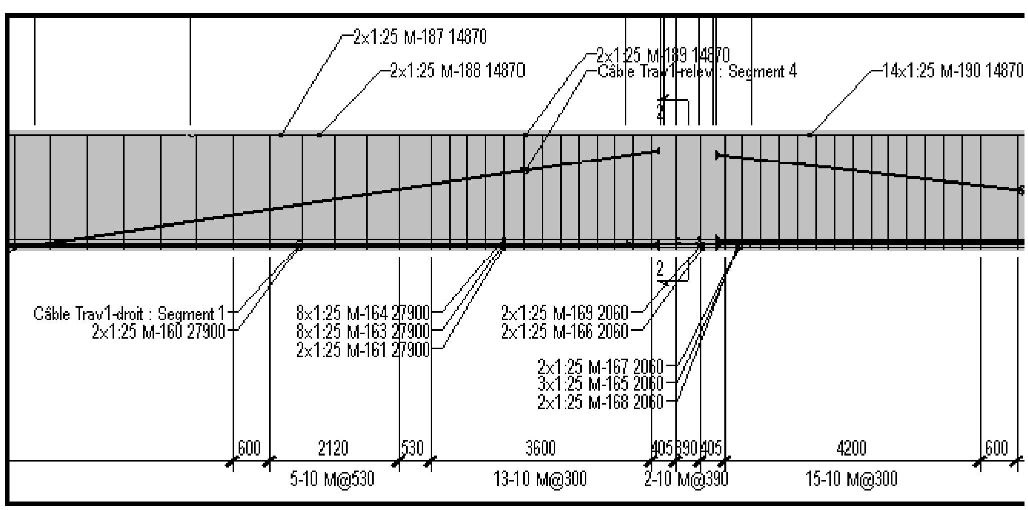




Stirrups:

The description of stirrups is as follows:

15 – 10M@ 300: 15 bars no. 10 (Metric), 300 mm centre to centre.



Export drawings in DXF (AutoCAD compatible):

Rebar placement drawings can be exported to AutoCAD using the **Export** function of **File** menu.

Display Forces and Resistance Diagrams

To verify the resistance of prestressed concrete elements and make sure that you are conforming to the code, display the beams forces and resistance diagrams through the **Rebar Placement** tab of the **View Options** dialog box.

The order of appearance of mostly used diagrams is as follows (from top to bottom):

Proportioning of Longitudinal Reinforcement (ΦAs Fy…vs. Mf/dv) according to General method (See clause 11.4.9.1, A23.3-94 standard);

1.2 Mcr vs. Mrx where Mrx must be greater than 1.2Mcr. If not so, then 1.33Mfx must be greater than Mrx. See below.

Vr vs. (Vf-ΦpVp) where ΦpVp is the shear forces of prestressing cables. You will notice that the stages within the diagram represent the part of shear forces taken by each cable group.

Mrx vs. (1.33Mfx & Mfx) must be verified with the 1.2Mcr vs. Mrx diagram.

Remark Select the **Colours** tab (**View Options)** to know the colours representing forces and resistances, within each of the displayed diagrams.

See also

Stresses at Construction Stages
Graphical Results
Loss of Prestress in Cables
General Results Spreadsheet

Editing Cables and Reinforcing Bars

Moving a Whole Cable Layout

The displacement of a cable in the **X** direction was already allowed through a cross-section. Now, the displacement of a whole cable layout can be done in the **Y** and **Z** direction through the elevation view of a continuous system. The **Cursor** mode must be activated before using the **Move** function and **Stretch** function.

Displacement in the Z direction:

To displace a cable layout in the Z direction: press the **Move** icon and press the **Ctrl** key while selecting the cable. Drag the mouse while you press down the left mouse button. Release the mouse button. Only cable layout ends can be displaced in the Z direction.

Displacement in the Y direction:

To displace a cable layout in the Y direction, press the **Move** icon and press the **Ctrl + Shift** keys while selecting the cable. Drag the mouse while you press down the left mouse button. Release the mouse button. Only cable layout ends and points with a null derivative (parabolic layout) can be displaced in the Y direction.

See also

Deforming a Cable Layout Segment
Modifying the Spacing between Cables
Properties
Delete
Activate Cursor mode
Move
Stretch

Deforming a Cable Layout Segment

A deformation can be applied on a cable in the Y and Z direction of continuous system by selecting points on a layout segment.

Z Deformation:

A Z deformation can be applied at the beginning and end of any segment composing a cable layout. To do so, press the **Move** icon and select a point on a segment. Drag the mouse while you press down the left mouse button. Release the mouse button.

Y Deformation:

A Y deformation can be applied at the beginning and end of any segment composing a cable layout and at the point of null derivative for parabolic segments. In the latter case, the derivative is preserved at the junction with other segments, except when moving maximum or minimum points of the parabola within the segment. When displacing straight or circular segments, VisualDesign will not consider any continuity in the derivative with contiguous segments. To move a point in the Y direction, activate the **Move** function and press the **Shift** key while selecting a point on a segment. Drag the mouse while you press down the left mouse button. Release the mouse button.

Modifying the Spacing between Cables

You are allowed to modify the spacing between cables in a cross-section, in the x direction. To do so, activate the **Cursor** mode and click on the **Stretch** function. Then, click on a cable layer. Drag the mouse while you press down the left mouse button. Release the mouse button.

Reminder - Editing Keys

Look at tables below to learn how to edit elements in the *Rebar Placement* window. Some editing is done with function **Stretch** or **Move** but some others must be done by pressing down a control key such as [Ctrl] or [Shift], and by clicking on either **Stretch** or **Move** function.

Editing Longitudinal Reinforcing Bars

Location	Action	Control Keys	Select Function
Cross-section	Move the selected bar layer towards direction x	N/a	Move
Cross-section	Modify the spacing between selected bars towards x-axis	N/a	Stretch
Cross-section	Move selected bars towards direction y	[Shift] (Optional) to move towards y without moving the x coordinate.	Move
Elevation view	Stretch selected bars towards direction z	N/a	Stretch
Elevation view	Move selected bars towards direction z	N/a	Move

Editing Stirrups

Location	Action	Control Keys	Select Function
Cross-section	Move selected stirrup towards direction x or y.	N/a	Move
Cross-section	Stretch selected stirrup leg towards direction x or y.	N/a	Stretch
Elevation view	Move the whole selected stirrup sequence along continuous system (z-direction)	N/a	Move
Elevation view	Modify the spacing of stirrup in the selected sequence (z-direction)	N/a	Stretch

Editing Prestressing Cables

Location	Action	Control Keys	Select Function
Cross-section	Move selected cable towards direction x	N/a	Move
Cross-section	Modify spacing between selected cables (x-direction)	N/a	Stretch
Elevation view	Move selected cable towards direction y	[Ctrl]+[Shift]	Move
Elevation view	Move selected cable towards direction z (along continuous system length)	[Ctrl]	Move
Elevation view	Create a cable deformation towards y-direction	[Shift]	Move
Elevation view	Create a cable deformation towards z-direction.	N/a	Move

Editing Cross-sections

Location	Action	Control Keys	Select Function
Cross-section	Selected the cross-section outline and move it towards y or z-direction. The line of cut on elevation view will not move.	N/a	Move
Line of cut on Elevation view	Move the line of cut along continuous system (z-direction). The cross-section will move along the line of cut.	N/a	Move

See also

Activate Cursor mode
Move
Stretch
Editing Cross-sections
Save Rebar Placement As (Copy Identical Continuous system)
Moving a Whole Cable Layout
Deforming a Cable Segment
Modifying Spacing between Cables

Graphical Results

Graphs - Prestressed Concrete Results

The "Graphs Results" icon of View Toolbar (Rebar Placement Window)

To verify your design, display the results in the form of graphs, according to a chosen construction stage. To do so, open the *Rebar Placement* Window and go to **Results/Graphs**.

For *Service* and *Construction Stage* Load Combinations, available graphs are:

Stresses in concrete:

Sigma and Position within continuous system

In the section ***Type of Graph***, activate the "Ratio of Stresses in Concrete" radio button. This will activate the "Ratio of Stresses in Concrete" zone. You can display the ratio of stresses in concrete for the following location in concrete element and according to a chosen construction stage:

- Top of Section (Sigma_Ss);
- Bottom of Section (Sigma_Si);
- Top of Slab (Sigma_Ds);
- Bottom of Slab (Sigma_Di);

On this graph, you will also find other results: The maximum tensile stress **Ft max** and maximum compressive stress **Fc max** in the slab and the maximum tension and compression forces in the whole element.

In the "Stage" drop-down list box, select a *Service* or *Construction Stage* load combination.

Ratio of Stresses in Cables:

Sigma/fpu and Position within continuous system

In the section ***Type of Graph***, activate the "Ratio of Stresses in Cables" radio button. This will activate the "Ratio of Stresses in Cables" zone. For a chosen construction stage, you can display the ratio of stresses for the following types of cable:

- Pre-tensioned cables (stage 3);
- Post-tensioned cables (stage 4);
- Composite post-tensioned cables (stage 7);

- External post-tensioned cables (stage 10).

In the "Stage" drop-down list box, select a *Service* or *Construction Stage* load combination.

Deflection under service:

Deflection v and Position within continuous system

For a chosen construction stage or service load combination, you can display the deflection with or without moving loads:

- Deflection at construction stage;
- Deflection at construction stage plus max and min deflection due to moving loads, if any.

N.B. The calculation of deflection due to creep and shrinkage is done according to the creep/shrinkage model chosen by the user. These effects are included into calculated results before and after each construction stage. (For the calculation of stresses and forces, values are factored according to load factors defined in the load combination. However, the deflection that is displayed in this graph is obtained considering 100% of creep/shrinkage effects.)

Bending Moments:

Mx and Position within continuous system

In the section ***Type of Graph***, activate the "Bending and Shear Forces" radio button. This will activate the "Bending and Shear Forces" zone. Tick off the box that corresponds to results that you want to visualize and for a given construction stage:

- Mx due to shrinkage and creep effects;
- Mx due to secondary prestressing

In the "Stage" drop-down list box, select a *Service* or *Construction Stage* load combination.

Shear Forces:

Vy and Position within continuous system

In the section ***Type of Graph***, activate the "Bending and Shear Forces" radio button. This will activate the "Bending and Shear Forces" zone. Tick off the box that corresponds to results that you want to visualize and for a given construction stage:

- Vy due to shrinkage and creep effects;
- Vy due to secondary prestressing;

In the "Stage" drop-down list box, select a *Service* or *Construction Stage* load combination.

Stresses due to Fatigue:

Variation of stresses and Position within continuous system

You must have defined at least one *Fatigue* load combination beforehand.

In the section ***Type of Graph***, activate the radio button corresponding to "Stresses due to Fatigue" and choose the construction stage in the drop-down list box.

Legend and other graph functionalities

Double-click the legend box to move it outside the diagram box.

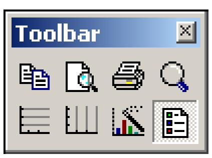
In a diagram, point your cursor on the curve and right click to obtain the point coordinates. They are displayed near your cursor.

Use the **Graphs** toolbar functions that are posted in every diagram box. To learn about these functions, see the topic Graphs toolbar.

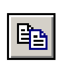
See also

Graphs Toolbar
Cables Prestressing Losses
Stresses at each Construction Stage

Graphs Toolbar (ChartFX)



A toolbar is provided in all the graphic results to help you managing diagrams (copy, print, view options, etc.). You will find below, a description of functions represented by icons and also a description of dialog boxes that can be called up through icons.

Copy to clipboard: 

Click on this icon to copy the diagram as a Bitmap, as a Metafile, as Text (data only) or as OLE Object.

Print Preview: 

Click on this icon to call up the *Page setup* dialog box. Complete the dialog box and look at the diagram. The table below explains the parameters included in the dialog box.

Field	Description
Options	
Separate Legend	Check this box to move the legend outside the diagram box.
Use Printer Resolution	Check this box to use the printer resolution.
Force Colors	If your printer is black and white, the printer driver will match the chart colors to a specific grayscale pattern.
Margins (inches)	Define the left, top, right and bottom margins, in inches.
Orientation	
Portrait	Check this box to look at the diagram in the vertical way.
Landscape	Check this box to look at the diagram in the horizontal way.
Use by default	Check this box to use the above parameters by default.

Print: 

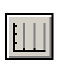
Press this icon to print the diagram. In the *Layout* tab, choose the "Landscape" orientation. This option must be selected even if you activated the "Landscape" orientation in the *Page Setup* dialog box (Print Preview function).

Zoom: 

Click on this icon and, with your cursor, draw a window around the elements that you wish to zoom in. The icon will still be activated to let you zoom again. To go back at a normal view, click on the icon again.

Horizontal grid: 

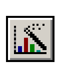
Click on this icon to display a horizontal grid.

Vertical grid: 

Click on this icon to display a vertical grid.

Series Legend: 

Click on this icon to mask or display the series legend of the diagram.

Properties: 

Click on this icon to modify the diagrams view options through the **Chart FX 98 Properties** dialog box. It is composed of the ***General***, ***Series***, ***Axes*** and ***3D*** tabs. Look at the tables below to learn more about each tab.

The *General* tab: This dialog box allows setting the displayed chart, such as *Major Unit Gridlines* and *Colors*.

Field	Description
Effects	
3D	Check this box to switch from 2D/3D views.
Cluster (Z-Axis)	When displaying a clustered chart, each series will have its own position in the z-axis. This means, if you have a 3 series chart, each data series will occupy one row of data and there will be 3 rows (z-axis clusters) in the chart.
Stacked	This function is not activated.
Axes styles	This option allows you to change the axis drawing style.
Grid lines	This option allows you to draw vertical and/or horizontal gridlines.
Colors	
Palette	This option allows you to change the entire chart palette. This will affect all elements in the chart.
Scheme	This option allows you to change the colour scheme for markers.
Background	This option allows you to set a colour for the background. This background is the box where the chart is enclosed.
Chart box	This option allows you to set a colour for the chart box. This is the background where the markers are enclosed.
Title	This option allows you to set a Top Title for the chart.

The *Series* tab: This dialog box allows you to set specific settings, including visual attributes for series in the chart.

Field	Description
Combo Box	When "All series" is selected in the Combo Box, settings will apply to all series in the chart and the property set is different if a specific series is selected. Therefore, 2 different screen shots are included for this particular dialog.
Marker fill	This option allows the user to change colours for series.
Lines	
Same color as markers	This option is activated if a specific series is selected. It will match the colour of lines to the one used for markers.
Custom	This option is activated if a specific series is selected. Change the line colour, style and weight for this specific series. Look at the sample that is displayed in the dialog.
Color	Choose a line colour for this specific series.
Style	Choose a line style for this specific series.
Weight	Choose a line weight for this specific series.
Markers	Markers represent points on the series.
Gallery:	This option allows you to change the chart type
Shape:	Choose a marker shape such as cylinders or cones for a *bar chart* or *cube chart*.
Show every:	Repetition of markers
Size:	Markers size
Show point labels	This option allows you to show/hide point markers on the series.
Visible	This option allows you to show/hide a particular series in the chart.
Sample	Look at the sample displayed in this box.

The *Axes* tab: This dialog allows manipulating axis settings, including *Major Interval*, *Minor Intervals*, *Scales*, *Min*, *Max*, and *Gridlines & Tickmarks*.

Field	Description
Combo Box	Choose the axis for which you want to look at parameters. To change parameters, press on the *Details…* button posted in the bottom of the dialog. Check the "Visible" box to show/hide a particular axis, including labels and tickmarks.
Major unit	This option allows you to set the Major interval for the selected axis. Check the "Show gridlines" box to show/hide gridlines on the major interval.
Tick mark type	Choose the tickmark type for the major interval.
Minor unit	This option allows you to set the Minor interval for the selected axis. . Check the "Show gridlines" box to show/hide gridlines on the minor interval.
Tick mark type	Choose the tickmark type for the minor interval
Details… Button	Press this button to access the **Axis Properties** dialog to change the selected axis properties. See explanation below.

The **Axis Properties** dialog box includes the selected axis properties and is composed of the following tabs: ***General***, ***Scale***, ***Labels*** and ***Grid lines***.

The *General* Tab:

Field	Description
Selected Axis	
Major unit	Choose the unit that will define the major interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of major interval.
Minor unit	Choose the unit that will define the minor interval of the selected axis.
Tick mark type	Select the tickmark type and show/hide gridlines of minor interval.

The *Scale* Tab: This dialog allows you to manipulate axis scales, including min, max, and logarithmic scales among others.

Field	Description
Selected Axis	
Minimum	This option allows you to set the minimum for the selected axis.
Maximum	This option allows you to set the maximum for the selected axis.
Scale unit	Set the scale unit for the selected axis.
Format	Set the axis format: None, Number, Currency, Scientific notation or Percentage.
Decimal places	Set the number of decimals for the selected axis.
Logarithmic scale	Set logarithmic or linear scales.
Zero line	Set the starting point at zero. For example, if you have a bar chart with a minimum value of –50, the starting point will be zero and you will have bars that go up and down, depending on their value.

The *Labels* Tab: This dialog allows you to manipulate labels displayed on the axis, including Rotation, Fonts and Axis Title.

Field	Description
Selected Axis	
Orientation	Use the *Degrees* arrows to set the rotated angle for labels in the selected axis.
Show labels	Check this box to show the labels in the selected axis.
2 Levels	Check this box to show the selected axis labels on two levels.
Rotate with chart	Check this box to rotate labels with chart.
Title	Set the title for the selected axis.

The *Grid lines* Tab: This dialog allows you to customize gridlines on both the major and minor intervals.

Field	Description
Selected axis	
Major gridlines	
Color:	Choose the colours for major gridlines in the selected axis.
Style:	Set the major gridline styles in the selected axis.
Weight:	Set the major gridline width in the selected axis
Minor gridlines	
Color:	Choose the colours for minor gridlines in the selected axis.
Style:	Set the minor gridline styles in the selected axis.
Weight:	Set the minor gridline width in the selected axis
Align with labels	Align labels with tickmarks and gridlines in the selected axis.
Interlaced	Set interlaced gridlines. The interlaced colours are achieved with the major grid lines colour and the background of the chart.

See also

Prestressed Concrete Results - Graphs

Numerical Results

Losses of Prestress in Cables

This results spreadsheet is part of the **Results** menu of *Rebar Placement* window. VisualDesign™ calculates prestressing losses in cables, at each construction stage and for all cable groups (according to the prestressing stage defined in the Cable Groups spreadsheet)

You will notice that ***VisualDesign™ also calculates prestressing gains in cables***. These gains are represented by positive values in the spreadsheet. Actually, depending on construction stages, some loads may add tension in the cables. The types of prestressing losses calculated by VisualDesign™ are: (For example, see Clause 18.6 of A23.3-94 Standard).

- **ES:** Elastic shortening of concrete;
- **REL**: Relaxation of tendon stress;
- **CR**: Creep;
- **SH**: Shrinkage;
- **Δpi**: Sum of losses for this stage;
- **Δp**: Sum of losses for all construction stages (last column of the spreadsheet).

Group: Results – Prestressed Concrete

Column	Description	Editing
ID	Automatically calculated	No
Number	Cable name or number (12 alphanumerical characters).	No
Stage	Prestressing Stage.	No
ESi	Elastic shortening of concrete at stage i.	No
RELi	Relaxation of tendon stress at stage i.	No
CRi	Creep of concrete at stage i.	No
SHi	Shrinkage of concrete at stage i.	No
Δpi	Sum of losses at stage i.	No
Δp	Sum of losses for all stages.	No

Stresses under Service Loads

This spreadsheet includes minimum and maximum stresses in the concrete, ratios of stresses in prestressing cables and deflections results at each stage, with and without moving loads. Results are available through the **Results / Stresses** menu of *Rebar Placement* window, for *Service* load combinations and *Construction Stage* load combinations.

Construction Stages:

Select **Results** / **Stresses** / **Stage X**.

Other Service Load Combinations:

Select **Results** / **Stresses** / **Serviceability Limits States**

Group: Prestressed Concrete Results

Column	Description	Editing
ID	Automatically calculated	No
Number	Cable number or name (12 alphanumerical characters)	No
Position	Position within continuous system.	No
Section Top σss min	Minimum stress in concrete at the section superior fibre, for a chosen construction stage.	No
Section Top σss max	Maximum stress in concrete at the section superior fibre, for a chosen construction stage.	No
Section Bottom σsi min	Minimum stress in concrete at the section inferior fibre, for a chosen construction stage.	No
Section Bottom σsi max	Maximum stress in concrete at the section inferior fibre, for a chosen construction stage.	No
Slab Top σds min	Minimum stress in concrete at the top of the slab, for a chosen construction stage.	No
Slab Top σds max	Maximum stress in concrete at the top of the slab, for a chosen construction stage.	No

Column	Description	Editing
Slab Bottom σdi min	Minimum stress in concrete at the bottom of the slab, for a chosen construction stage.	No
Slab Bottom σdi max	Maximum stress in concrete at the bottom of the slab, for a chosen construction stage.	No
Cable Initial Pre-tension σ/fpu min.	Ratio of minimum stresses in initial pre-tensioned cables.	No
Cable Initial Pre-tension σ/fpu max.	Ratio of maximum stresses in initial pre-tensioned cables.	No
Cable Initial Post-tension σ/fpu min.	Ratio of minimum stresses in initial post-tensioned cables.	No
Cable Initial Post-tension σ/fpu max.	Ratio of maximum stresses in initial post-tensioned cables.	No
Cable Composite Post-tension σ/fpu min.	Ratio of minimum stresses in composite post-tensioned cables.	No
Cable Composite Post-tension σ/fpu max.	Ratio of maximum stresses in composite post-tensioned cables.	No
Cable External Post-tension σ/fpu min.	Ratio of minimum stresses in external post-tensioned cables.	No
Cable External Post-tension σ/fpu max.	Ratio of maximum stresses in external post-tensioned cables.	No

Column	Description	Editing
Deflection Stage	Deflection at this construction stage.	No
Max Deflection Stage + Trucks	Maximum deflection at this stage, including moving loads.	No
Min Deflection Stage + Trucks	Minimum deflection at this stage, including moving loads.	No

See also

Graphs

Loss of Prestress in Cables

Intermediate Results

Intermediate results are available once that the Prestressed Concrete design is completed. Open the *Rebar Placement* window and go to **Results** menu.

The spreadsheet includes the following columns:

Group: Prestressed Concrete Results

Column	Description	Editing
Stage	Number of days for this construction stage.	No
Member number	Number of this member (16 alphanumerical characters).	No
Shape	Shape of this member.	No
f'ci Section	Compressive strength of concrete at time of prestress transfer, for the section.	No
f'ci Slab	Compressive strength of concrete at time of prestress transfer, for the slab.	No
Eci Section	Modulus of elasticity of concrete at time of prestress transfer, for the section.	No
Eci Slab	Modulus of elasticity of concrete at time of prestress transfer, for the slab.	No
Ix Section	Section inertia for strong axis.	No
Ix Composite	Transformed inertia of composite section for strong axis.	No

Column	Description	Editing
Area Section	Section Area	No
Area Composite	Area of transformed section.	No
ybi	Distance from bottom to centre of gravity of composite section.	No
yhi	Distance from top to centre of gravity of composite section.	No
Compression Limit Slab	Compressive strength limit for the slab.	No
Compression Limit Section	Compressive strength limit for the section.	No
Tension Limit Slab	Tension strength limit for the slab.	No
Tension Limit Section	Tension strength limit for the section.	No
Creep Slab	Shrinkage differential factor for the slab between current and previous construction stage.	No
Creep Section	Shrinkage differential factor for the section between current and previous construction stage.	No
Shrinkage Slab	Differential of concrete shrinkage for the slab between current and previous construction stage.	No
Shrinkage Section	Differential of concrete shrinkage for the section between current and previous construction stage.	No

See also

Graphs (Results)
General Results
Loss of Prestress
Ratio of stresses in concrete and cables

General Results spreadsheet

This spreadsheet is available once that the reinforced concrete design is completed. It includes many results that you can display in a graphic form through the view options of *Rebar Placement* window.

Please verify that ratio **c/d** does not exceed **c/d max** (clause10.5.2 of Code A23.3-95), which is given as a result in this spreadsheet.

If you used the ***Bridge Evaluation*** module, you will also find results that are about this evaluation.

Please note that some results will be available in the spreadsheet only if they are displayed in the *Rebar Placement* window. This is the case for results on cracking factors (β2 or z) and fatigue in longitudinal rebars (*Variation of stresses in rebars* diagram). Do not forget that, if you want to obtain these results, you must define load combinations with a *Service* or *Fatigue* status.

Lines that are marked with yellow mean that there are some parameters exceeding the limits permitted per code.

The **General Results** spreadsheet in split into five spreadsheets to make consultation easier:

Positive Bending Moment tab
Negative Bending Moment tab
Shear Force tab
Axial Force tab
Position of Cables

Positive Bending Moment tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Mfx max	Envelope of maximum bending moment for strong axis.	No
R'	**Factored resistance** of tensioned reinforcement, considering stresses that can be developed, according to clause 11.4.9.1 of Code A23.3:	No
F'	**Forces** in tensioned reinforcement, considering stresses that can be developed, according to equation 11.4.9.1 of Code A23.3:	No

Column	Description	Editing
Mrx	Factored moment resistance.	No
Mnx	Nominal moment (Refer to Ch. 21 of A23.3-95 standard)	No
Mpx	Probable moment (Refer to Ch. 21 of A23.3-95 standard)	No
Design load	Bending moment at this location within continuous system.	No
Bridge Evaluation module only		
F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to flexural resistance.	No
Mcr	Bending moment when a tensile stress of fcr is induced in the concrete.	No
fcr	Cracking strength of concrete.	No
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system. N.B. VisualDesign considers all tensioned rebars, even those that have not attained the yield stress fy. (The value of d that appears here does not correspond to the value of d used in the code equations, except at the beginning of continuous system)	No
dv	Distance between tension and compression resultant forces due to bending, at this location within continuous system.	No
bw	Width of concrete section web at this location within continuous system. For prestressed concrete elements: this width is reduced by total sheath diameter if there is no grout or 1/2 sheath diameter if it is grouted.	No
As	Area of reinforcing steel in tension. (VisualDesign considers all rebars in tension, even those that have not reached fy). However, when considering the strain – deformation compatibility in calculations, the useful area of each bar in tension is factored with this ratio: fs/fy.	No

Column	Description	Editing
ϱ	Percentage of steel reinforcement at this location within continuous system.	No
For a Prestressed Concrete Project		
ωp	Ratio of reinforcing bars and prestressing reinforcement according to clause 9.6.8.5.1 of Code S6-88. See the note below.	No
Φp*fps*Aps	Factored tensile strength of prestressing cables.	No
εx	Strain in longitudinal reinforcement used for calculating the shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that considers the shear resistance of cracked sections.	No
c/d	c/d ratio of clause 10.1.4 - Code A23.3, relative to the balanced condition *Concrete Deformation and yield strength of steel*. Compare with c/d max below.	No
c/d max	Maximum c/d ratio.	No
For load combinations with a "Service" status. β2 : S6-00, Cl. 8.12.3 z : A23.3-95, Cl. 10.6.1		
β2 or z	Function modified by the proportioning and type of longitudinal rebars.	No
β2 max or z max	Value limiting the proportion of longitudinal rebars.	No
Fcr	S6-00 only: Factor that controls cracking in buried structures as per clause 7.8.9.1. This column is present if a "Buried" type of structure was select in the concrete specification.	No
Mw	S6-00 only: Moment at a section where a tensile stress of 0.4 fcr is induced in the concrete.	No
Ms	S6-00 only: Flexural moment at a section under consideration at the SLS load.	No
fs	Tensile stress in reinforcing bars, considering stress-strain compatibility.	No
I eff	Effective moment of inertia.	No

Column	Description	Editing
I cr	Moment of inertia of the cracked section, transformed to concrete.	No

<u>Explanatory Note on Variable ωp: (See clause 9.6.8.5.1 of Code S6-88)</u>

ωp represents the verification of steel and prestressing reinforcement ratio used in the calculation of the factored resistance of a rectangular section:

$$\frac{(\rho_p \cdot f_{ps})}{f'_c} + \frac{(\rho \cdot f_y)}{f'_c} - \frac{(\rho' \cdot f_y)}{f'_c} \leq 0.3$$

Where

ϱp = Ratio of prestressing reinforcement;
fps = Calculated stress in prestressing reinforcement for Ultimate Limit States (MPa);
fy = Yield strength of reinforcement steel (MPa);
f'c = Specified compressive strength of concrete at 28 days (MPa);
ϱ = Ratio of steel reinforcement in tension;
ϱ' = Ratio of reinforcement steel in compression.

See Also

General Results spreadsheet
Negative Bending Moment tab
Shear Force tab
Axial Force tab
Cable Positions tab

Negative Bending Moment tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Mfx min	Envelope of minimum bending moment for strong axis.	No
R'	**Factored resistance** of tensioned reinforcement, considering stresses that can be developed, according to clause 11.4.9.1 of Code A23.3:	No

Column	Description	Editing
F'	**Forces** in tensioned reinforcement, considering stresses that can be developed, according to equation 11.4.9.1 of Code A23.3:	No
Mrx	Factored moment resistance.	No
Design load	Bending moment at this location within continuous system.	No
Bridge Evaluation module only		
F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to flexural resistance.	No
Mcr	Moment at a section where a tensile stress of fcr is induced in the concrete.	No
fcr	Cracking strength of concrete.	No
Mn	Nominal moment nominal (Refer to Ch. 21 of A23.3-95 standard)	No
Mp	Probable moment (Refer to Ch. 21 of A23.3-95 standard)	No
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system. N.B. VisualDesign considers all tensioned rebars, even those that have not attained the yield stress fy. (The value of d that appears here does not correspond to the value of d used in the code equations, except at the beginning of continuous system)	No
dv	Distance between tension and compression resultant forces due to bending, at this location within continuous system.	No
bw	Width of concrete section web at this location within continuous system. For prestressed concrete elements: this width is reduced by total sheath diameter if there is no grout or 1/2 sheath diameter if it is grouted.	No

Column	Description	Editing
As	Area of reinforcing steel in tension. (VisualDesign considers all rebars in tension, even those that have not reached fy). However, when considering the strain – deformation compatibility in calculations, the useful area of each bar in tension is factored with this ratio: fs/fy.	No
ϱ	Percentage of steel reinforcement at this location within continuous system.	No
For a Prestressed Concrete Project		
ωp	Ratio of reinforcing bars and prestressing reinforcement according to clause 9.6.8.5.1 of Code S6-88. See the note below.	No
Φp*fps*Aps	Factored tensile strength of prestressing cables.	No
εx	Strain in longitudinal reinforcement for the calculation of shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that takes into account the shear resistance of cracked sections.	No
c/d	c/d ratio of clause 10.1.4 - Code A23.3, relative to the balanced condition *Concrete Deformation and yield strength of steel*. Compare with c/d max below.	No
c/d max	Maximum c/d ratio.	No
For load combination with a "Service" status. β2 : S6-00, Cl. 8.12.3 z : A23.3-95, Cl. 10.6.1		
β2 or z	Function modified by the proportioning and type of longitudinal rebars.	No
β2 max or z max	Value limiting the proportion of longitudinal rebars.	No
Fcr	S6-00 only: Factor that controls cracking in buried structures as per clause 7.8.9.1. This column is present if a "Buried" type of structure was select in the concrete specification.	No
Mw	S6-00 only: Moment at a section where a tensile stress of 0.4 fcr is induced in the concrete.	No

Column	Description	Editing
Ms	S6-00 only: Flexural moment at a section under consideration at the SLS load.	No
fs	S6-00 only: tensile stress in reinforcing bars.	No
I eff	Effective moment of inertia.	No
I cr	Moment of inertia of the cracked section, transformed to concrete.	No

See Also

General Results spreadsheet
Positive Bending Moment tab
Shear Force tab
Axial Force tab
Cable Positions tab

Explanatory Note on Variable ωp: (See clause 9.6.8.5.1 of Code S6-88)

ωp represents the verification of steel and prestressing reinforcement ratio used in the calculation of the factored resistance of a rectangular section:

$$\frac{(\rho_p \cdot f_{ps})}{f'_c} + \frac{(\rho \cdot f_y)}{f'_c} - \frac{(\rho' \cdot f_y)}{f'_c} \leq 0.3$$

Where

ϱp = Ratio of prestressing reinforcement;
fps = Calculated stress in prestressing reinforcement for Ultimate Limit States (MPa);
fy = Yield strength of reinforcement steel (MPa);
f'c = Specified compressive strength of concrete at 28 days (MPa);
ϱ = Ratio of steel reinforcement in tension;
ϱ' = Ratio of reinforcement steel in compression.

Shear Force tab

For a Beam type of continuous system, you will find the following results:

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Vfy' max	Envelope of maximum shear (strong axis), Vfy max, minus the shear carried on by prestressing cables, ϕpVp.	No
Vfy' min	Envelope of minimum shear (strong axis), Vfy min, minus the shear carried on by prestressing cables (ϕpVp).	No
Design load for Shear	Design load for shear at this location within continuous system.	No
εx	Strain in longitudinal reinforcement for the calculation of shear resistance on strong axis.	No
θ	Angle of inclination between diagonal stains in compression and the element longitudinal axis.	No
β	Factor that takes into account the shear resistance of cracked sections.	No
Vc	Concrete shear resistance.	No
Vs	Shear resistance of steel reinforcement	No
Vr'	Shear resistance Vr of the section at this location within continuous system minus the shear carried on by prestressing cables (ϕpVp).	No
d	Distance from extreme fibre in compression to the centre of gravity of the tensioned reinforcement, at this location within continuous system.	No
dv	Distance between extreme fibre in compression and centre of gravity of bending reinforcement. VisualDesign uses the smallest calculated value, if it is relevant.	No

Column	Description	Editing
Prestressed Concrete only		
ϕpVp	Shear strength of prestressing cables.	No
Bridge Evaluation only		
F	Live load Capacity Factor (Code S6-00).	No
U*Xi	Reduction factor applied to shear resistance.	No

Axial Force tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in Project Configuration.	No
Nfz max	Envelope of maximum axial force	No
Mfx max	Envelope of maximum bending moment on strong axis.	No
Nfz min	Envelope of minimum axial force	No
Mfx min	Envelope of minimum bending moment on strong axis.	No
Maximum Mf-Nf Design Load	Maximum design load due to combined bending & forces for a continuous system with variable geometry of the *Slab* type.	No
Minimum Mf-Nf Design Load	Minimum design load due to combined bending & forces for a continuous system with variable geometry of the *Slab* type.	No

Position of Cables tab

Group: Concrete Results

Column	Description	Editing
Member Number	The number of the member that is part of this continuous system.	No
z	Subdivisions of continuous system as specified in the Analysis tab of Project Configuration.	No
Name of segment i y	Position of cable segment "i" according to continuous system local y-axis.	No
Name of segment i Slope	Slope of cable segment "i" at this position along continuous system.	No
Name of segment i Area	Area of cables in segment "i", at this position along continuous system.	No
Name of segment n y	Position of cable segment "n" according to continuous system local y-axis.	No
Name of segment n Slope	Slope of cable segment "n" at this position along continuous system.	No
Name of segment n Area	Area of cables in segment "n", at this position along continuous system.	No

See also

Cable Layouts spreadsheet

Chapter 15

BRIDGE EVALUATION

TABLE OF CONTENTS

Chapter 15 Bridge Evaluation

General

VisualDesign™ Approach for Bridge Evaluation

Modification of UϕR values

The engineer can specify up to nine factor of reduction that will be apply to the calculated value of UϕR, for each member of his structure. These nine factors have been defined to modify the following values:

- m_fRedMrxPos * UϕMrx (positive bending – strong axis)
- m_fRedMrxNeg * UϕMrx (negative bending – strong axis)
- m_fRedMryPos * UϕMry (positive bending – weak axis)
- m_fRedMryNeg * UϕMry (negative bending – weak axis)
- m_fRedMrz * UϕMrz (torsion)
- m_fRedCrz * UϕCrz (compression)
- m_fRedTrz * UϕTrz (tension)
- m_fRedVrx * UϕVrx (shear – weak axis)
- m_fRedVry * UϕVry (shear – strong axis)

In a general manner, we call these reduction factors "Xi" and the new calculated resistance is called "XiUϕR". Reduction factors can be used in evaluating steel, reinforced concrete or prestressed concrete bridges. Pre-tensioned members can also be reduced in capacity.

In the **Evaluation** spreadsheet, the field "System" and "Element" allows identifying the system and element behaviours. These fields will not be used when calculating factor F (live load capacity factor) but they are available for sorting into the spreadsheet (for example, if you have elements of different categories).

Deterioration of rebars and prestressing cables

Cables, main reinforcement and stirrups can have their capacity reduced by a deterioration factor.

See

Deterioration of rebars and cables
Deterioration of members

Dead Load Categories

Many types of dead loads are already defined in VisualDesign for S6-00 standard:

- Prefab Components (equivalent to D1)
- Cast-in-place (equivalent to D2)
- Wearing surface (equivalent to D3)
- Etc.

VisualDesign™ will apply αD load factors according to these dead load categories.

αD and αL Values

From the **Specific Options** page of the **Load Combination Generation Wizard**, the user can generate load combinations for bridge evaluation by calling up a dialog box. He has to define parameters that will be used by VisualDesign™ to calculate the *Beta Reliability Index* and load factors αD and αL.

Load factors αD and αL are applicable to all elements of the structure for database management and architectural reasons. If categories of system behaviour and element behaviour vary, the engineer will have to run extra analyses of the structure.

Specifications

Existing specifications in VisualDesign™ can be used or new ones can be created. In the "Type of analysis" column of the **Specifications** spreadsheet, the user will choose the *Bridge Evaluation* option. This option will activate the reduction of capacities ϕR by U value and deterioration factors.

Diaphragm action

Rigid floors should not be used to model a bridge, for the same reason explained above. Furthermore, the diaphragm effect will act opposite to temperature effects and finally, prestressing effects will be absorbed within the diaphragm.

See also

Specifications for Bridge Evaluation
Live Load Capacity Factor
Bridge Evaluation Results
Load Factors for Bridge Evaluation
The Member Dialog Box
Steel Design Results
General Results spreadsheet
Reinforced Concrete Design Module

Deterioration

Deterioration of Rebars and Cables

According to Code S6-00, deterioration factors can be applied to prestressing cables, main reinforcing bars and stirrups. These factors are calculated as follows:

Deterioration of Prestressing Cables

Cable area will be reduced according to the specified percentage of deterioration specified in the **Cable Layout spreadsheet**. This deterioration will be applied on a composite section on a long-term period and on a statically indeterminate structure according to the end conditions that have been specified by the user.

Deterioration of Main Reinforcing Bars

The calculation of capacities Crz, Mrx and Mry will be done with reduced effective area of rebars according to the specified deterioration in the **Longitudinal Rebars spreadsheet**.

Deterioration of Transverse Reinforcing Bars (strong axis)

When calculating Av, the calculation of capacity Vry will be modified by considering a number of planes that will be reduced by the deterioration factor (Av = Nos. Plane * [1-deterioration] Area of rebar) specified in the **Transverse Rebars spreadsheet**.

Deterioration of Transverse Reinforcing Bars (weak axis)

When calculating Av, the calculation of capacity Vrx will be modified by considering a number of planes that will be reduced by the deterioration factor (Av = Nos. Plane * [1-deterioration] Area of rebar) specified in the **Transverse Rebars spreadsheet**.

Deterioration of Members

If members are deteriorated due to corrosion or other, enter a deterioration factor (or percentage) in the **Evaluation** tab of **Member Characteristics** dialog box. The calculated capacities will be multiplied by these factors. A factor of 1.0 or a percentage of 100% indicates that there is no deterioration. Refer to the **Evaluation** tab. See the **Units** tab of the **Project Configuration** dialog box.

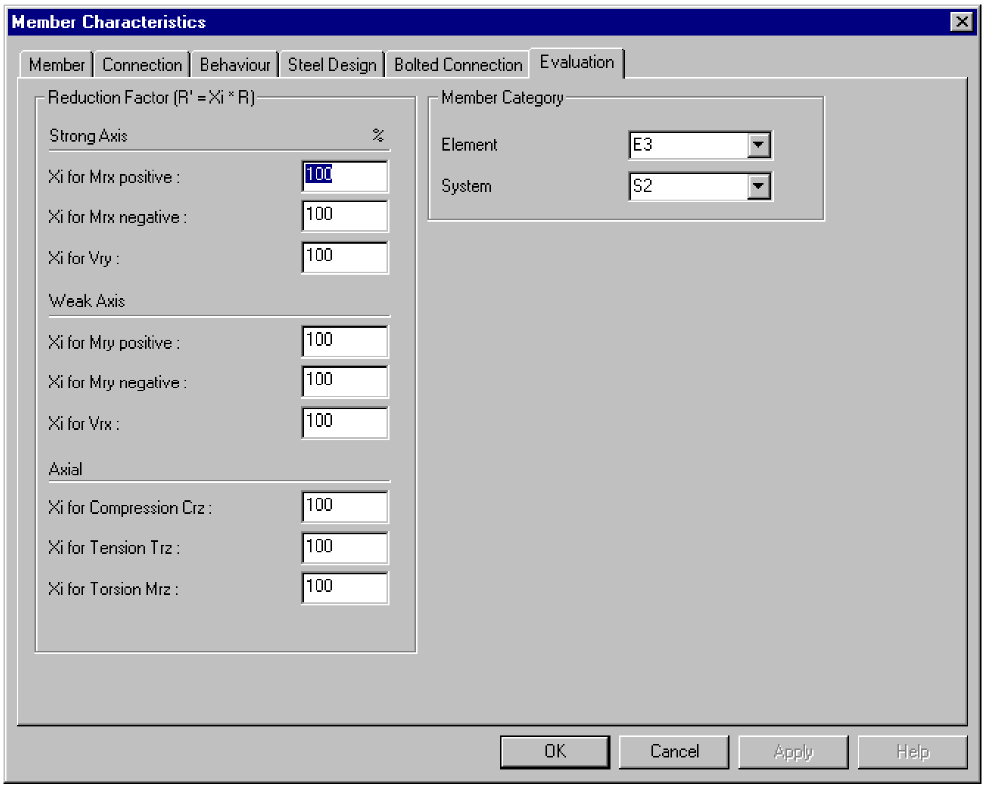
Particular case for steel structure

Deterioration factor cannot be applied to steel members. The engineer will have to create new sections that may have different flanges or web thickness.

Members and Specifications

The Evaluation tab

This tab is part of the **Member Characteristics** dialog box and includes reduction factors to be applied to members, along with the definition of member category (Element and System).



If members are deteriorated due to corrosion or other, enter a deterioration factor or percentage for reducing the calculated resistances listed below.

Capacities are multiplied by these factors. (A factor of 1.0 or a percentage of 100% indicates that there is no deterioration of the member.

Field	Description
Reduction factor	**R' = Xi * R**
Strong Axis	
Xi for Mrx positive	Enter a reduction factor or percentage to reduce the positive bending resistance according to strong axis, due to the deterioration of the member.

Field	Description
Xi for Mrx negative	Enter a reduction factor or percentage to reduce the negative bending resistance according to strong axis, due to the deterioration of the member.
Xi for Vry	Enter a reduction factor or percentage to reduce the member shear capacity, according to strong axis, due to its deterioration.
Weak Axis	
Xi for Mry positive	Enter a reduction factor or percentage to reduce the positive bending resistance according to weak, due to the deterioration of the member.
Xi for Mry negative	Enter a reduction factor or percentage to reduce the negative bending resistance according to weak axis, due to the deterioration of the member.
Xi for Vrx	Enter a reduction factor or percentage to reduce the member shear capacity, according to weak axis, due to its deterioration.
Axial	
Xi for Crz	Enter a reduction factor or percentage to reduce the member capacity in compression, due to its deterioration.
Xi for Trz	Enter a reduction factor or percentage to reduce the member capacity in tension, due to its deterioration.
Xi for Mrz	Enter a reduction factor or percentage to reduce the torsion resistance, due to the deterioration of the member.
Member Category	
Element	Category of element behaviour according to Code S6-00, for Bridge Evaluation.
System	Category of system behaviour according to Code S6-00, for Bridge Evaluation.

Evaluation spreadsheet

This spreadsheet is part of the **Member** multi-spreadsheets. Use the spreadsheet to sort data or modify values using the contextual menu.

The "Element" and "System" columns correspond to behaviour categories and are part of this spreadsheet for sorting purpose only. These categories of behaviour are defined in the Generation of Load Combinations for Bridge Evaluation dialog box.

Group: Structural Data

Column	Description	Editing
ID	Automatically calculated	No
Number	12 alphanumerical characters	Single click
Reduction Mrx pos.	Enter a reduction factor or percentage to reduce the positive bending resistance on strong axis, due to the deterioration of the member.	Single click
Reduction Mrx neg.	Enter a reduction factor or percentage to reduce the negative bending resistance on strong axis, due to the deterioration of the member.	Single click
Reduction Mry pos.	Enter a reduction factor or percentage to reduce the positive bending resistance on weak axis, due to the deterioration of the member.	Single click
Reduction Mry neg.	Enter a reduction factor or percentage to reduce the negative bending resistance on weak axis, due to the deterioration of the member.	Single click
Reduction Mrz	Enter a reduction factor or percentage to reduce the torsion resistance, due to the deterioration of the member.	Single click
Reduction Crz	Enter a reduction factor or percentage to reduce the member capacity in compression, due to its deterioration.	Single click
Reduction Trz	Enter a reduction factor or percentage to reduce the member capacity in tension, due to its deterioration.	Single click
Reduction Vrx	Enter a reduction factor or percentage to reduce the member shear capacity, on weak axis, due to its deterioration.	Single click
Reduction Vry	Enter a reduction factor or percentage to reduce the member shear capacity, on strong axis, due to its deterioration.	Single click
Element	Category of the element behaviour according to Code S6-00, for Bridge Evaluation.	Double-click
System	Category of the system behaviour according to Code S6-00, for Bridge Evaluation.	Double-click

Note. The "Element" and "System" correspond to behaviour categories and are part of this spreadsheet for sorting purpose only. These categories of behaviour are defined in the Generation of Load Combinations for Bridge Evaluation dialog box.

Specifications for Bridge Evaluation

Existing specifications in VisualDesign™ can be used or new ones can be created. In the "Type of analysis" column of the **Specifications** spreadsheet, the user will choose the *Bridge Evaluation* option. This option will activate the reduction of capacities ϕR by U value and deterioration factors.

Steel Specifications

- Select the **Steel Specifications** spreadsheet (**Structure / Specifications / Steel**) and choose *CAN/CSA-S6-00* as the code to be used for the evaluation. Then, in the "Type of Analysis" column, select *Bridge Evaluation* among the list box. Specify other parameters to be used for the evaluation.
- Assign this specification to all members composing the bridge.

Concrete Specifications

- Select the **Concrete Specifications** spreadsheet (**Structure / Specifications / Concrete**) and choose CAN/CSA-S6-00 as the code to be used for the evaluation. Then, in the "Type of Analysis" column, select *Bridge Evaluation* among the list box. Specify other parameters to be used for the evaluation.
- Assign this specification to all continuous systems through the **Continuous Systems** spreadsheet located in the **Structure** menu. Don't forget to specify concrete covers in this spreadsheet because it is not automatically initialized when Code S6-00 is selected. Refer to this Standard for appropriate concrete covers.

See also

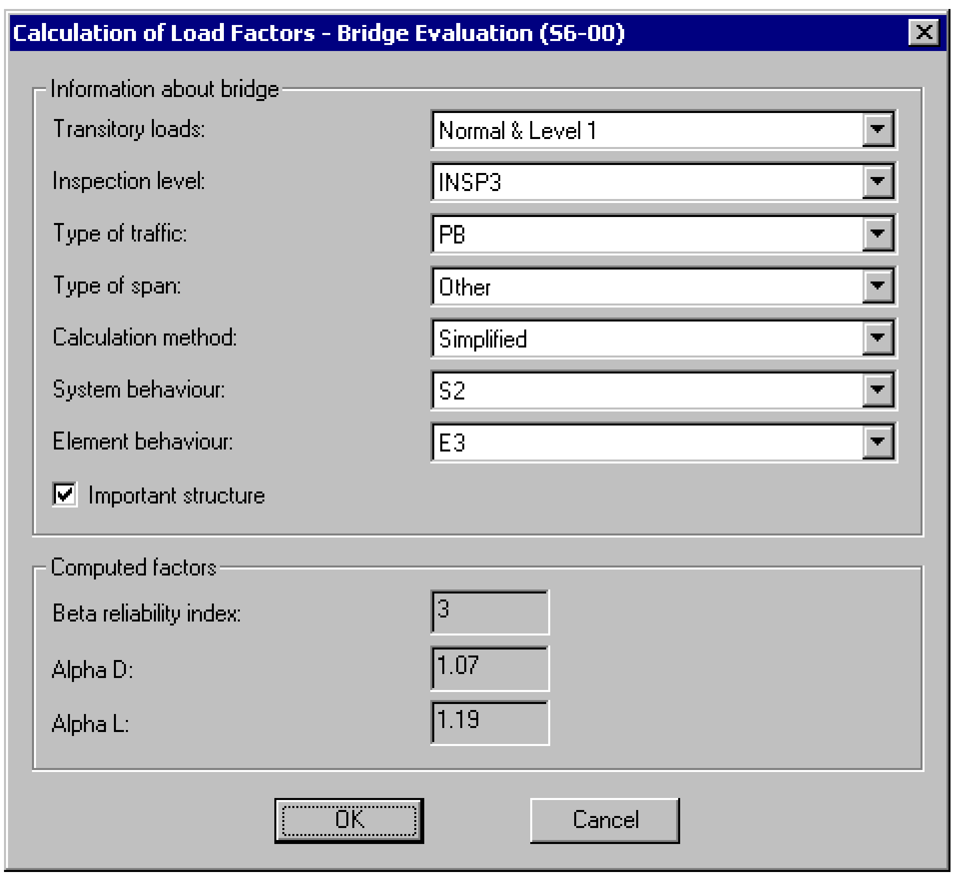
Load Factors for Bridge Evaluation
Steel Specifications
Concrete Specifications
Continuous Systems
Calculation of F (Live load capacity factor)

Load Combinations for Bridge Evaluation

Load Combination Generator

If you are evaluating a bridge according to code S6-00, the **Load Combinations Generator** generates required load combinations and appropriate load factors related to bridge evaluation.

- Go to **Loads / Load Combinations / Automatic Generation**.
- Select Code S6-00 in the first page (**General Options)** of the **Generator** and press the "Next" button.
- In the second page (**Specific Options)**, click the "Bridge Evaluation" button to call up the **Calculation of Load Factors** dialog box.



- Enter the required parameters in this specific dialog box. The Beta reliability index will be calculated, along with αD and αL load factors.

The table below describes the parameters included in the dialog box:

Field	Description
Information on the bridge	
Transitory loads	Specify the type of transitory loads applied to the bridge: Alternative, Level 1, Level 2 or Level 3.
Inspection Level	Choose among INSP1, INSP2 or INSP3.
Traffic Type	Choose among Normal, PA, PB, PC or PS.
Type of span	Specify the type of span: Short or Other.
Calculation Method	Choose a calculation method: Statically determinate, Sophisticated or Simplified.
System Behaviour	Specify the category S1, S2 or S3 that describes the system behaviour.
Element Behaviour	Specify the category E1, E2 or E3 that describes the element behaviour.
Important Structure	Check this option if the bridge is considered as an important structure according to code S6-00.
Calculated Load Factors	
Beta Reliability Index	According to the parameters that you entered, VisualDesign™ calculates the beta reliability index and load factors alpha D and alpha L.
Alpha D	Calculated load factor that will be applied to permanent loads.
Alpha L.	Calculated load factor that will be applied to live loads.

Live Load Capacity Factors

Calculation of Factor F

All ϕR values indicated in VisualDesign™ will be replaced by XiUϕR, which represents the resistance of a part that is modified by an adjusting factor U and by another factor, Xi, that reduces the capacity according to the engineer judgment (factor Xi may be equal to 1.0).

A limit state Live Load Capacity Factor "F" will be calculated (clause 14.14.2) as follows:

$$F = \frac{XiU\phi R - \sum \alpha_D D - \sum \alpha_A A}{\alpha_L L(1-I)}$$

αi load factors are those specified for load combinations ULSL no 1 to 9.

Calculation of F for Steel Members

We use the following equation for considering interaction equations for steel members:

$$F = \frac{XiU\phi R - \sum \alpha_D D - \sum \alpha_A A}{\alpha_L L(1-I)}$$

We say:

= XiUϕR

IntL = Interaction due to factored live loads (αLL(1+I));

= CL/CR + aMLX/MRX + bMLY/MRY

IntF = Interaction due to ELUL load combination;

= CF/CR + aMFX/MRX + bMFY/MRY

IntAD = Interaction due to types A and D loads;

IntAD = IntF - IntL

Therefore, the calculation of F will be:

F = (1.0 - IntAD)/ IntL

Calculation of F for Concrete Members

The calculation of factor F will also be done that way for Vrx, Vry and Mrx, for a given continuous system. Values ϕR for columns interaction curves (2D and 3D) will be replaced by $XiU\phi R$ values. For the moment, no factor F is applied to column.

Calculation of F for Prestressed Concrete Members

For prestressed concrete elements, the factor F will be calculated for compression and tension stresses in concrete (long term only).

$$F = \frac{\sigma_{SLS} - \sum \alpha_D \sigma_D - \sum \alpha_A \sigma_A}{\alpha_L \sigma_L (1+I)}$$

αi Load factors are those specified for the load combination ULST no.1.

σSLS value for concrete in tension is $0.4\lambda\sqrt{f'c}$.

σSLS value for concrete in compression is $0.45f'c$.

Note: Calculation of "U" for sections having passive and prestressing reinforcement:

Value ωp ($=\rho p\ fps\ /\ f'c$), which is used to determine U, is replaced by:

$\omega p = (\rho p\ fps\ /\ f'c + \rho fy\ /\ f'c - \rho' fy\ /\ f'c)$

Note: Calculation of bw to be used in the calculation of ρ for steel:

For rectangular, T and L shapes, clause 9.6.8.5.1 of Code S6-88 is considered. For composite NEBT and AASHTO sections, VisualDesign™ uses the smallest compressed width that it found when calculating Mr.

Note: Calculation of "d" used for the calculation of ρ for combined steel and prestressing reinforcement:

When calculating the value of Mr for sections having steel and prestressing reinforcement, VisualDesign™ calculated the value of "di" for each cable and rebar. Simultaneously, it calculates the sum of moments with terms di*Ai*fyi (Ai and fyi will have the appropriate value according to the associated deformation and type of material). From this sum, and simultaneously calculating the sum of terms Ai*fyi, we will calculate a "d" value that represent the presence of steel reinforcement and cables.

To get an accurate value for dv, the same will be done for the calculation of the concrete element centre of gravity.

Calculation of F considering the proportioning of longitudinal reinforcement

Clause 8.9.3.10.1 concerns the proportioning of longitudinal reinforcement and replaced the former equation Mr > Mf. For the bend part of the element, we suggest the following for the calculation of F:

dv' = dv theoretical dv calculated from stress/strain compatibility

dv = dv according to Code (minimum of 0.9d, 0.72h or dv')

cotθ: always a function of Mf, Nf and Vf

R' = XiU(ϕsAsfy + ϕpApsfps) = XiU Mrx/dv'

F' = Mf/dv + 0.5Nf + (Vf – 0.5Vs - ϕpVp)cotθ

F' = Mf/dv + (0.5Nf + Vf *cotθ) – (0.5Vs* + ϕpVp) cotθ

Clause 8.9.3.10.1 becomes:

R' >= F'

Where F' is:

F' = AD' + L' – SP'

With:

SP' = (0.5Vs* + ϕpVp)cotθ

AD' = Mad/dv + (0.5Nad + Vad *cotθ)

L' = Ml/dv + (0.5Nl + Vl *cotθ)

With:

Ml = αL(1+I)*ML (same principle for Nl and Vl)

$Mad = \Sigma \alpha_A M_A + \Sigma \alpha_D M_D$ (The same applies to Nad and Vad)

Finally, the F value for bending will becomes:

F = (R' + SP' – AD')/L' = (R' – F' + L')/L'

Bridge Evaluation Results

Steel Design

The **Steel Design Results** spreadsheet, available in the **Results/Structure Design** menu of VisualDesign main window, includes the Live Load Capacity factor F that governed the evaluation of each member. Members can be composite (steel/concrete) or standard.

Reinforced and Prestressed Concrete Design

The **General Results** spreadsheet, available in the **Results** menu of VisualDesign *Rebar Placement* window, supplies the Live Load Capacity Factor, F, and reduction factor (U*Xi) for each concrete member composing the analysed continuous system.

The calculated Live load Capacity Factor, F, and reduction factor (U*Xi) are indicated in the **Positive Bending Moment** tab, **Negative Bending Moment** tab and the **Shear Force** tab.

Chapter

16

CULVERT GENERATION AND CONCRETE DESIGN

TABLE OF CONTENTS

Chapter 16 Culvert Generation and Design

General

Culvert Generation Module

This module is a culvert generator (structural model). To design culverts, you need the *Moving Load Analysis* module and the *Reinforced Concrete Design* module.

You must be acquainted with VisualDesign. To help you, use the Online Help ([F1] key or **Help** menu) or read appropriate chapters in the User's Manual, such as *Basic Principles*, *Moving Load Analysis,* and *Reinforced Concrete Design*.

Use VisualDesign as a Generator

VisualDesign generates the structure, applies design criteria, and creates specifications. A 2D unitary moving load case is also defined and is specific to 2D culvert model.

The following types of culverts can be generated:

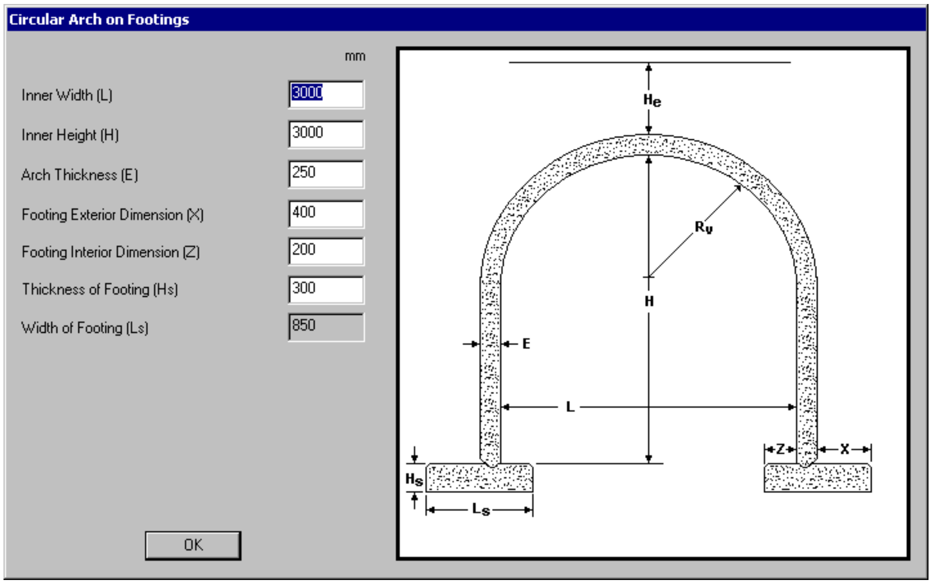
- Circular arch
- Elliptical arch
- Arch with 2 diagonals (equal groins)
- Arch with 3 diagonals (equal groins)
- Depressed arch with unequal groins
- Arch with 2 ellipses, with or without flanges
- Surbased arch with 3 radiuses
- Ogival arch with a slab
- Rigid frame on footings
- Rigid box culvert
- Rigid box culvert with haunches
- Elastic box culvert
- Elastic box culvert with haunches
- Open frame on footings
- Open frame with haunches on footings

What you have to do

All you have to do is to select a type of culvert among a list, enter dimensions and other data. When the model is generated, use the **Load Combination Generator** to generate the required load combinations as per CAN/CSA-S6-00 Standard. Include moving load envelopes into the generation. Launch the design. Each of these steps will be detailed further in this document.

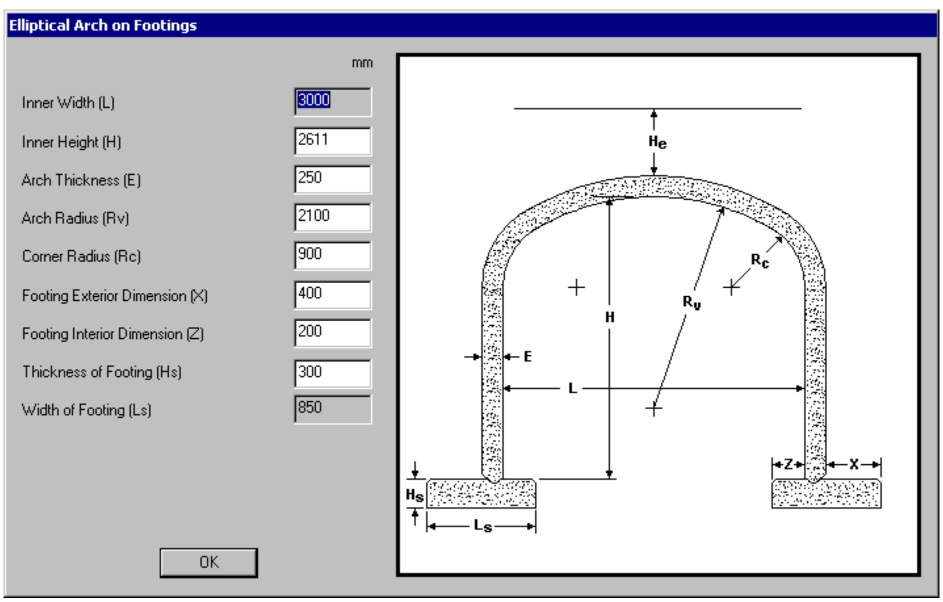
VisualDesign Culvert Models

Circular Arch



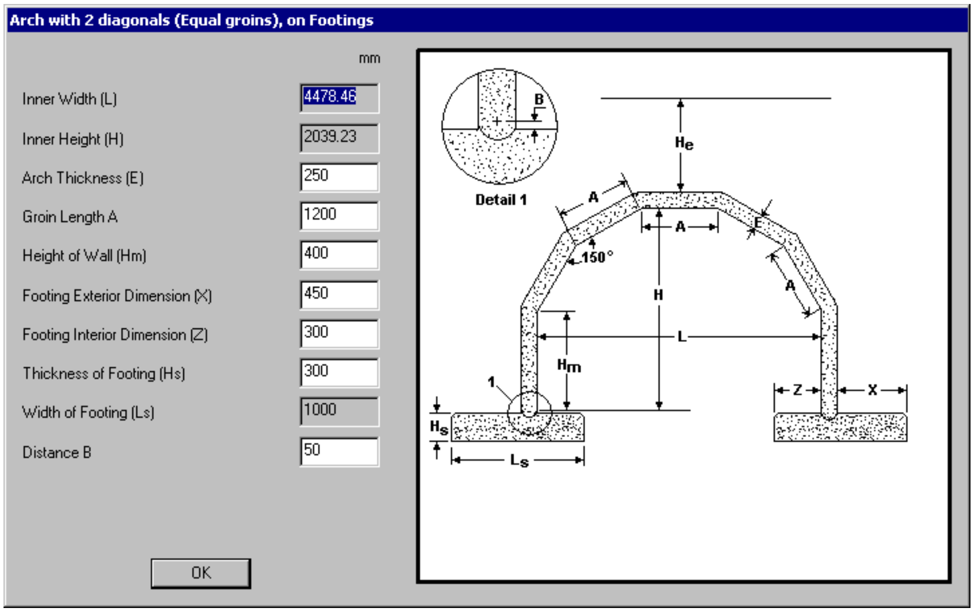
This model is also available with pedestals or a slab.

Elliptical Arch



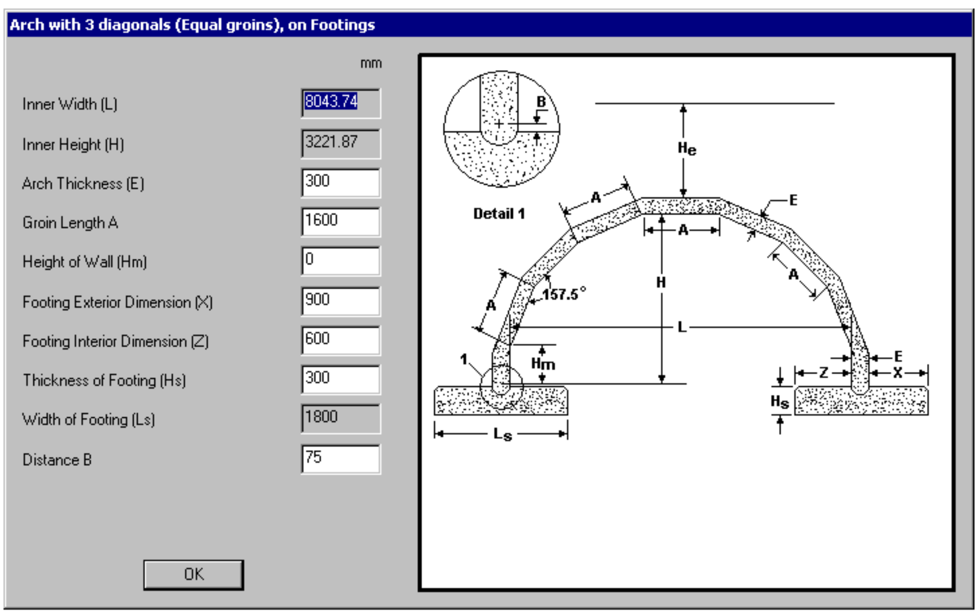
This model is also available with a slab.

Arch with 2 Diagonals (Equal groins)



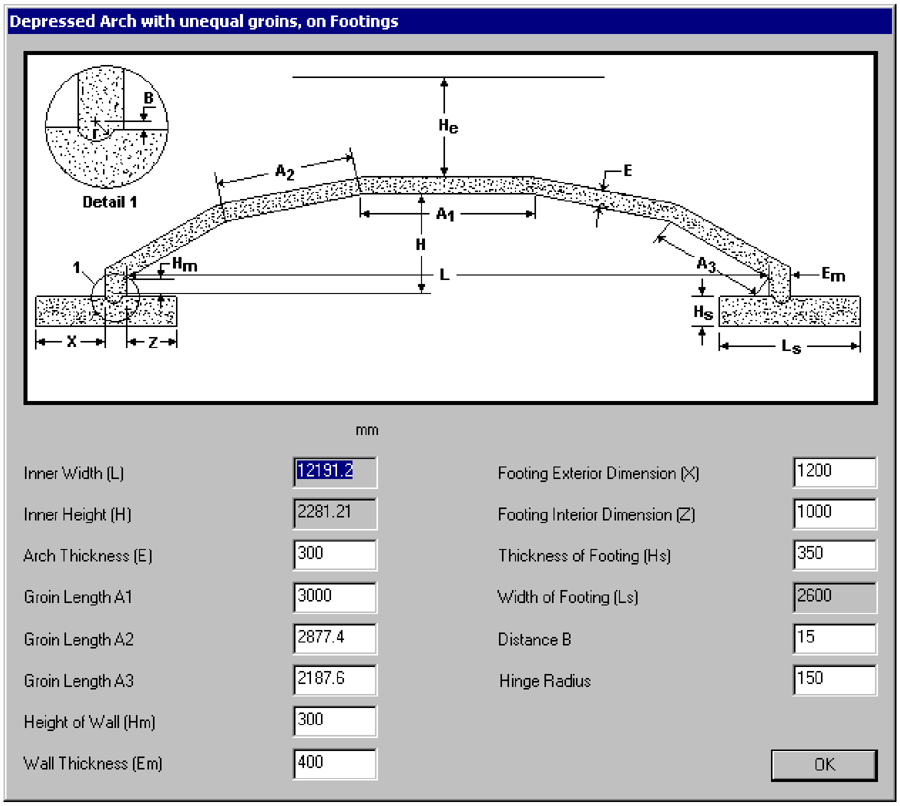
This model is also available with pedestals or a slab.

Arch with 3 Diagonals (Equal groins)



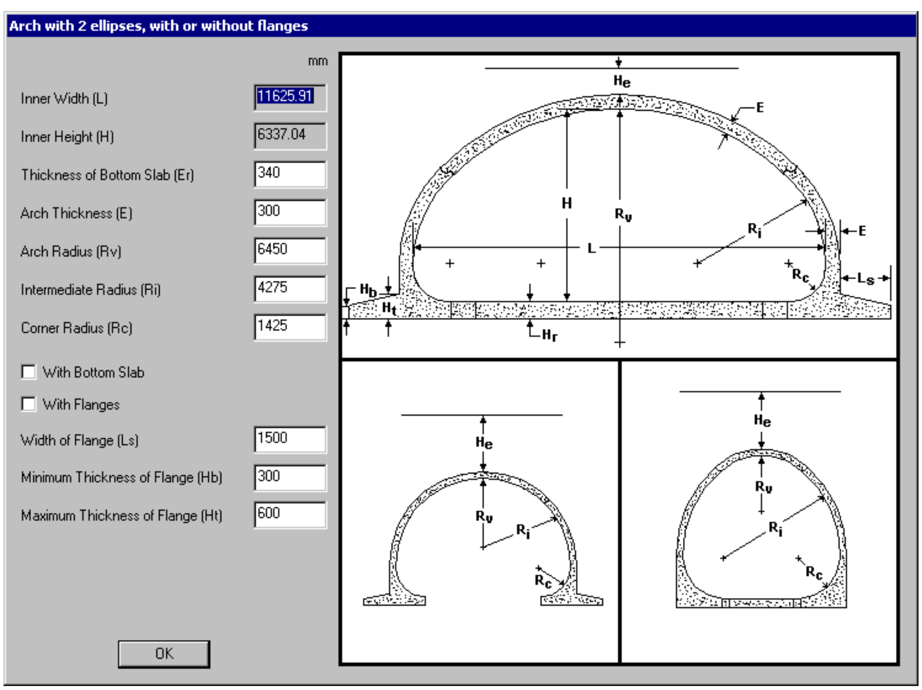
This model is also available with pedestals or a slab.

Depressed Arch with Unequal Groins

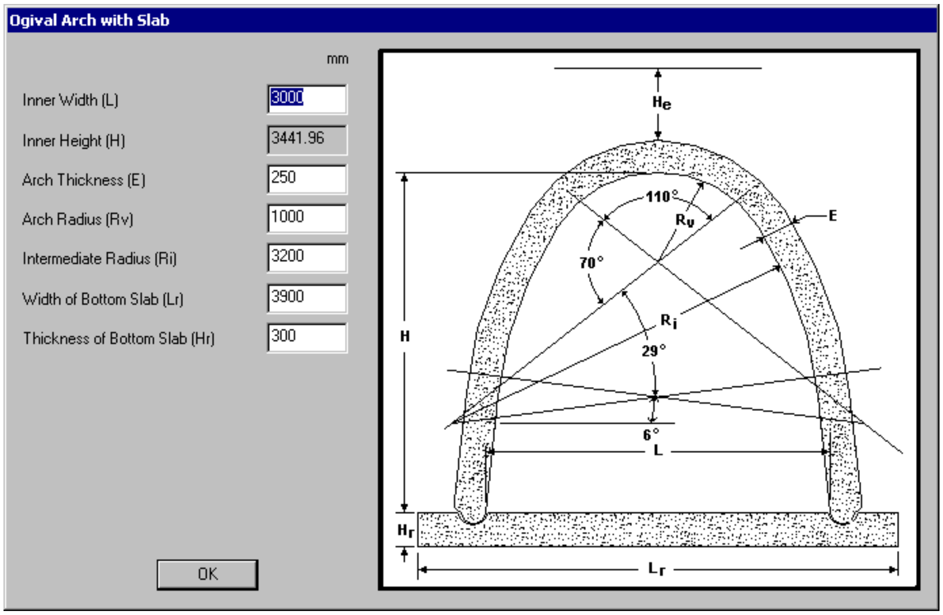


This model is also available with pedestals or a slab.

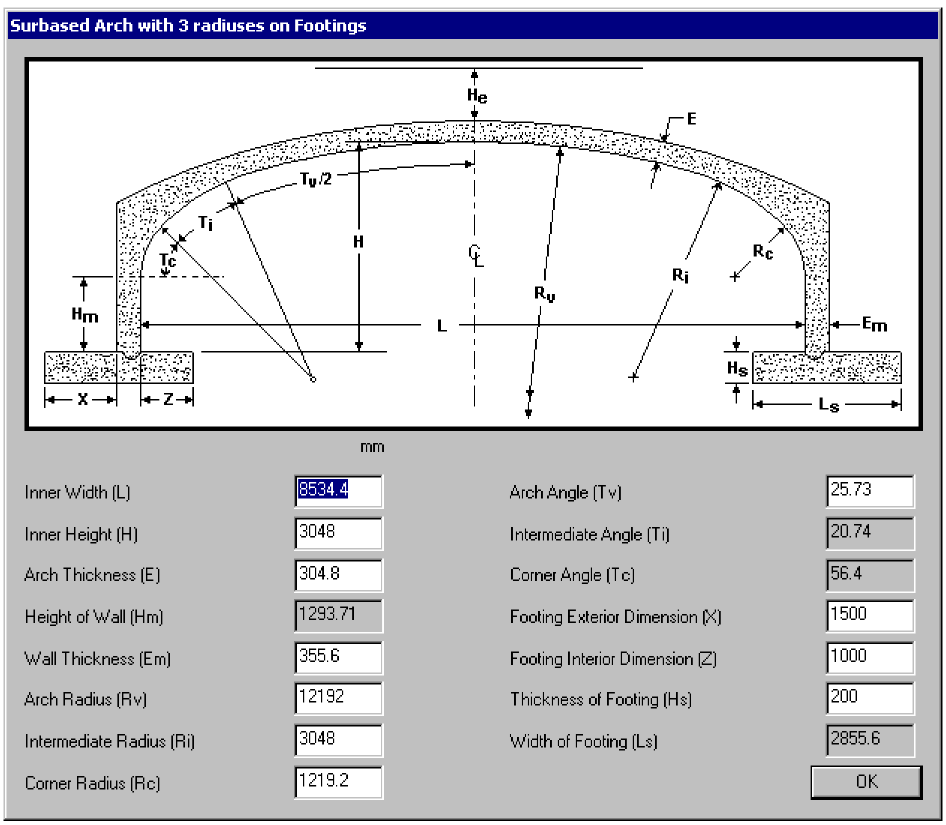
Arch with 2 Ellipses, with or without flanges



Ogival Arch with Slab

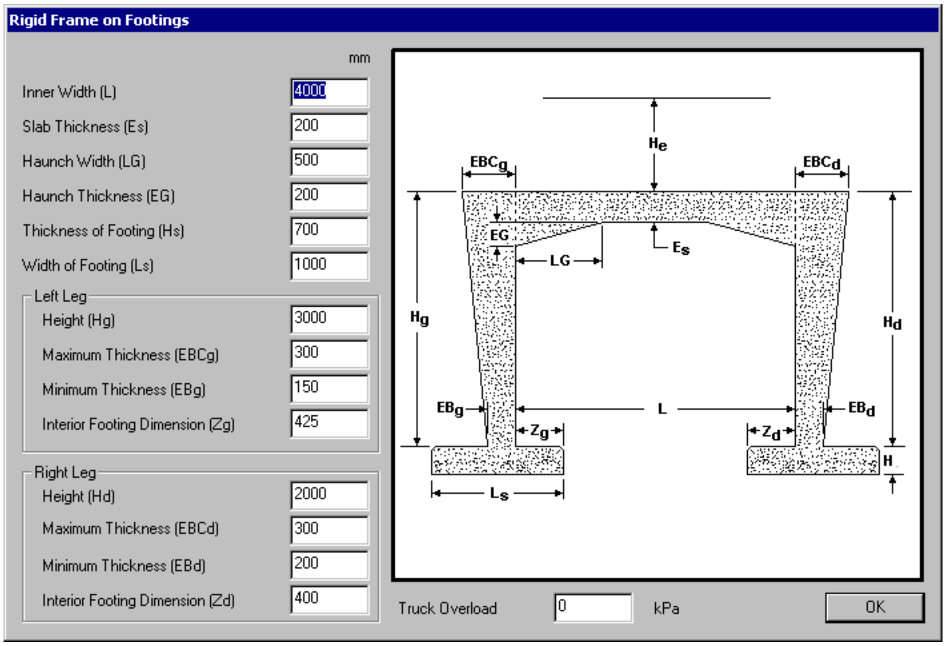


Surbased Arch with 3 radiuses



This model is also available with pedestals or a slab.

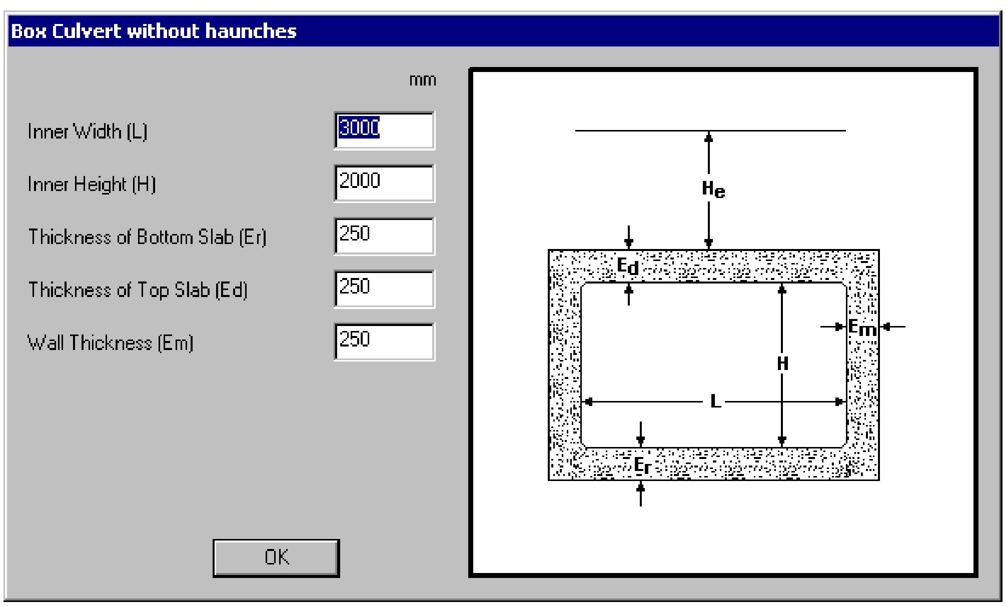
Rigid Frame on Footings



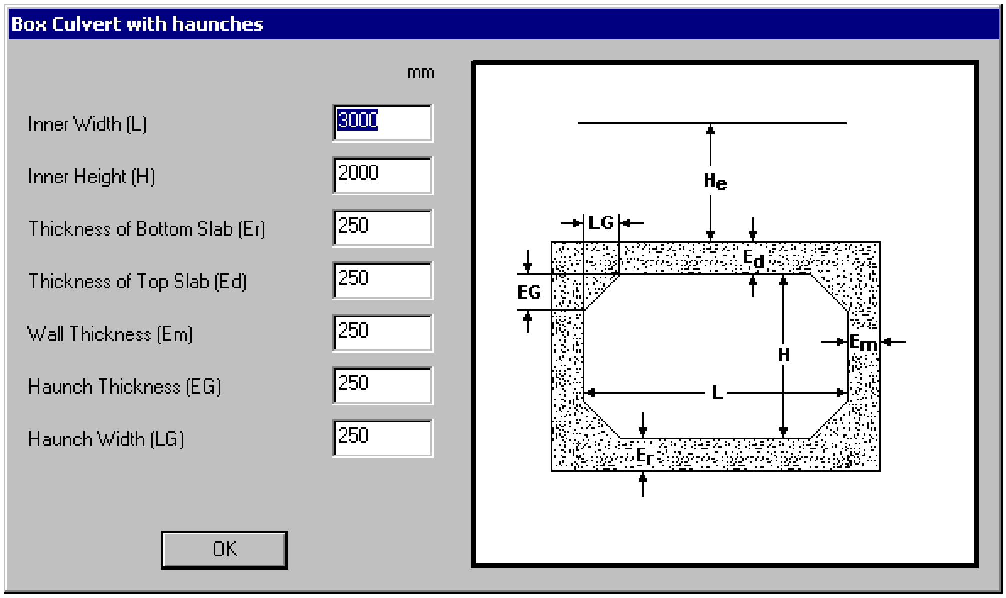
PARTICULARITIES

Backfill cannot be modeled over this culvert model. Therefore, the elevation of mobile will always be fixed to 0.0 in the Moving Load Cases spreadsheet, meaning that it is located right at the top of culvert. You must choose a moving load other than the initialized [2D]-CL-Unit and define 2D axle factors.

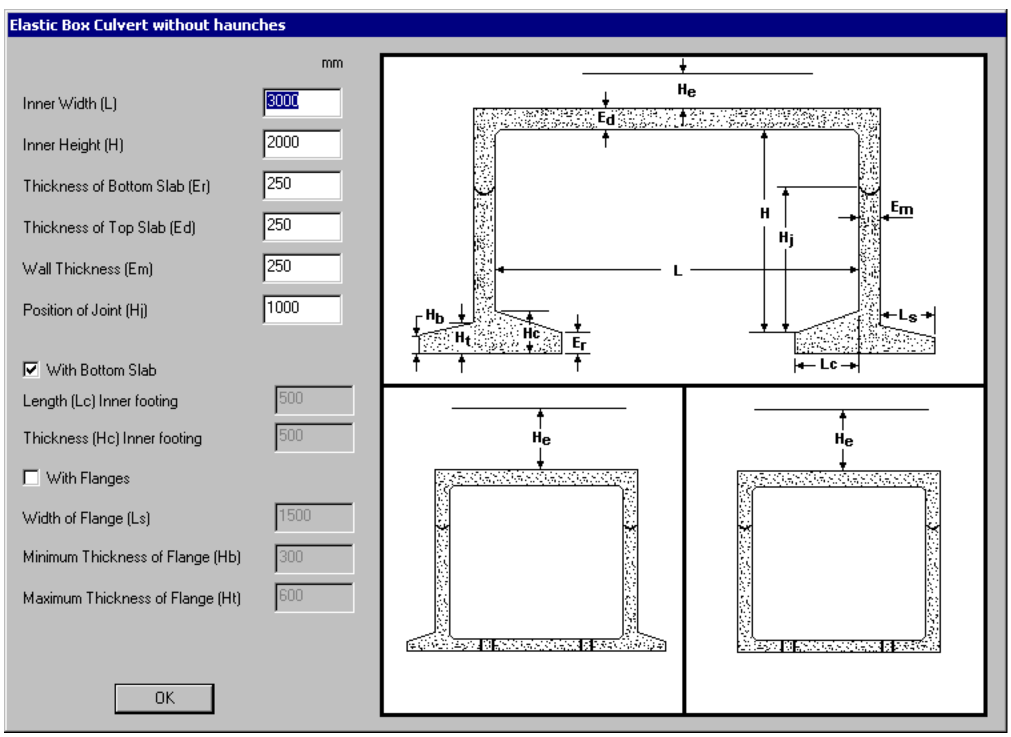
Rigid Box Culvert



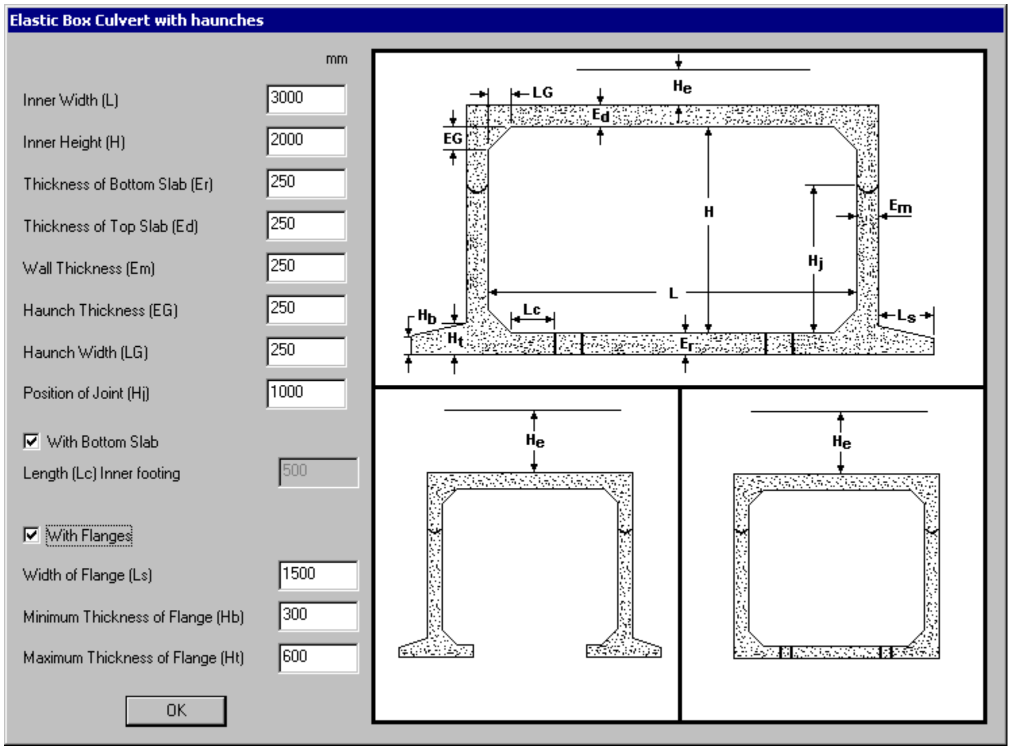
Rigid Box Culvert with Haunches



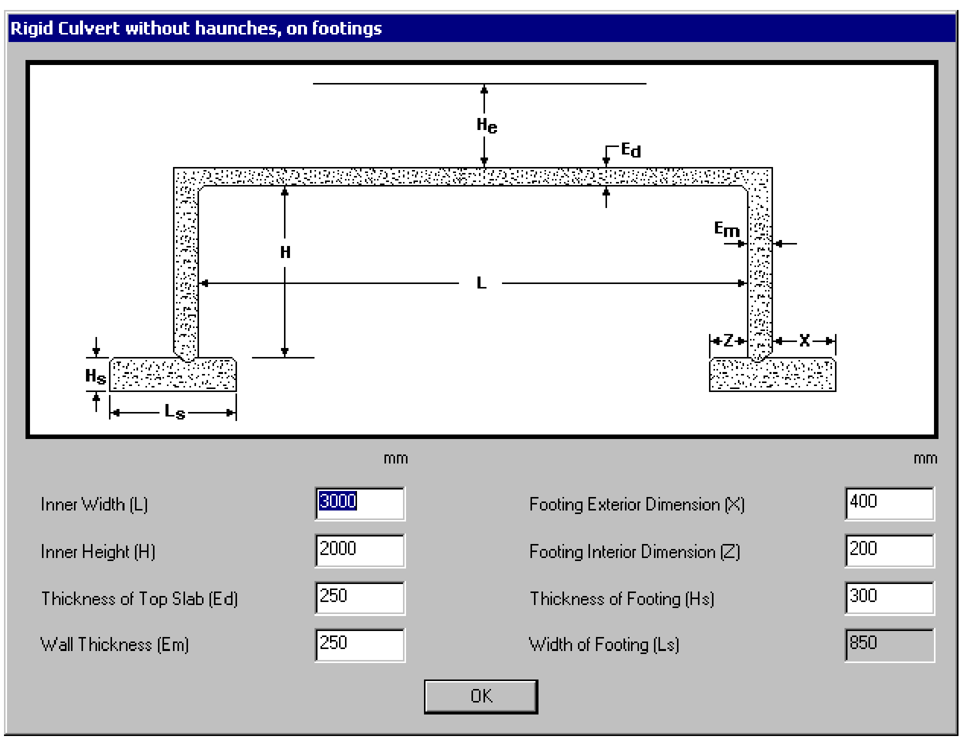
Elastic Box Culvert



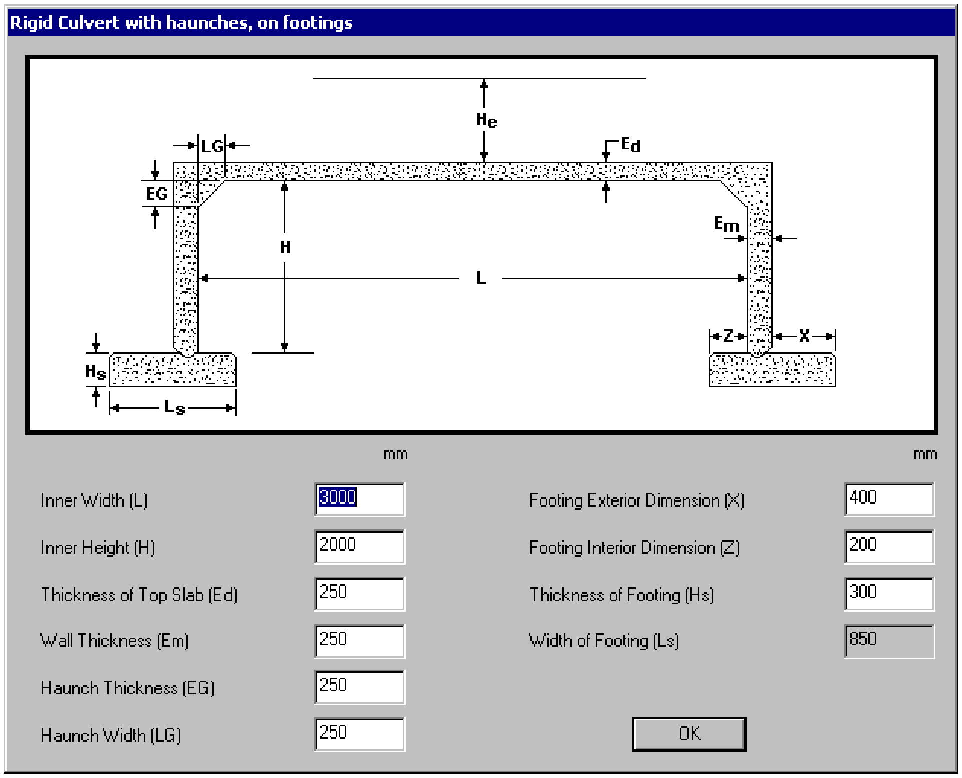
Elastic Box Culvert with Haunches



Open Frame on Footings



Open Frame on Footings – with Haunches



Culvert Generator

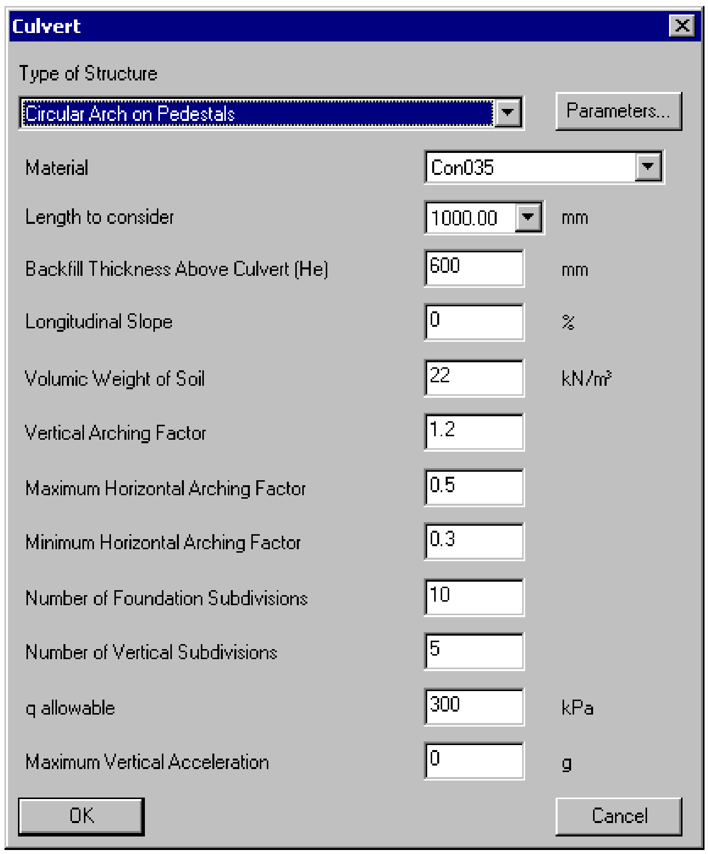
Generation of Culvert

Start a project

- Create a new file by selecting **New** in the **File** menu.

Generator

- Go to **Structure** / **Generator / Culvert** and select a culvert model. The **Culvert** dialog box will appear on your screen.



- Click on the arrow and select a culvert model among the drop-down list box. Select a concrete material and other parameters.

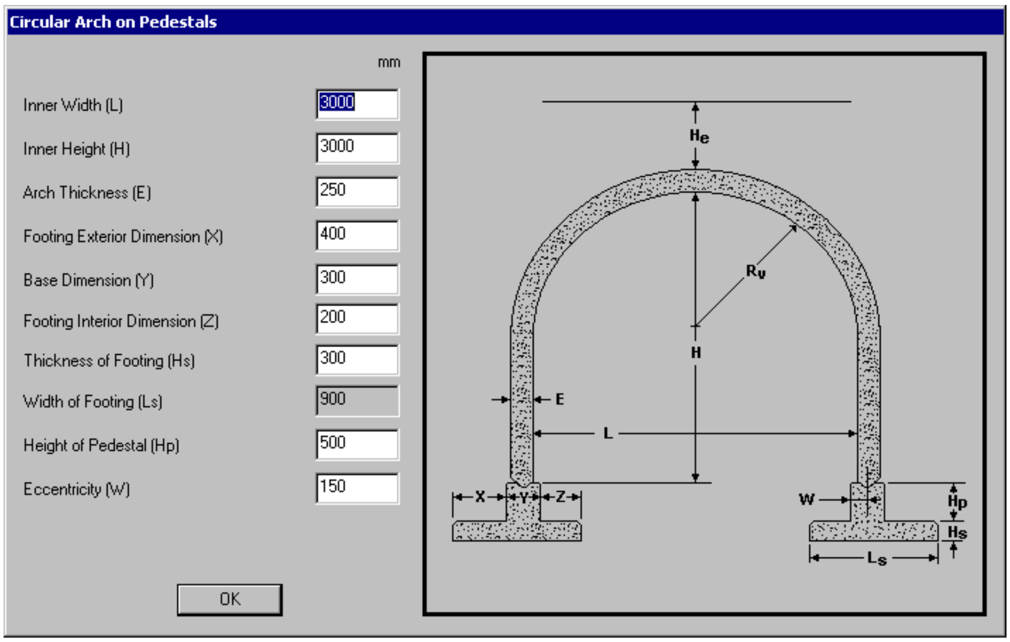
Description of the Culverts Dialog Box:

Field	Description
Type of Structure	List box that includes all culvert models
Parameters Button	This button opens a specific culvert dialog box where dimensions can be specified.
Material	List box that includes materials. Click the arrow to open the list box and select a material.
Length to consider	Select the length of the culvert The list includes lengths varying from 250 mm to 1500 mm.
Backfill thickness above culvert (He)	Specify the thickness of backfill above the culvert. The backfill must be at least 600mm thick.
Longitudinal Slope	Specify the backfill longitudinal slope
Unit weight of soil	Enter the backfill unit weight.
Vertical Arching Factor (λv)	Factor used to analyse soil/structure interaction effects over vertical earth pressure for buried structures.
Maximum horizontal arching factor (λh max)	Factor used to analyse maximum soil/structure interaction effects over horizontal earth pressure for buried structures.
Minimum horizontal arching factor (λh min)	Factor that is used to analyse minimum soil/structure interaction effects over horizontal earth pressure for buried structures.
Number of subdivisions for the foundation	This number of subdivisions will be applied to footings and on culvert invert. The software will calculate and display forces that are located at each subdivision.
Number of vertical subdivisions	Subdivisions will be applied to pedestals and elastic box culvert (vertical walls). The program will calculate and display forces that are located at each subdivision.
q allowable	Allowable bearing capacity for this soil. According to *Bowles* theory, the stiffness of a spring support modeling the soil will be equal to q *allowable* *40 * SF * tributary area of spring support, where SF = 3.
Maximum vertical acceleration	This value (g) will be used to calculate seismic loads to be applied to the culvert.

Parameters

- Press the *Parameters* button included in the **Culverts** dialog box. (You can modify default values before or after you pressed down the "Parameters" button.)

- Enter the culvert dimensions and press OK.



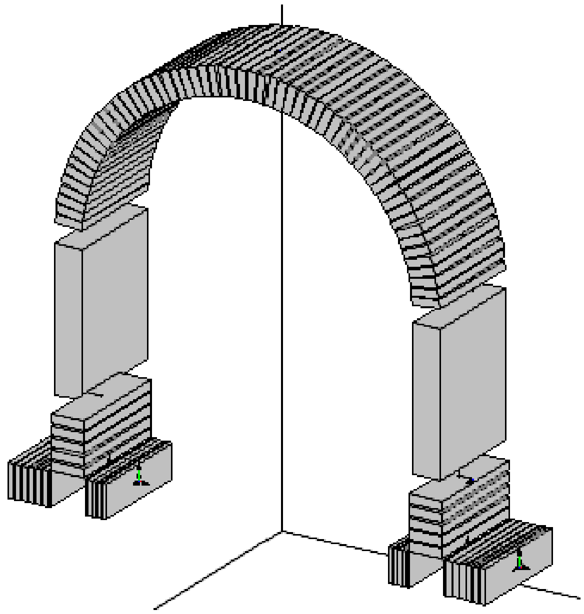
- You will be back in the **Culverts** dialog box. To exit and save data, press OK.

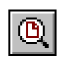
VisualDesign will automatically generate the culvert according to the dimensions that you specified.

- Save the file.

Generated model

Here is an example of a generated culvert model using the *3D Display* option for members (**View Options** dialog box):



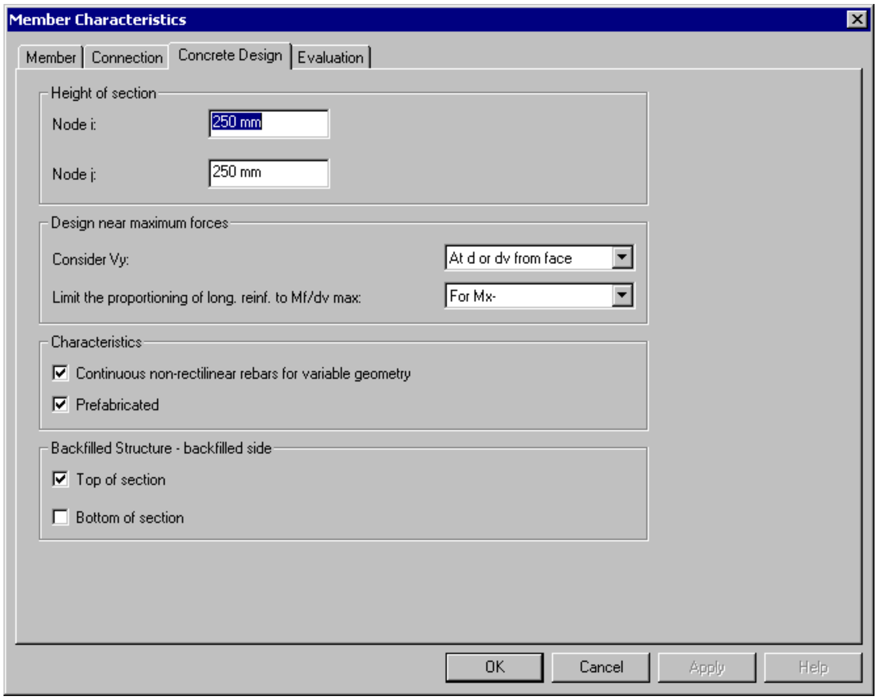
- Use the **Global Zoom** function  of **View** toolbar to centre the image. To zoom in or out, use **Zoom +**  or **Zoom -**  icons (or use your keyboard key + or-)

Culvert Members

To have a look at culvert member characteristics, double-click on any member or click once and press the **Properties** icon to open the **Member Characteristics** dialog box. In the **Member** tab, you will notice that all arched elements are part of the moving load axis. The chosen member section is (1000x1000). You do not have to change this section because it is a fictitious one.

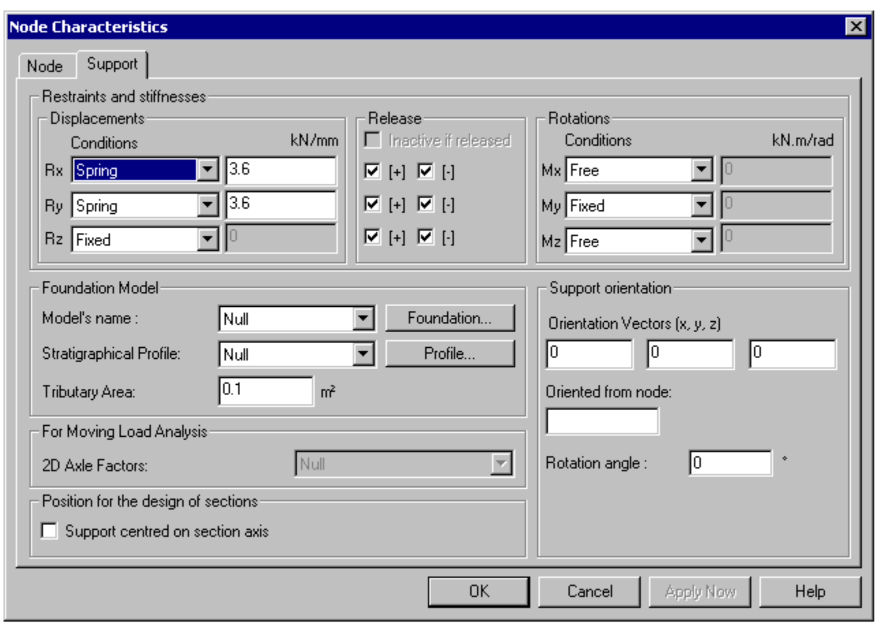
In the **Connection** tab, VisualDesign automatically aligned the member axis at the top of the sections. Therefore, all nodes are located at the top of sections. Hide member offsets through the **Attributes** tab of **View Options** dialog box.

In the **Concrete Design** tab, section heights at node i and j are equal to the culvert wall thickness, which was specified by the user. This thickness is fictitious one. Option "Non rectilinear continuous reinforcement" is automatically activated so VisualDesign will place main reinforcement along the walls of variable geometry. The *Prefabricated* option is activated to consider the appropriate resistance factor.



Spring Supports

Spring supports have been generated under the footing and slab. The stiffness of spring supports is depending on the bearing capacity of soil specified in the field *q allowable* in the **Culvert** dialog box. The tributary area of each support is also calculated.



Spring supports' stiffness for lateral backfill on culverts

In VisualDesign, we use the following equation:

$$K_{sh} = 192000(N/m^3) \cdot \frac{\alpha}{H_T} \cdot \left[\frac{(H_T + 0.3)}{H_T} \right]^2$$

Here is an extract of the document from Béton Provincial Ltée – *Homologation des conduits Matière- Rev. 0 –Novembre 2001, p26-27* on which we based our calculations of spring supports' stiffness located laterally along the walls of a buried culvert.

[…]

2.1.4 Lateral Backfill

The nature of backfill materials used to cover a structure is defined by Standards from the Ministry of Transportation of Quebec (MTQ). In the present project, the granular borrow material is of gage CG20 for the pad and coating. The latter is installed using layers of a maximum of 150mm thick and compacted to a minimum density of 95% of the maximum dry density as defined with the modified Proctor test.

It is important to note that the studied concrete structure is deformable. The distribution of lateral strains is variable with depth and the biggest value is located at mid-height of the structure. These conditions are similar to the distribution of stresses observed at the base of a footing that is lying over a granular material. Then, the approach would be to consider the foundation width "H", as the height of the structure. J.E. Bowles (1996) formulation used previously could be applied as follows:

$$K_{sh} = FS \cdot q_a \cdot \left(\frac{1000}{\Delta h} \right)$$

where K_{sh} is the coefficient of horizontal elastic reaction (kN/m³).

The method that is mostly used in Quebec for the calculation of net allowable bearing capacity "q_a" is the one proposed by G.G. Meyerhof (1956) and explained in the Canadian Manual of Foundation Engineering (1994).

$$q_a = 8 \cdot N \cdot K_d \cdot \left[\frac{(H_{Total} + 0.3)}{H_{Total}} \right]^2$$

The granular material CG20 is compacted at 95% of PM and thus, considered in a compact state vertically. Laterally, we are considering a value of 8 for ratio "N", which is smaller that the one that is usually held in the vertical direction. It represents secure conditions.

Lateral settlement "Δh" is considered in the form of a fraction of "H", as recommended in literature, namely $\Delta h = H/\alpha$ (m). "α" value increases with the height increase, limiting the settlement and allowing a good structural behaviour. The variation of "α" goes as follows:

H_{Total} (m)	α
1,0 to ≤ 3,0	180
3,1 to ≤ 5,0	220
5,1 to ≤ 7,0	245
7,1 to ≤ 9,0	280
≥ 9,1	310

The "Ksh" formulation becomes:

$$K_{sh} = 192 \cdot \frac{\alpha}{H_{Total}} \cdot \left[\frac{(H_{Total} + 0.3)}{H_{Total}} \right]^2 \quad [\ldots]$$

REFERENCES

Terzaghi, K. 1995. Evaluation of coefficients of subgrade reaction. Géotechnique, vol. 5, no.4, pp. 297-326. Discussion in vol.6, no.2, pp.94-98.

Terzaghi, K. et Peck, R.B. 1948. Soil Mechanics in engineering practice. John Wiley and Sons, inc. New York.

Manuel Canadien d'Ingén1iereie des Fondations, seconde edition 1994. Société Canadienne de Géotechnique (page 183). C/o Bitech publishers Ltd. Canada.

Bowles, J.E. 1996, Foundation Analysis and Design, fifth edition (PP. 501-506). The McGraw-Hill companies inc., new York.

Meyerhof, G.G. 1956. Penetration tests and bearing capacity of cohesionless soils. ASCE Journal of the soil Mechanics and Foundations Division, vol. 82, no. SM1, (pp. 417-430)

Culvert Specification

You can modify the culvert specification that was generated for the design of required reinforcement. Select **Specifications / Concrete** in **Structure** menu. The generated concrete specifications are called *Culvert,* one for design and the second for verification.

The General tab

The selected code is CAN/CSA-S6-00. By default, VisualDesign uses 10M, 15M, 20M, 25M, 30M, or 35M for the design of culvert members and slab. To add or withdraw type of bars from this list, double-click in the cell and choose rebars by checking boxes in the *Rebars* selection tree.

The chosen optimization is *Constant "s" (M+ M-)*, to get a constant spacing for positive and negative bending moment, or a multiple of this spacing, such as 50mm, 100mm, 150mm.

The option *Maximize Vr* is automatically selected. VisualDesign will not consider tensioned reinforcing bars at the top. Therefore, the distance "d" or "dv" will be greater and, Mr, will be smaller. This design will maximize the shear resistance of the member.

The Beam-slab tab

The minimum spacing for main reinforcement is fixed to 150mm. If the transverse reinforcement is composed of a welded mesh, the default spacing is set to 200mm.

The option *Buried-Precast* is selected as type of structure. Therefore, the design of culvert will be done according to code S6-00 chapter 7 *Buried Structures*, clauses 7.8.8 to 7.8.12.

Culvert Continuous Systems

While you are working in "Structure" mode, select the **Continuous Systems** spreadsheet in **Structure** menu and look at criteria that have been specified in the *Culvert* specification.

VisualDesign automatically defines continuous systems according to culvert model. Usually, one will be created for the arched structure and a second one or two for foundations. Culvert walls and footings are continuous systems of the *Beam-slab type*, with a variable geometry. No stirrup design is done. Concrete covers are fixed to 50mm, as default. To modify concrete covers, click in cells and enter a new value. Crack control parameters are fixed to a value of 70000 N/mm.

See also Reinforced Concrete Design Chapter

Continuous Systems spreadsheet (Chapter 10)
The Beam-slab Type of Continuous System with Variable Height

Generated Loads

Generated Loads

VisualDesign generates the required loads for each culvert. To have a look or add new loads, go to **Loads / Load Cases / Definition**.

Dead Loads

Standard, *Backfill min.* and *Backfill max* dead loads are included in the spreadsheet. Backfill loads are created according to the maximum and minimum horizontal arching factors specified in the **Culverts** dialog box. They are applied at the right and left side of the culvert. If these factors are equal, only one backfill load will be created.

Live Loads

Live loads *AW right* and *AW left* are corresponding to the overload due to the approaching wheel located over the backfill and close to the buried structure. They are generated according to Code S6-00, except for the rigid frame model. These overloads are defined in Code S6-00, chapter 7, code provision 7.8.5.3.3 for the design of buried structures.

Seismic Loads

Finally, seismic loads *EQ min* and *EQ max* are calculated from the value that is entered in the field *Max Vertical Acceleration* in the **Culvert** dialog box. They are applied on the right and left side of the culvert.

Moving Loads

Generated Moving Loads

VisualDesign created the mobile *[2D]-CL-Unit* as the moving load case, which is specific for buried culverts. If you selected the rigid frame on footings as the culvert model, select another 2D moving load case and define 2D Axle factors because this structure is not buried.

Go to **Loads** / **Moving Load Cases / Definition** to consult the generated moving load case: The mobile *[2D]-CL-Unit* is selected and the moving load envelope Lm01, the type of culvert, and elevation of backfill above the culvert are also specified.

The moving load envelope will be available when the design will be completed.

Truck Elevation over the Backfill

VisualDesign will convert punctual moving loads into surface loads from the specified backfill elevation ("Elevation" column of **Moving Load** tab). This elevation, which is automatically calculated, corresponds to the top of culvert plus the thickness of backfill above it. Loads will be distributed through the backfill, down to the culvert foundation. Refer to topic *Distribution of moving loads through backfill*

Description of specific fields used for this calculation:

Field	Description
Use Elevation	Indicate if the structure is an arch or a box culvert (rectangular) for the calculation of moving loads distribution through backfill.
Elevation	Y-coordinate above backfill, where moving loads will be applied.
Longitudinal Slope (%)	Longitudinal slope of moving load surface, as specified in the **Culvert** dialog box. This option is not yet activated.

Distribution of Moving Loads through Backfill

This topic applies to the design of buried arched and box culverts, according to Code S6-00 and for a span that does not exceed 15 meters.

Culvert type and backfill elevation are specified in the **Moving Load Cases** spreadsheet.

Mobile [2D]-CL-Unit:

The [2D]-CL-Unit truck is composed of two unitary loads that are 7.2 m apart. This configuration is used for the calculation of influence lines only. 7.2 meters is the distance between axle no. 4 (175 kN) and the centreline of axles 2 and 3 (125 kN each).

VisualDesign will compute the distribution of moving loads through the backfill according to Code S6-00. The Dynamic Load Allowance (DLA) depends on the type of structure and position of axles, as explained below:

Dynamic Load Allowance (DLA) - Code provision 3.8.4.5.2

Arched Culverts

DLA = 0.4*(1.0 – 0.5 De), but not inferior to 0.1.

"De" is the thickness of backfill above the culvert.

Box Culverts (Code provision 3.8.4.5.3)

b) DLA = 0.4 * (1.0 – 0.5De) but > 0.1, where only one axle of the CL-W truck is used.

c) DLA = 0.3 * (1.0 – 0.5De) but > 0.1, where any two axles of the CL-W truck, or axles 1, 2 and 3, are used.

d) DLA = 0.25 * (1.0 – 0.5De) but > 0.1, where three axles of the CL-W truck, except for axles

First Set of Loadings Relatively to Axle #4 (175 kN)

Case A) One mobile – One wheel without interference with other wheels – thickness of backfill above the structure varies from 0.6m to 0.686m

A wheel supporting a load of 175 kN/2 creates a pressure equal to:

P = DLA * 175 kN/2 / (Lp*Ll), where:

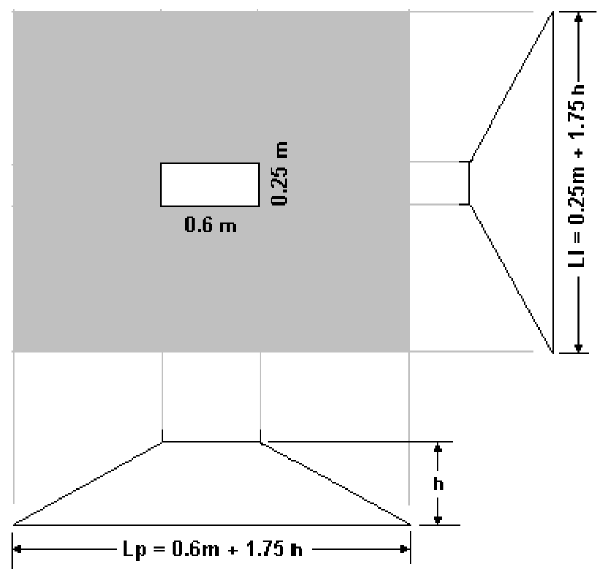
DLA = Dynamic Load Allowance;

Lp = 0.6m + 1.75h;

Ll = 0.25m + 1.75h = longitudinal width studied;

h = Depth of load distribution.

$$P = DLA * 87.5\ kN / (3.0625h^2 + 1.4875h + 0.15)$$



Case B) One mobile: Two Transverse wheels with interference between the two – Thickness of backfill above the structure varying from 0.686m to 0.772m

Two wheels are supporting a load of 175 kN/2 each and are creating a pressure that is equal to:

P = DLA * 175 kN/(Lp*Ll)

Where:

DLA = Dynamic Load Allowance;

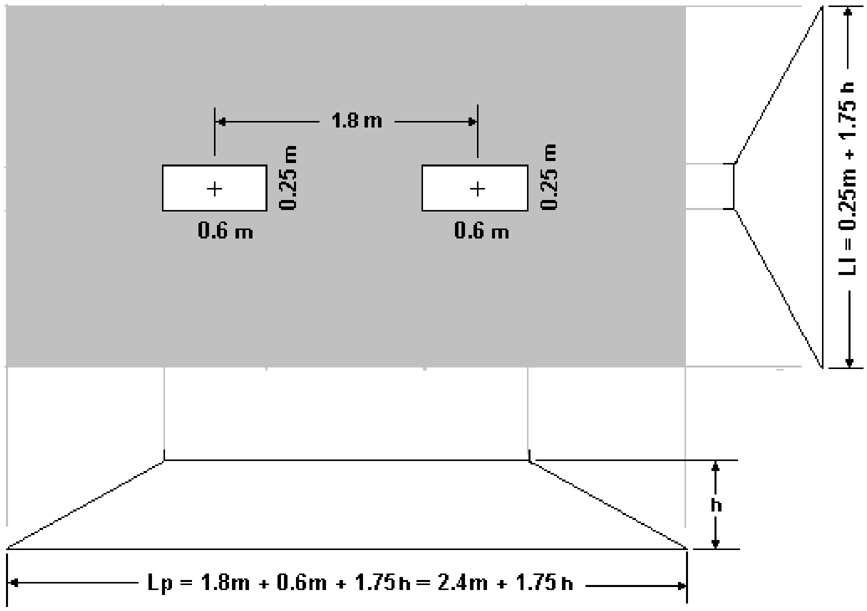
Lp = 2.4m + 1.75h;

Ll = 0.25m + 1.75h = studied longitudinal width;

h = depth of load distribution.

Then,

$$P = DLA * 175\ kN/(0.6 + 4.6375h + 3.0625h^2)$$



Case C) Two mobiles: Four transverse wheels interfering with each other – Thickness of backfill above the structure is of 0.772m or more

Four wheels support a load of 175 kN/2 each and create a pressure equal to:

P = DLA * 0.9 * 2 * 175 kN / (Lp * Ll)

Where:

DLA = Dynamic Load Allowance;

Lp = 5.4m + 1.75h

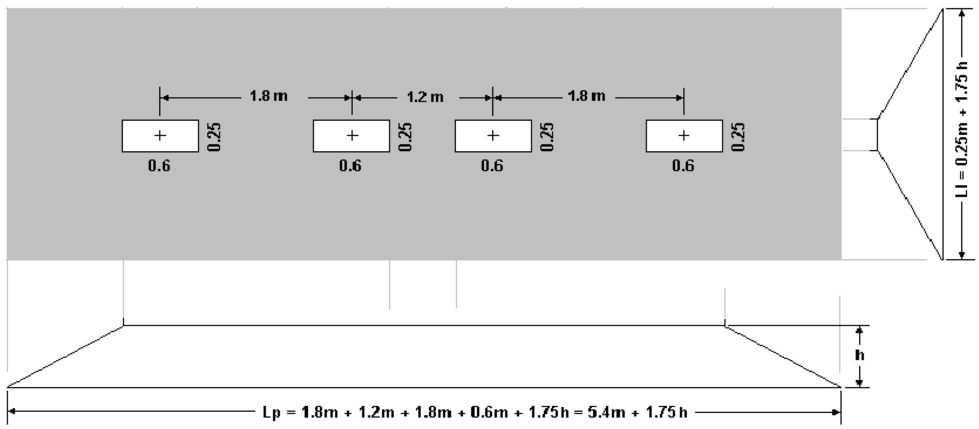
Ll = 0.25m + 1.75h = studied longitudinal width;

h = depth of load distribution.

Then,

P = DLA * 0.9 * 2 * 175 kN / (1.35 + 9.8875h +3.0625h^2)

P = DLA * 315 kN/(1.35 + 9.8875h +3.0625h^2)



Case D) Three mobiles: six transverse wheels interfering with each other (N.B. This case is never critical)

Four wheels supporting a load of 175 kN/2 each create a pressure equal to:

P = DLA * 0.8 * 3 * 175 kN / (Lp * Ll)

Where:

DLA = Dynamic Load Allowance;

Lp = 8.4m + 1.75h

Ll = 0.25m + 1.75h = studied longitudinal width;

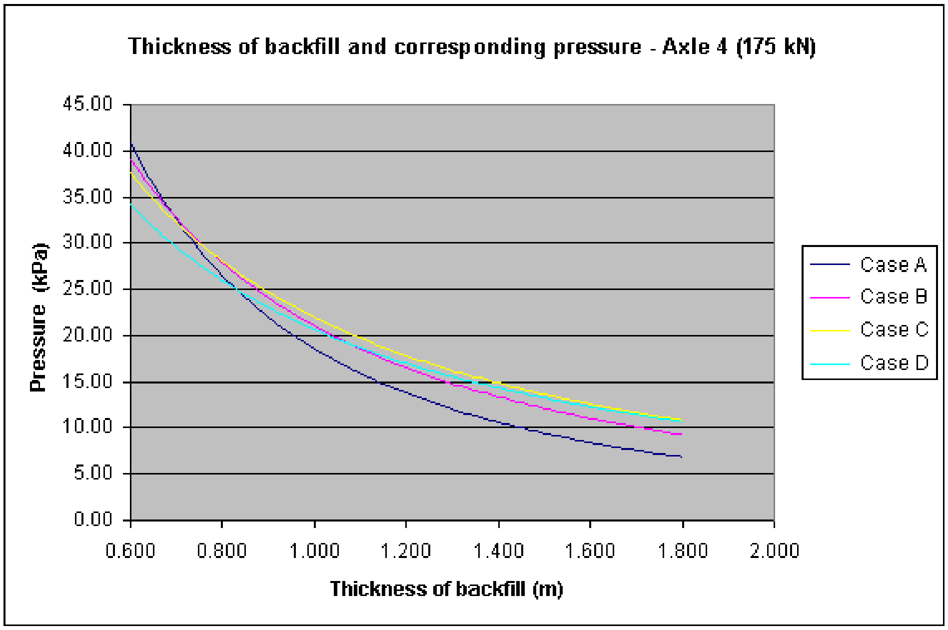
h = depth of load distribution.

Then,

P = DLA * 0.8 * 3 * 175 kN / (2.1 + 15.1375h +3.0625h²)

P = DLA * 420 kN/(2.1 + 15.1375h +3.0625h²)

The summary for this set of loadings is as follows:



Second Set of Loadings Relatively to axles #2 & #3 (125 kN each)

Case A) One mobile: Two longitudinal wheels with no interference with other wheels– Thickness of backfill above the structure varies from 0.6m to 0.686m

Two longitudinal wheels supporting a load of 125 kN/2 each create a pressure equal to:

P = DLA * 125 kN/2 / (Lp * Ll) where

DLA = Dynamic Load Allowance;

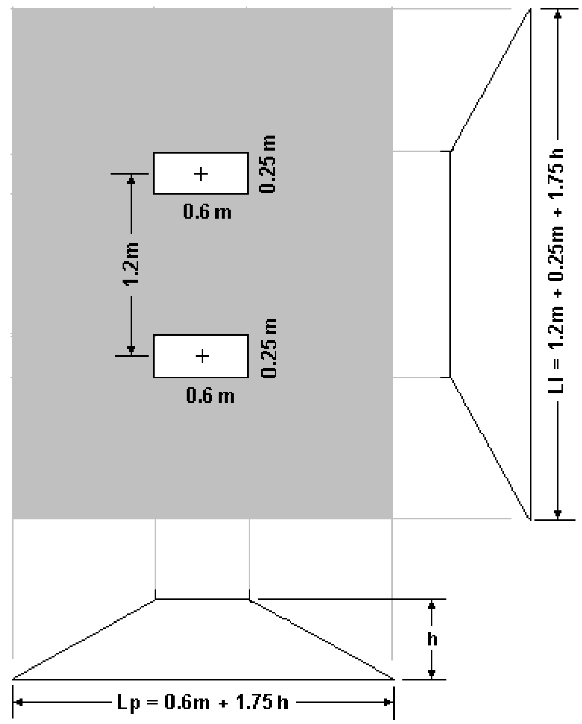
Lp = 0.6m + 1.75h

Ll = 1.2m + 0.25m + 1.75h = 1.45m + 1.75h = studied longitudinal width;

h = depth of load distribution.

P = DLA * 62.5 kN / ($3.0625h^2$ + 3.5875h + 0.87)

P = DLA * 62.5 kN / ($3.0625h^2$ + 3.5875h + 0.87)



Case B) One mobile: Two transverse and two longitudinal wheels with interference with each other – Thickness of backfill above the structure varies from 0.686m to 0.772m

Four wheels supporting a load of 125 kN/2 each, create a pressure equal to:

P = DLA * 125 kN * 2 * 1.3 / (Lp * Ll), where:

DLA = Dynamic Load Allowance;

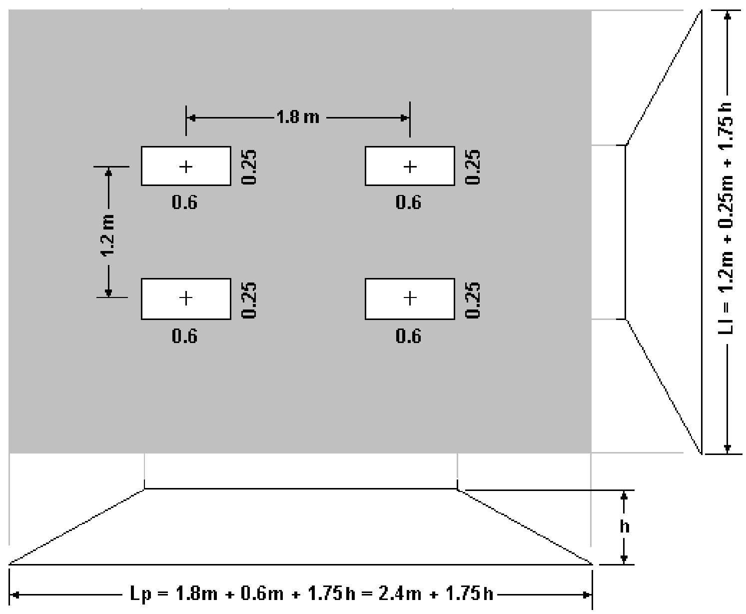
Lp = 2.4m + 1.75h

Ll = 1.45m + 1.75h = studied longitudinal width;

h = depth of load distribution.

P = DLA * 125 kN * 2 * 1.3 / (3.48 + 6.7375h +3.0625h^2)

P = DLA * 325 kN / (3.48 + 6.7375h +3.0625h^2)



Case C) Two mobiles: Eight transverse wheels with interference with each other – Thickness of backfill above the structure is equal or greater to 0.772m

Eight wheels supporting a load of 125 kN/2 each create a pressure equal to:

P = DLA * 0.9 * 125 kN * 4 / (Lp * Ll)

Where

DLA = Dynamic Load Allowance;

Lp = 5.4m + 1.75h;

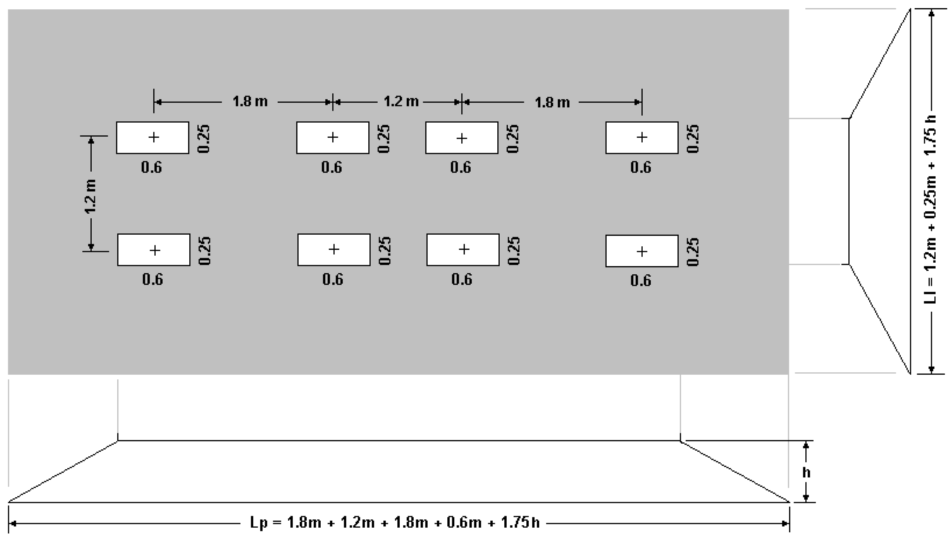
Ll = 1.45m + 1.75h = studied longitudinal width;

h = depth of load distribution.

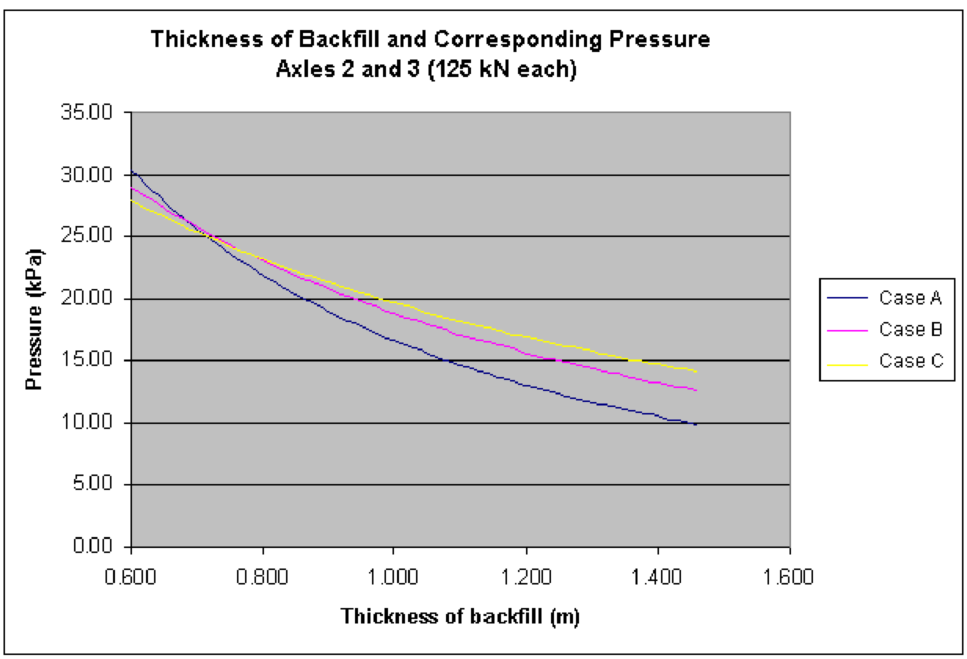
Then,

P = DLA * 0.9*4*125 kN / (1.35 + 9.8875h +3.0625h^2)

P = DLA * 450 kN / (7.83 + 11.9875h +3.0625h^2)



The summary for this set of loadings is as follows:



Load Combinations

Automatic Generation of Load Combinations

Call up the **Load Combinations Generation Wizard** in **Loads / Load Combinations** menu.

- In the **General Options** page, select Code S6-00 in the drop-down list box. Press the "Next" button.
- In the **Specific Options** page, select load combinations to be generated. In section *Special loads to be included*, tick off the "Moving load envelope Lm1" to include this envelope into the generation of load combinations. Press the "Next" button.
- In the **Selections** page, you will see load combinations that VisualDesign will be generating. If it is OK for you, click OK. Otherwise, uncheck load combinations that you do not want to include in the generation. Press the "Finish" button.
- The **Load Combinations** spreadsheet will appear on the screen. The generated load combinations are listed in the spreadsheet. You are allowed to modify load combination statuses before launching a design. If you do not want to analyse a load combination, disable the option "Required" by double clicking in the cell.
- VisualDesign has included load factor for each load included in load combinations that it generated. Have a look at **Load Factors** tab.

Analysis and Procedure

Analysis and Design

- Click the **Analysis and Design** icon  of Tools toolbar. In the dialog box, press the “Analyse” button to launch the design of culvert.

  The moving load analysis will automatically be launch first. Then, VisualDesign will begin its cyclic design according to your specifications.

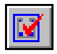
- Close the dialog box when the design is completed.

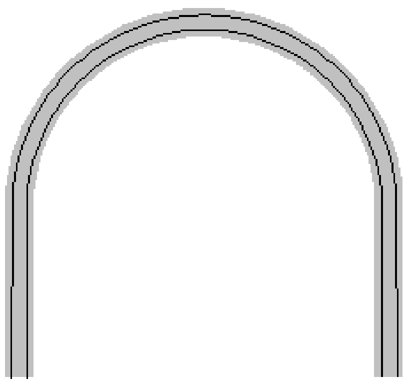
Procedure

- Create a new file by selecting **New** in **File** menu.
- Go to **Structure / Generator / Culverts**.
- Select a culvert model and specify dimensions.
- Add other load titles in the **Load Definition** spreadsheet, if needed.
- Generate load combinations (**Loads / Load Combinations / Automatic Generation**).
- Launch a *Reinforced Concrete Design* with function **Analysis and Design**.
- Double-click on any continuous system to open the Rebar Placement window and use the **View Options** to display force and resistance diagrams, dimensions, etc. Look at design results.

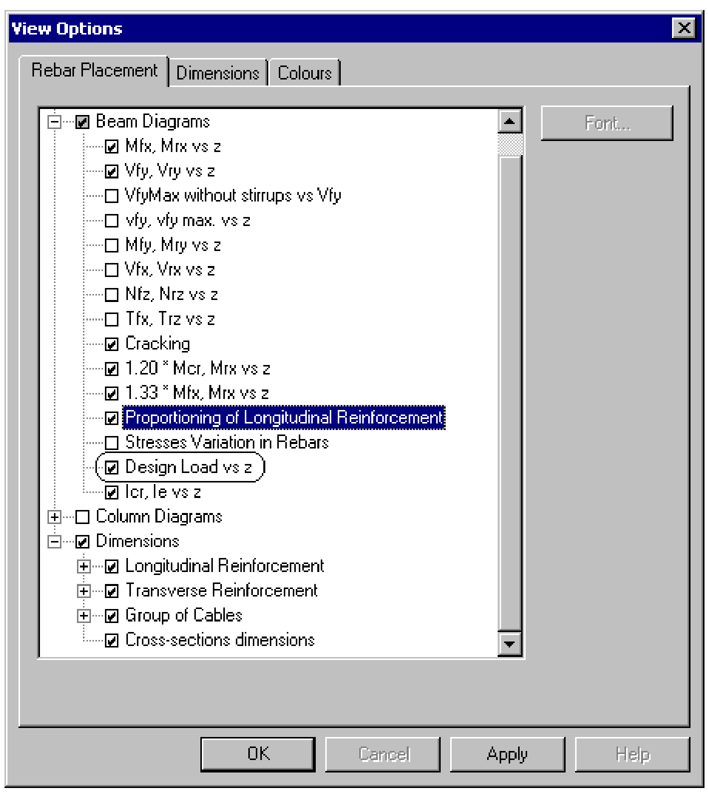
Culvert Design Results

Rebar Placement window

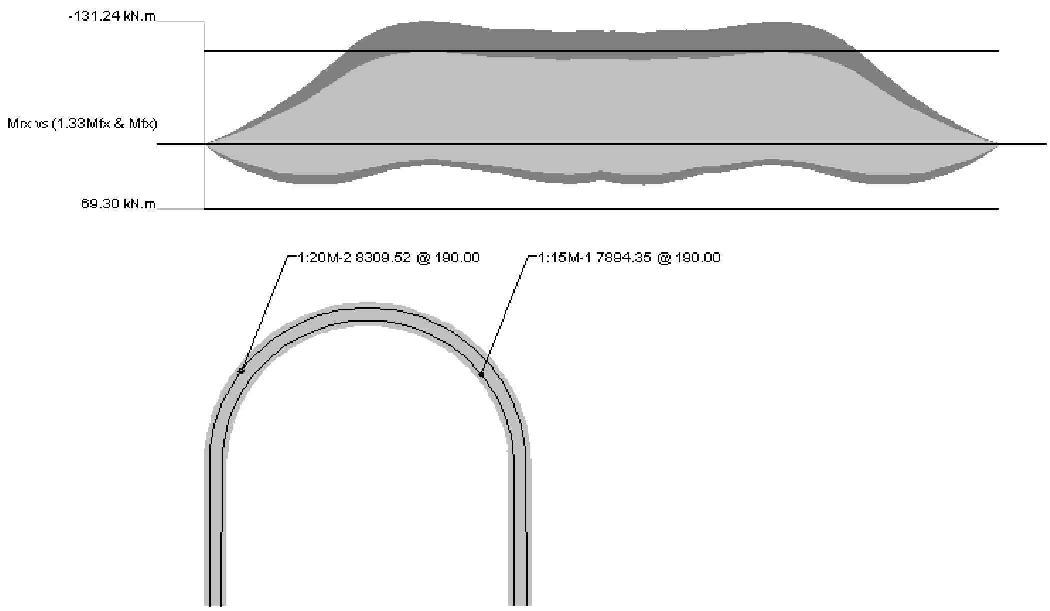
- To access the *Rebar Placement* window, activate the *Rebar Placement* mode  on Activation toolbar. Then, double click on any continuous system. The elevation view of the selected continuous system will be displayed on screen.



- Open the **View Options** dialog box. Check the *Dimensions* root and expand the *Beam Diagrams* roots. Select the diagrams that you want to look at.

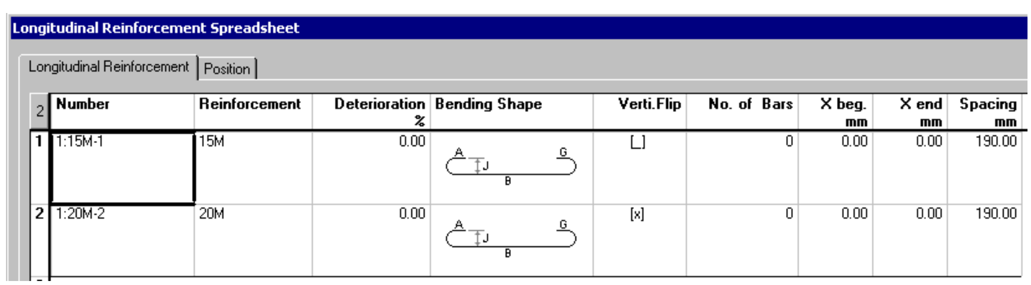


Note: The diagram "Design load vs z" is for culvert design. It shows the percentage of used capacity of the culvert for analysed load combinations. The diagram must be within limits of +/- 100%.



Design of Main Reinforcing Bars

- Main rebars are displayed on screen. The details are available in the **Main Reinforcement** spreadsheet (**Rebar Placement** menu).
- To open this spreadsheet and edit rebars, double click on a main rebar on the elevation view. The rebar dimensions can be changed and resistances will be automatically recalculated and displayed on diagrams.



Longitudinal Reinforcement Spreadsheet

Longitudinal Reinforcement | Position

2	Number	Reinforcement	Deterioration %	Bending Shape	Verti.Flip	No. of Bars	X beg. mm	X end mm	Spacing mm
1	1:15M-1	15M	0.00	A J B G	[ ]	0	0.00	0.00	190.00
2	1:20M-2	20M	0.00	A J B G	[x]	0	0.00	0.00	190.00

The Position tab

This second spreadsheet includes information on nodes that are part of the continuous non-rectilinear reinforcement of the arch. The main rebars are placed end to end, forming a sole object.

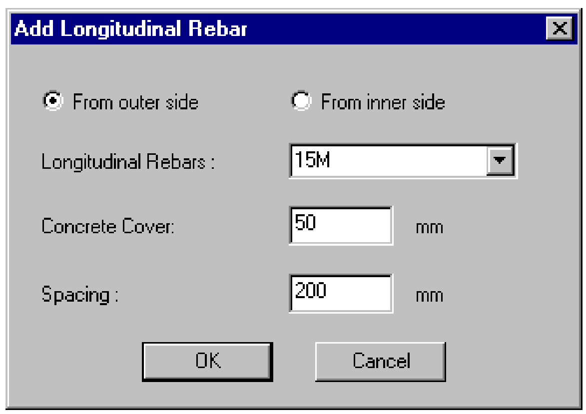
Add a Main Reinforcing Bar

The "Add a main rebar" icon of Edit toolbar

Use this function, available on **Edit** toolbar of *Rebar Placement* window, to graphically add a main rebar right on the elevation view of any continuous system of the *Slab* type with a variable geometry.

Procedure

- Activate the icon on Edit toolbar.
- Use your cursor and click on a first point (beginning of the rebar) and then on a second point (end of rebar) representing the length of the rebar that you want to add. Then, the following dialog box will appear on your screen:



- Choose the location of the bar relatively to local axis system of continuous system (outer or inner side). Select a type of rebar, concrete cover and spacing.
- Click OK.
- Check force and resistance diagrams displayed on your screen.

General Results Spreadsheet

Select this spreadsheet in the **Results** menu to consult numerical results about the concrete design. If lines are marked with a yellow colour, it means that the design, at this location in the continuous system, is not good. Some parameters are exceeding the limits fixed in the building code that was selected in the **Continuous Systems** spreadsheet.

General Results Spreadsheet - S-1

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

719	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
1	A60	0.00	0.01	379.77	89.84	69.66	93.40	121.42	23.66	2.37	27.11
2	A60	159.16	8.71	379.77	114.11	69.66	93.40	121.42	30.05	2.37	27.07
3	A60	318.32	16.03	379.77	136.43	69.66	93.40	121.42	35.92	2.37	27.03
4	A60	477.48	21.98	379.77	166.97	69.66	93.40	121.42	43.97	2.37	26.99
5	A60	636.64	26.62	379.77	197.39	69.66	93.40	121.42	51.98	2.37	26.95
6	A60	795.79	29.99	379.77	220.79	69.66	93.40	121.42	58.14	2.37	26.91
7	A60	954.95	32.15	379.77	237.51	69.66	93.40	121.42	62.54	2.37	26.88
8	A60	1114.11	33.13	379.77	247.92	69.66	93.40	121.42	65.28	2.37	26.84
9	A60	1273.27	32.99	379.77	252.34	69.66	93.40	121.42	66.45	2.37	26.80
10	A60	1432.43	31.77	379.77	251.04	69.66	93.40	121.42	66.10	2.37	26.76
11	A60	1591.59	29.53	379.77	245.29	69.66	93.40	121.42	64.59	2.37	26.72
12											
13	A58	1591.59	29.69	379.77	228.76	69.66	93.40	121.42	60.24	2.37	26.77
14	A58	1600.75	29.54	379.77	228.42	69.66	93.40	121.42	60.15	2.37	26.76
15	A58	1609.91	29.39	379.77	228.06	69.66	93.40	121.42	60.05	2.37	26.76
16	A58	1619.08	29.23	379.77	227.68	69.66	93.40	121.42	59.95	2.37	26.76
17	A58	1628.24	29.08	379.77	227.28	69.66	93.40	121.42	59.85	2.37	26.75

OK | Cancel

Chapter

17

PIERS, ABUTMENTS & RETAINING WALLS GENERATION AND DESIGN

TABLE OF CONTENTS

Chapter 17 Piers, Abutments & Retaining Walls Generation and Design

General

Generation of Abutment, Pier and Retaining Wall

This module generates structural models for piers, abutments, and retaining walls. It applies loads to the generated structure, generates specifications and design criteria. Users only have to define a foundation model and generate load combinations as per CAN/CSA-S6-00 Standard through the **Load Combination Generator**.

Users must be acquainted with VisualDesign. Use the Online Help ([F1] key or **Help** menu) or read appropriate chapters in the User's Manual, such as *Basic Principle, Foundation Design* and *Reinforced Concrete Design*.

See also

Axis System Convention
Pier Components
Abutment Components
Definition of Shallow and Deep Foundation
Generation of Structure
Modification of an existing project
Concrete Specifications
Applying Loads on Supports
Analysis Steps
Step-by-step procedures
Summary of procedures

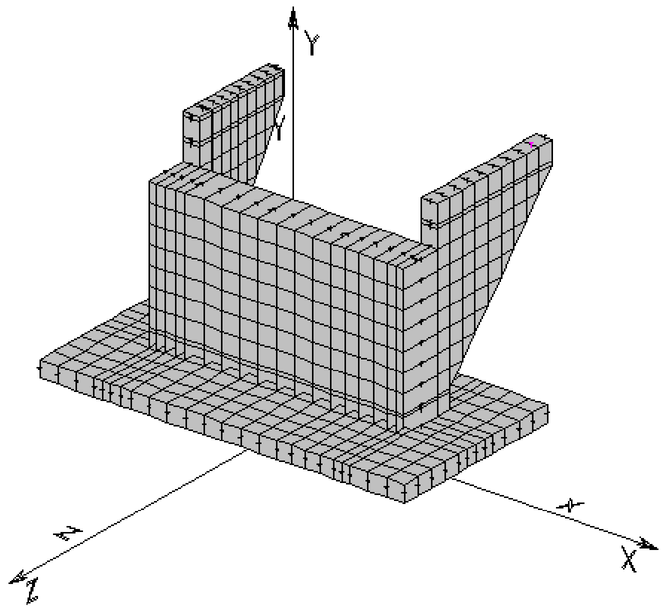
Convention for Global Axes System

In this module, the axis convention is as follows:

The global z-axis is parallel to the road;

The global x-axis is transverse to the road;

The global y-axis is pointing up. Gravity loads are acting in the negative direction.



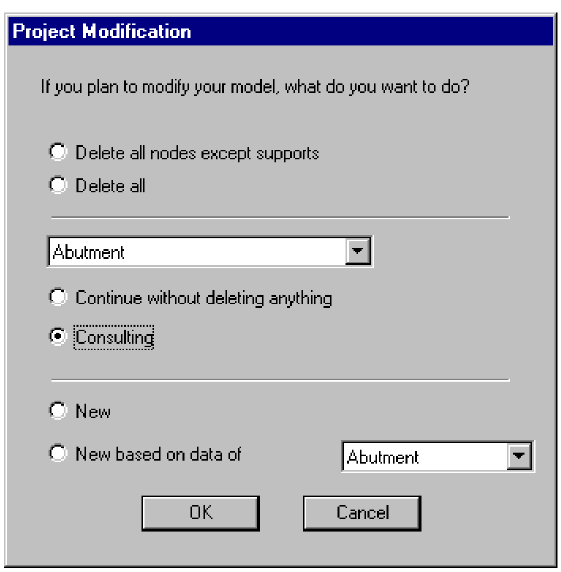
Modification of an Existing Project

To speed up the modeling of a structure or the editing of an existing one, VisualDesign allows consulting, editing, or creating a new project from an existing one. You can select existing elements that you wish to keep in the model.

Procedure

Open the existing project. Then, call up the generator (**Structure** / **Generator / Abutment, Piers and Retaining Walls)**.

The **Project Modification** dialog box will appear on screen.



Options "Delete all" and "Delete all except supports"

A new structure will be generated, and will be based on the name of the structure that is selected in the list box. The data that were entered beforehand in the dialog boxes will remain and they can be modified.

Options "New" and "New based on data"

Elements can be added to an existing model by selecting its name in the list box. The data that were entered beforehand in the dialog boxes will remain and they can be modified.

The name of the model is specified in the **Piles and Abutment** dialog box, when the generator is called up.

Option "Consulting"

Consult an existing project by activating this option. Modifications cannot be done to the model.

Member with a Linear Behaviour

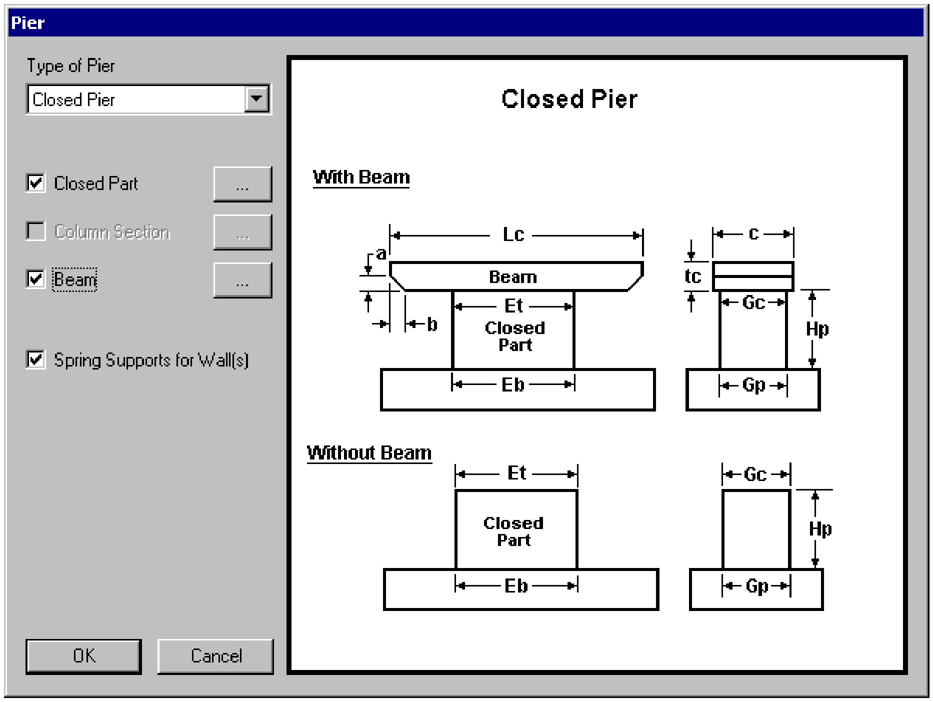
It is possible to define a member having a linear behaviour at all times even for a non-linear analysis.

This functionality is useful to model the small members that are located between bridge supports and pier supports. With a linear behaviour, these members will not induce horizontal components during the non-linear analysis, such as a design. Consequently, only axial forces will be transferred and the convergence will be faster than before.

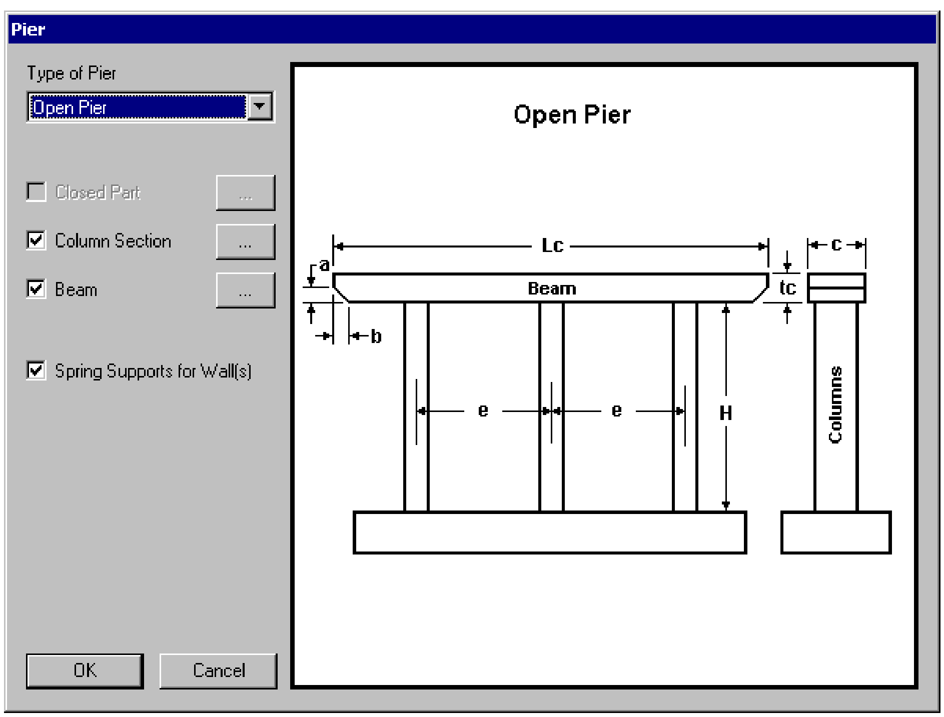
To assign a linear behaviour to these small members, select them while working in the Structure mode, and open the **Member Characteristics** dialog box by pressing the **Properties** icon. Specify hinged end conditions and select the option "Linear only" among the *Behaviour* drop-down list box of the **Member** tab.

Pier Models

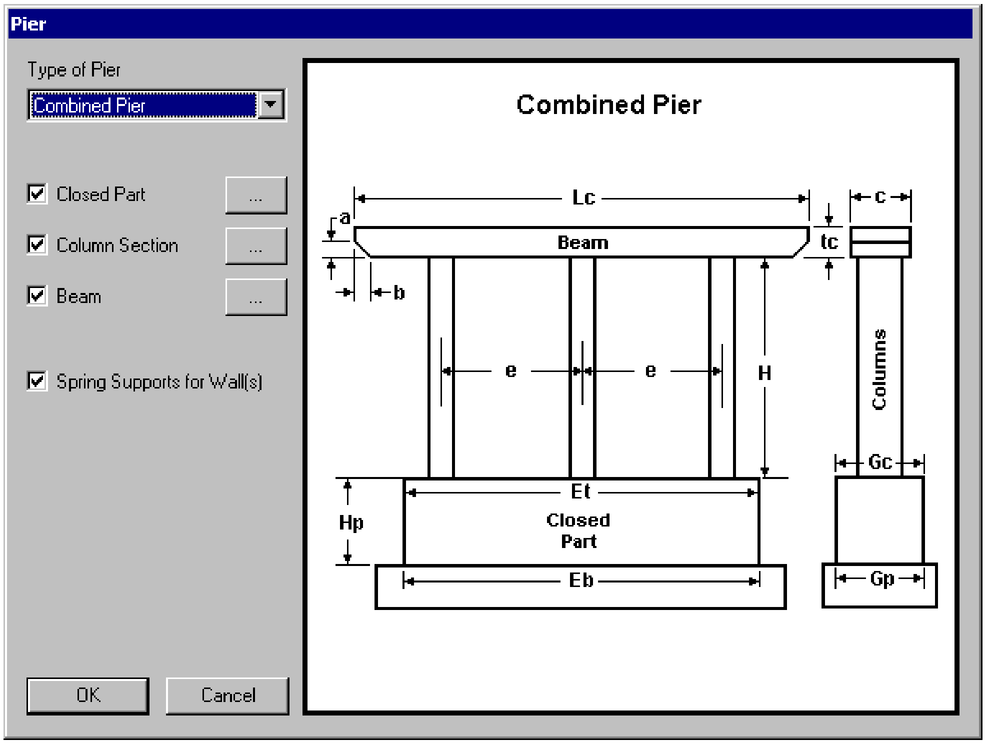
Closed Pier with or without Beam



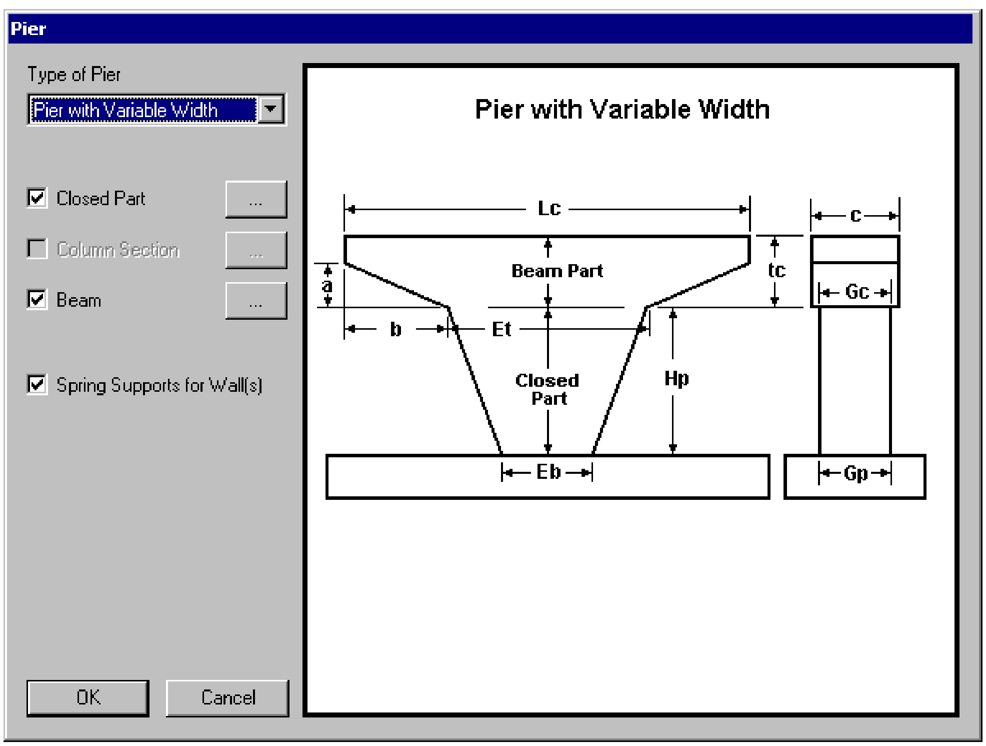
Open Pier



Combined Pier



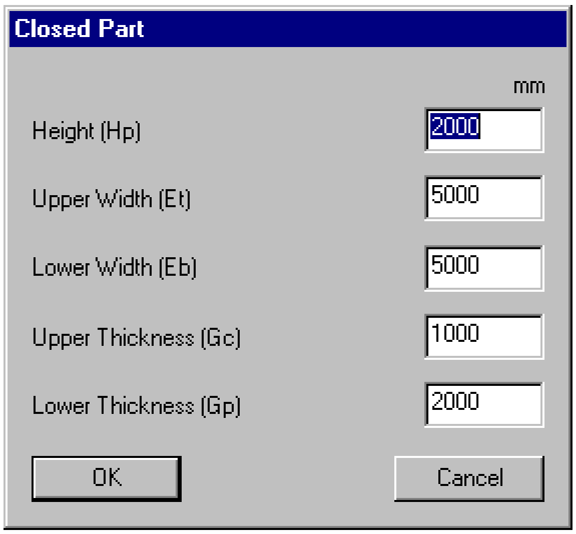
Pier with a Variable Width



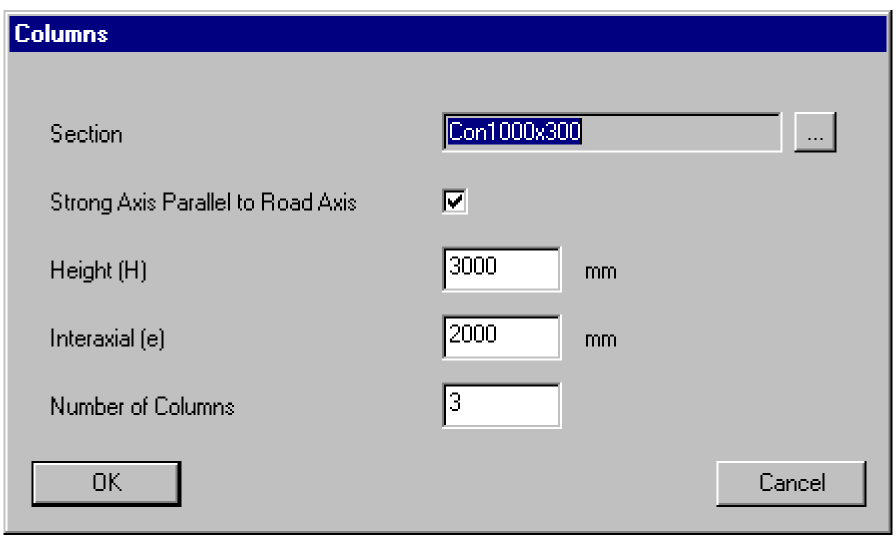
Pier Components

Components for a closed pier, a pier with variable width, an open pier, and a combined pier are defined with the help of specific dialog boxes, which are accessible in the **Pier** dialog box depending on the chosen model.

Closed Part



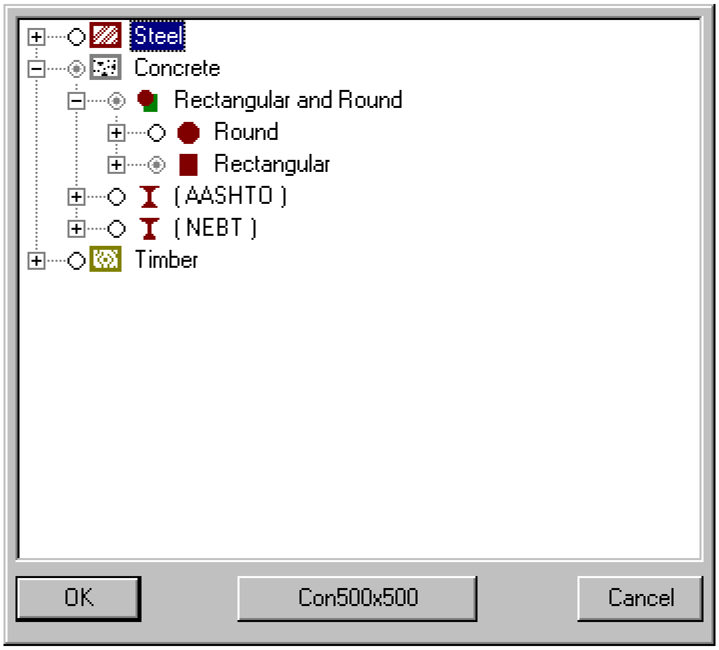
Columns



Column section

To choose a section, press the button […] located next of field ***Section***. The selection tree will appear on your screen, as shown below.

Expand the *Concrete* root, the "Round" or "Rectangular" branches and activate the appropriate radio button. Press OK.



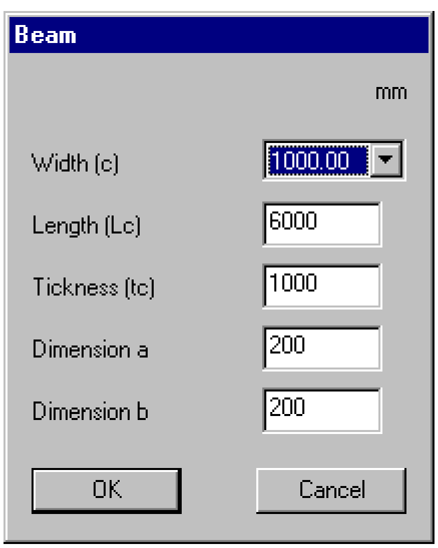
Beam

Beam width "c"

Click the arrow to open the drop-down list box and select the beam width.

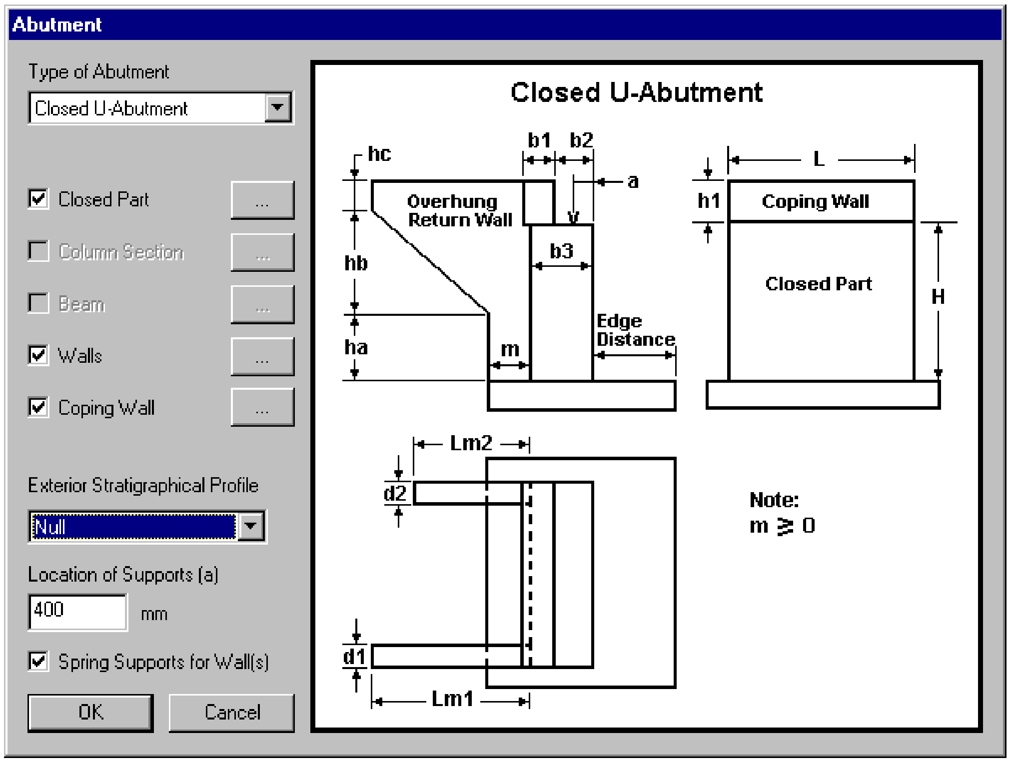
If the dimension is not part of the list, you will have to create a new section. Go to VisualDesign **Common/Sections** menu and select the **Round** or **Rectangular Sections** spreadsheet. Insert a line at bottom of spreadsheet and enter new section dimension "b" and "d". Other section properties are automatically calculated.

Go back to your project and choose the appropriate width.



Abutment Models

Closed U-Abutment



Exterior Stratigraphical Profile

If there is a backfill located at the right side of the structure, the corresponding stratigraphical profile must be specified in this list box. Therefore, a second stratigraphical profile must be defined beforehand in the **Stratigraphical Profiles** spreadsheet.

N. B. The backfill (with or without side slope) located at the left of the structure is always specified through the stratigraphical profile that is chosen in the **Foundation Model Definition** dialog box.

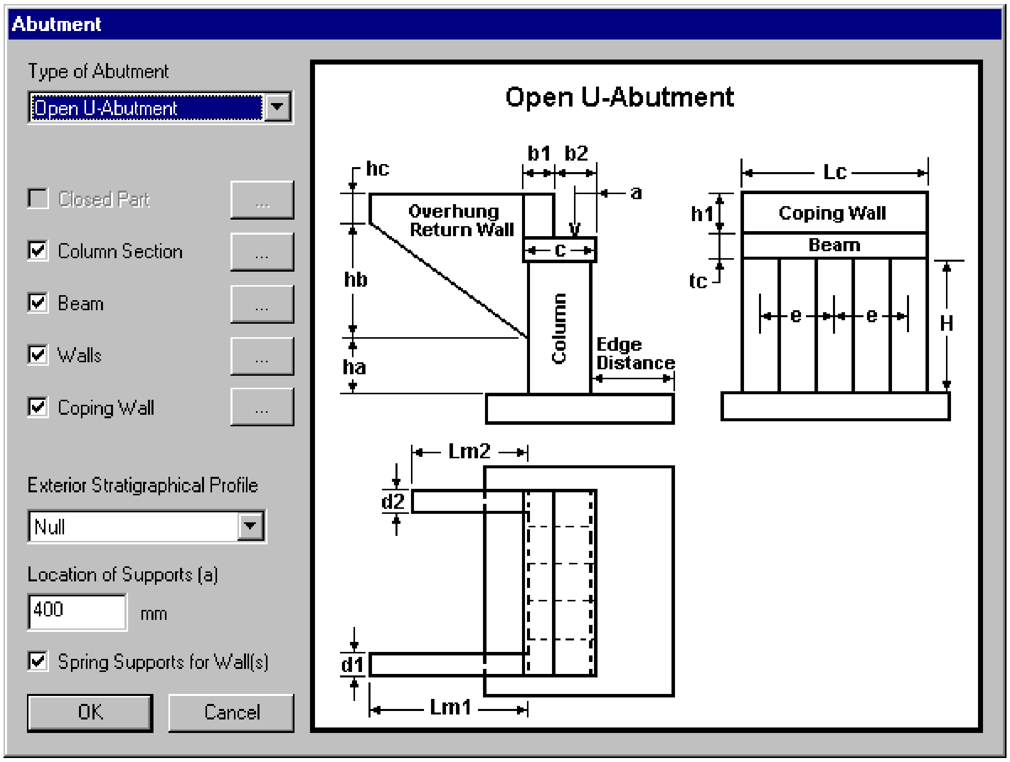
Spring Supports

Activate this option to model spring supports along the backfilled wall.

See also

Abutment Components

Open U-Abutment



Exterior Stratigraphical Profile

If there is a backfill located at the right side of the structure, the corresponding stratigraphical profile must be specified in this list box. Therefore, a second stratigraphical profile must be defined beforehand in the **Stratigraphical Profiles** spreadsheet.

N. B. The backfill (with or without side slope) located at the left of the structure is always specified through the stratigraphical profile that is chosen in the **Foundation Model Definition** dialog box.

Spring Supports

Activate this option to model spring supports along the backfilled wall.

See also

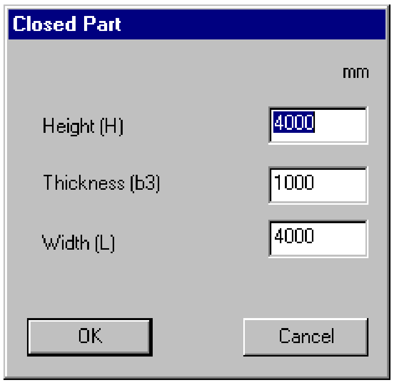
Abutment Components

Abutment Components

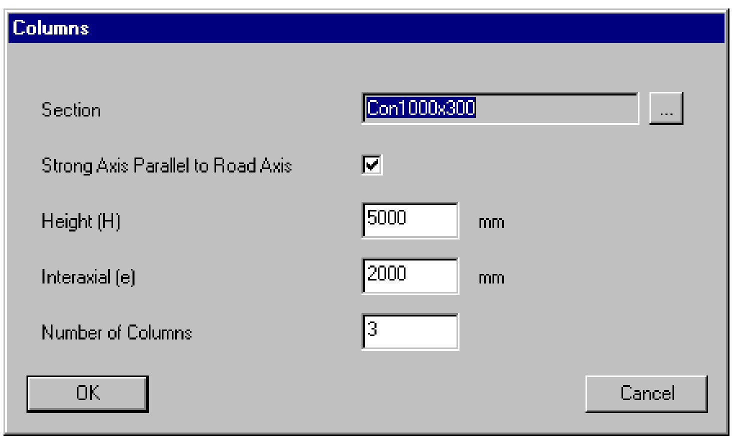
Components for closed and open U-abutments are defined with the help of specific dialog boxes, which are accessible in the **Abutment** dialog box, depending on the chosen model.

Below, you will find a description of each dialog box. Values are default ones.

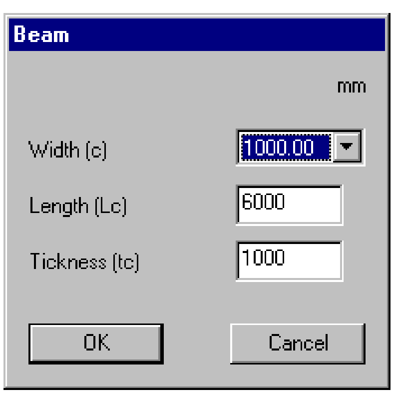
Closed Part



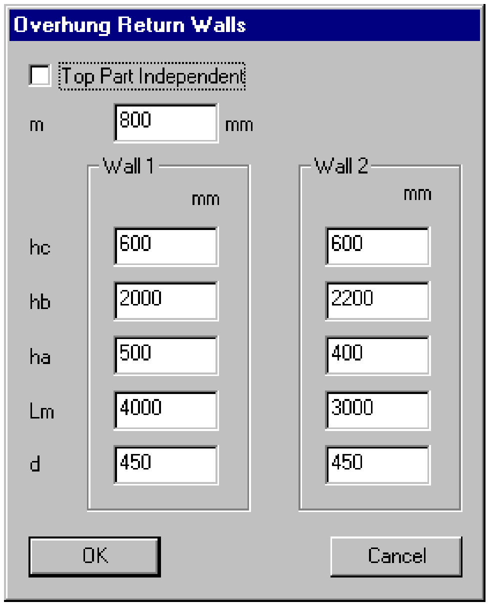
Columns



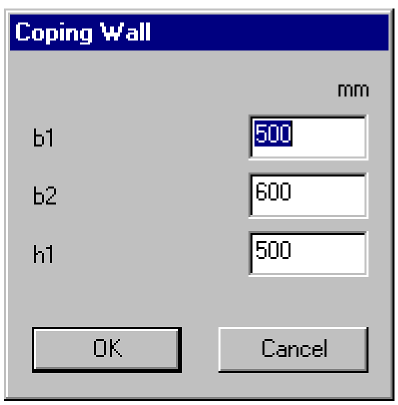
Beam



Overhung Return Walls

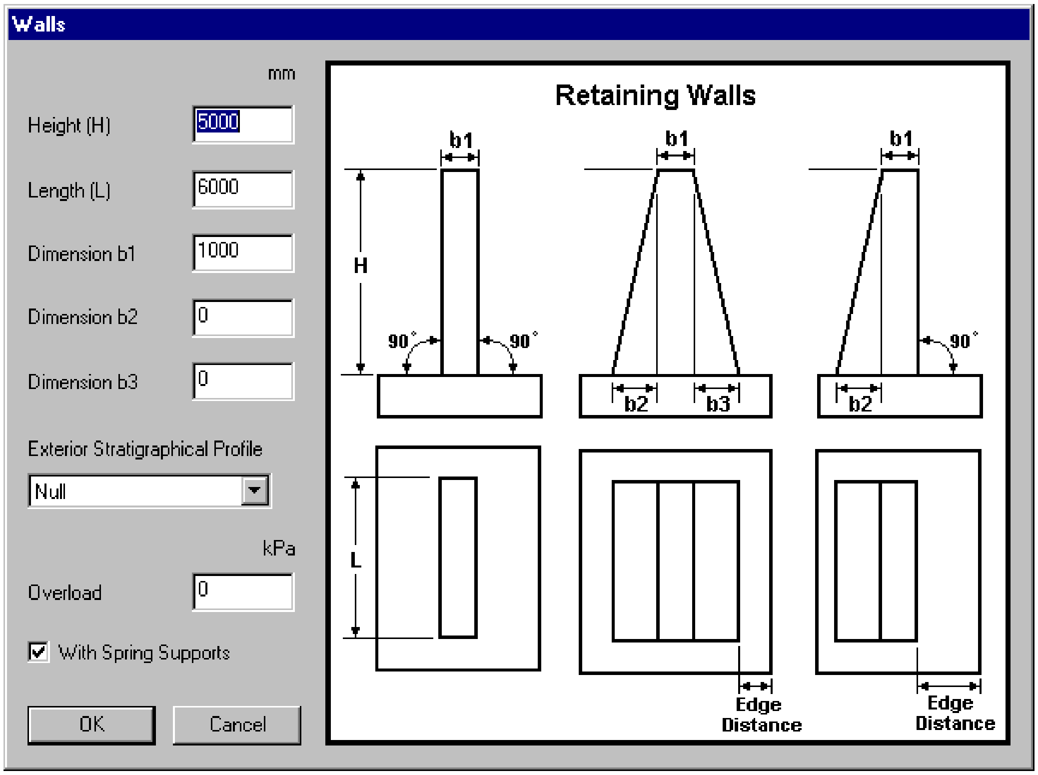


Coping Wall



Retaining Wall Models

Retaining Walls



Exterior Stratigraphical Profile

If there is a backfill located at the right side of the structure, the corresponding stratigraphical profile must be specified in this list box. Therefore, a second stratigraphical profile must be defined beforehand in the **Stratigraphical Profiles** spreadsheet.

N. B. The backfill (with or without side slope) located at the left of the structure is always specified through the stratigraphical profile that is chosen in the **Foundation Model Definition** dialog box.

Overload

Use this field to define an overload over the backfill that is located at the left side of the wall.

Spring Supports

Activate this option to model spring supports along the backfilled wall.

Soils & Stratigraphical Profiles

Soils

Make sure that soils properties in VisualDesign spreadsheets (**Common / Soils**) are corresponding to the geotechnical study obtained for your project. If not, create your own soils in the following spreadsheets, by adding a line at the bottom of the spreadsheet.

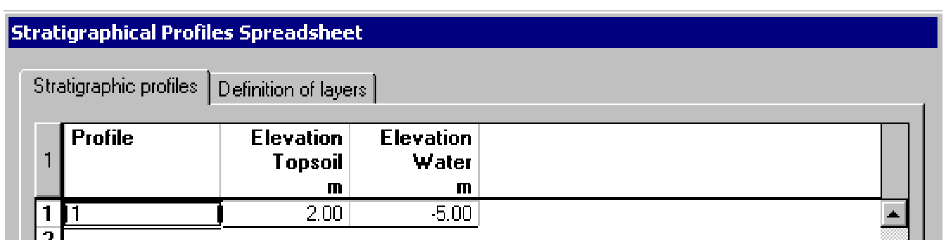
- **Cohesive Soils** spreadsheet;
- **Granular Soils** spreadsheet;
- **Rocks** spreadsheet.

For more details about required soil data, refer to Online Help's Chapter 6 *Foundation Design*.

Stratigraphical Profiles

- Open the **Stratigraphical Profiles** spreadsheet (**Structure** menu). Insert a line and give a name to this new profile. Enter the ground and water table elevation.

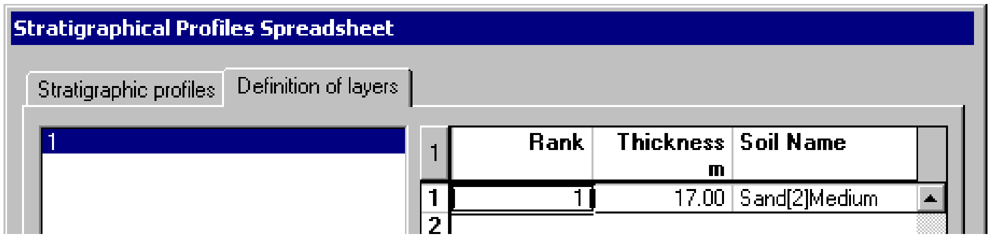
Elevation of Topsoil



Warning! This information is important because you will specify the elevation of the top of footing in the **Abutment, Pier and Retaining Wall** dialog box. The top of footing must coincide with an elevation that is between ground elevation and the bottom of the deepest soil layer.

Definition of Layers

- Select the **Layer Definition** tab.
- Insert as many lines as the number of soil layers included in the profile. (N. B. First rank is located just below natural ground.)
- Enter the rank of each layer. Double-click in the "Soil name" column and choose one among the drop-down list box.



- Press OK to save data entries.

Definition of a Foundation Model

General

You can define the foundation model before generating the structural model.

A foundation model can be created in 2 ways:

Foundation Modeling Wizard

The **Foundation Modeling Wizard** , located in **Structure / Foundation Models**, allows a quick generation of a deep or shallow foundation model. When the generation is completed, data can be access through the **Foundation Model** spreadsheets.

Foundation Model Dialog Box

The foundation model can be define in the **Foundation Model** dialog box, which is accessible through **Structure** / **Foundation Model** / **Shallow** or **Deep**.

The **Foundation Model** dialog box is reached through the **Foundation Model** spreadsheet, using the contextual menu: Insert a line in the spreadsheet, click any cell, right click to open the contextual menu and choose the function ***Detail***.

See also
Shallow Foundation Model
Deep Foundation Model

Shallow Foundation Models

Go to **Structure** /**Foundation Models** / **Shallow** or **Deep.** When the spreadsheet is open, click in a cell, right click, and select the command "Details" in the contextual menu.

The **Shallow Foundation Models** dialog box is composed of four tabs, namely, **Definition of Models**, **Footing**, **Column** and **Design** tabs. Enter foundation parameters and dimensions.

Complete the tabs in the **Shallow Foundation Models** dialog box and press OK. You will go back to the **Shallow Foundation** spreadsheet. Press OK to exit

Please refer to Chapter 6 – Foundation Design to know more about the **Shallow Foundation Models** dialog box.

N. B. The direction of Z-Axis is parallel to the road axis. X-Axis is transverse to the road and Y-Axis represents the gravity axis.

Specification for Shallow Foundation

A specification for the design (or verification) of the shallow foundation must be selected in the **Design** tab of **Shallow Foundation Models** dialog box. This specification is accessible in **Structure / Specifications / Shallow Foundations**.

Shallow Foundation Specifications Spreadsheet

2	Number	Code	Type of analysis	Bx max m	Bz max m	Optimize a Dimension	Saf	Waf	Rebars Material
1	Footing-Design	CAN/CSA-A23.3-95	Design	6.00	6.00	Bx = Bz	1.25	1.50	G30.18-400R
2	Found.- Check.	CAN/CSA-A23.3-95	Verification	6.00	6.00	n/a	1.25	1.50	G30.18-400R

Piles Foundation Models

The **Deep Foundation Models** dialog box is composed of five tabs, namely, **Definition of Models**, **Footing**, **Column, Piles** and **Piles Layout** tabs.

Complete the tabs in the **Deep Foundation Models** dialog box and press OK. You will go back to the **Deep Foundations** spreadsheet. Press OK to exit.

Please refer to Chapter 6 – Foundation Design to know more about the **Deep Foundation Models** dialog box.

Specification for Deep Foundation

A steel specification is required for the verification of the structural capacity of the steel piles. It must be selected in the **Deep Foundation Specifications** as shown below. The geotechnical capacity of piles will be checked according to the specified pile length and parameters entered in the **Piles** tab of **Deep Foundation Model** dialog box.

Deep Foundation Specifications Spreadsheet

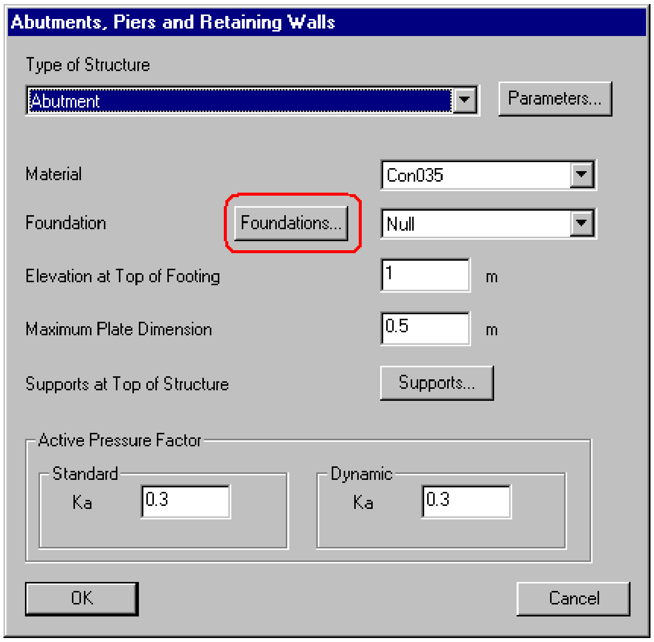
1	Number	Type of analysis	Max. L m	Steel specification
1	Piles	Verification	15.00	S16-01-Verif.
2				

Assigning the Foundation Model to the Structure

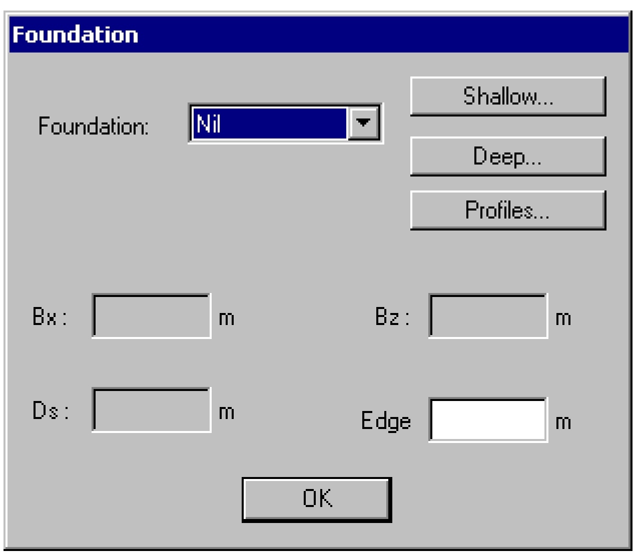
The foundation model is assigned to the structure through the **Abutments, Piers, and Retaining Walls** dialog box. Click the arrow to open the "Foundation" list box and select a model.

The "Foundation" button

If no model has been defined, click the "Foundation" button, as shown below:



The **Foundation** dialog box opens.



The *Shallow*, *Deep*, and *Profiles* Buttons

These buttons opens respectively the **Shallow Foundations Models** spreadsheet, the **Deep Foundations Models** spreadsheet, and the **Stratigraphical Profiles** spreadsheet.

- Click the "Profiles" button first because a profile must be assigned to a foundation model (shallow or deep). When the stratigraphical profile will be defined, you will go back to the **Foundation** dialog box.
- Click the "Shallow" or "Deep" button to open the **Shallow Foundations Models** spreadsheet or the **Deep Foundations Models** spreadsheet. When the foundation model will be defined, you will go back to the **Foundation** dialog box.
- Select the name of the foundation model through the "Foundation" list box. Footing dimensions and thickness will be indicated.

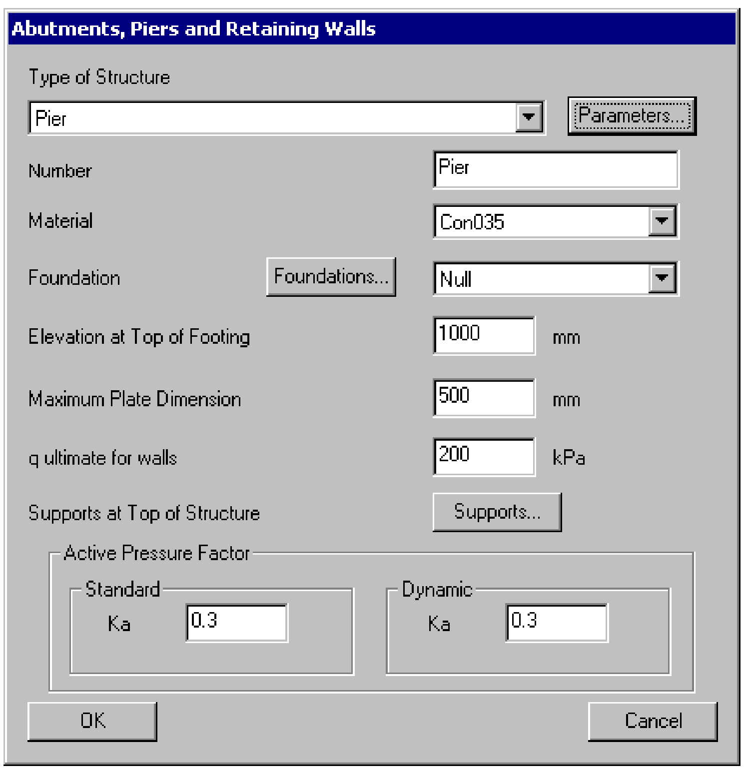
See also

Shallow Foundation
Deep Foundation

Generator

Generation of the Model

- Create a new file (**File / New**).
- Define the foundation models that you need for your project.
- Go to **Structure / Generator / Abutments, Piers, and Retaining Walls**. The **Abutments, Piers, and Retaining Walls** dialog box will open on screen
- Select a type of structure among the "Type of Structure" list box.
- Click the "Parameters" button. Select a specific model (abutment or pier) in the Abutment dialog box, Pier dialog box or Retaining Wall dialog box and enter the dimensions of selected model. You will go back to the **Abutments, Piers and Retaining Walls** dialog box.



- Give a number to this model. (If you call up the **Project Modification** dialog box afterwards, this number allows consulting input data for this model, or regenerating another model from this one).
- Select a concrete material.

- Select the foundation model among the drop-down list box. If the foundation is not yet defined, press the "Foundations…" button.
- In the **Foundation** dialog box. use the *Profiles*, *Shallow* or *Deep* buttons to define a stratigraphical profile and a shallow or deep foundation model. Then, select the model.
- Still in the **Abutments, Piers, and Retaining Walls** dialog box, enter elevation at the top of footing. (This elevation must be compatible with elevations specified in the **Stratigraphical Profiles** spreadsheet.).
- Specify a value for q ultimate.

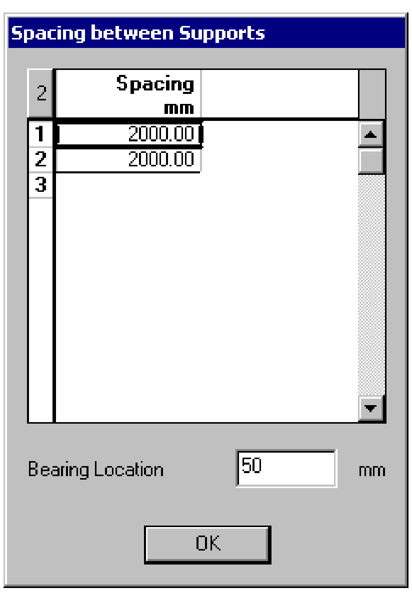
q ultimate for foundation walls

The value of *q ultimate* is used to generate elastic supports for the foundation wall when the foundation is transformed into finite elements of plates. At this stage, the value of q ultimate will be posted in a dialog box. VisualDesign will display the average of *q ultimate* calculated values for different load combinations. If no analysis has been carried on, the value will be the one entered in the **Soils** spreadsheet. If there is no value, you must enter a value different from zero.

VisualDesign will create a mesh with plate elements of maximum dimensions that correspond to the value that is entered in the "Maximum Plate Dimension". Default value is 0.5 m.

Definition of Supports

- Click the "Supports" button to specify the spacing of supports at the top of the beam. Insert lines and enter the spacing between supports. Specify the thickness of steel plate or grout. Press OK.

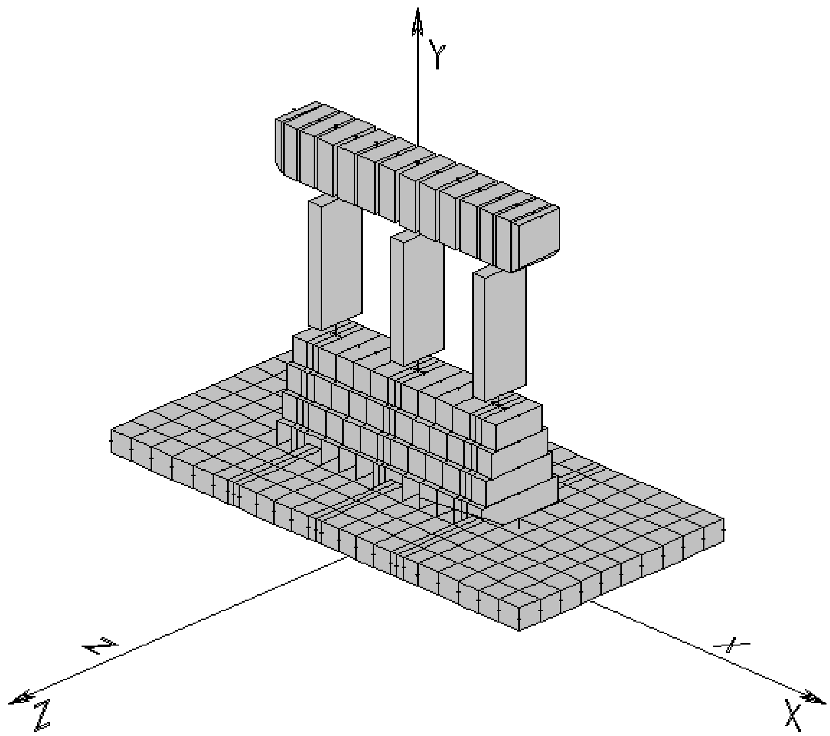


Active Pressure

- Enter *Ka* – values for a standard active pressure and dynamic one, if required.
- Click OK to generate the model and apply loads.

Generated Structure

Here is a 45 deg. view of a combined pier along with shallow foundation model.



Generated Concrete Specifications

VisualDesign generated three concrete specifications that will be used for the design of the beam, columns, footing, closed part and walls. To modify default values for specifications, go to **Structure / Specifications / Concrete**.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Beam-slab | Slab FE | Shear Wall

11	Number	Code	Type of analysis	Maximum Capacity Factor %	Calcul. Method Mr/Vr Positive	Calcul. Method Mr/Vr Negative
7	Beam	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
8	Column	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
9	Footing	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
10	Closed	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
11	Walls	CAN/CSA-S6-00	Design	100.00	Maximize Mr	Maximize Mr
12						

By default, the design is done according to Standard CAN/CSA-S6-00. By default, main rebars 20M, 25M, 30M and 35M, and stirrups 10M and 15M can be selected for the design.

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Beam-slab | Slab FE | Shear Wall

11	Epoxy Coated	Optimization Main reinforcement	Main Reinforcement Material	Selection of Main Reinforcement	Design method
7	[ ]	Weight	G30.18-400R	20M 25M 30M 35M	General method
8	[ ]	Weight	G30.18-400R	20M 25M 30M 35M	General method
9	[ ]	Constant "s" (M+ & M-)	G30.18-400R	20M 25M 30M 35M	General method
10	[ ]	Weight	G30.18-400R	20M 25M	General method
11	[ ]	Weight	G30.18-400R	15M 20M	General method
12					

Beams and columns

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Beam-slab | Slab FE | Shear Wall

11	Maximum no. of layers in tension	Maximum no. of layers in compression	Optimization Transverse reinforcement	Transverse Reinforcement Material	Selection of Transverse Reinforcement	Structure	Supports for stirrups
7	4	4	Weight	G30.18-400R	10M 15M	Standard	15M
8	4	4	Weight	G30.18-400R	10M 15M	Standard	15M
9	4	4	None	G30.18-400R	10M	Standard	15M
10	4	4	None	G30.18-400R	10M	Standard	15M
11	4	4	None	G30.18-400R	10M	Standard	15M
12							

Slabs (Finite elements)

Concrete Specifications Spreadsheet

General | Beam / Column / Joist | Beam-slab | Slab FE | Shear Wall

11	ID	Number	Temperature reinforcement
7	22	Beam	15M
8	23	Column	15M
9	24	Footing	15M
10	25	Closed	15M
11	26	Walls	15M
12			

Generated Continuous Systems

VisualDesign automatically creates continuous systems for abutment, piles, and retaining walls. Concrete specifications are also initialized in the **Continuous Systems** spreadsheet.

You will find the *Beam* type and *Column* type of continuous systems.

Continuous Systems Spreadsheet

4	Number	Specification	Type	Interaction	Exposure Top	Top Cover mm	Exposure Bottom	Bottom Cover mm	Exposure Left
1	Column+0	Column	Beam/Column	Bending / Compression	Manual	75.00	Manual	75.00	Manual
2	Column+1	Column	Beam/Column	Bending / Compression	Manual	75.00	Manual	75.00	Manual
3	Column-1	Column	Beam/Column	Bending / Compression	Manual	75.00	Manual	75.00	Manual
4	Beam	Beam	Beam/Column	Bending	Manual	60.00	Manual	40.00	Manual

Please refer to Chapter on Reinforced Concrete Design to know more about continuous systems.

Generated Loads

Go to **Loads / Load Cases / Definition.** You will notice that VisualDesign has automatically created the "Prefab Components", "Backfill", "Active", and "Hydrostatic" loads and other in the **Loads Definition** spreadsheet. The "Hydrostatic" load will be included if the elevation of water table is above natural ground. The "Active" load case is the active earth pressure and earth pressure at rest acting on walls. The "Backfill" load case is the backfill pressure on walls.

Loads Definition

Load Case | Dead | Live | Dynamic | Ice | Temperature

9	Number	Type	Family	Stage	A
1	Morte	(D1) Prefab Components	N/A	Stage n/a	
2	Backfill	(D4) Backfill	N/A	Stage n/a	
3	Slant Wall	(D4) Backfill	N/A	Stage n/a	
4	Hydro.	(E5) Hydrostatic pressure	N/A	Stage n/a	
5	Active	(E23) Pressure - At rest + active	N/A	Stage n/a	
6	Backfill Int	(D4) Backfill	N/A	Stage n/a	
7	Backfill ext	(D4) Backfill	N/A	Stage n/a	
8	Active dyn.	(EQ) Seismic	N/A	Stage n/a	
9	Coping Wall	(D1) Prefab Components	N/A	Stage n/a	
10					

Adding Load Cases

Insert lines in this spreadsheet and define the load case titles and types.

N.B. The **Load Combination Generator** generates load combinations according to the types of load cases included in the Load Definition spreadsheet.

Applying Loads to Supports

Activate the Load Case mode on Activation toolbar and select a load case title in the drop-down list box.

Activate the "Node" icon on Elements toolbar and select supports. Press the **Properties** icon to call up the **Forces on Nodes** spreadsheet. Enter loads according to global axis system.

Load Combinations

Load Combination Generator

To call up the generator, go to **Loads** / **Load Combinations** / **Automatic Generation**. Select *S6-00 Standard* in the "Code" list box. Press next.

In the **Specific Options** page, select generation options and press the "Next" button.

The **Selection** dialog box includes the load combinations that VisualDesign plans to generate for you. To remove load combinations from the list, uncheck the appropriate boxes. Press the "Finish button".

The **Load Combinations** spreadsheet appears on screen. To remove a load combination temporarily from analysis, disable it through the *Required* option. Press OK to save data and exit the spreadsheet.

See also from Chapter 4:
Load Combinations spreadsheet
Load Factors tab

Analysis and Design

Step-by-Step Procedure

- Create a new file by selecting **New** in **File** menu.

Foundation Model

- Open one the **Soils** spreadsheet (**Common** / **Soils)** and make sure that the parameters are corresponding to the geotechnical survey. If not, create a new soil by inserting a line and enter data. The bearing capacity will be calculated according to the soil parameters. "***q ult***." represents the ultimate bearing capacity of the soil if it is known.

- Define a stratigraphical profile and specify each layer of soil.

- Define a foundation model through dialog boxes OR use the **Foundation Modeling Wizard.**

Generation of the Model

- Start the generator: Go to **Structure** / **Generator / Abutment, Piers and Retaining Walls**.

- Select a structure in the **Piles and Abutments** dialog box and specify other parameters. Press the "Parameters" button and enter dimensions.

- Add other load titles in the **Load Definition** spreadsheet, if needed, and apply loads on the structure.

Generation of Load Combinations

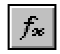
- Generate load combinations (**Loads** / **Load Combinations** / **Automatic Generation**).

Static Analysis of Foundation

- Launch a static analysis to compute the foundation bearing capacity. Look at results through spreadsheets (**Results** / **Foundations / Footings**) and the **View Options** dialog box.

  If the foundation results are not good, you must generate a new model with appropriate footing dimensions and thickness and launch another static analysis.

- If the foundation design is correct, save your file under another name before transforming the foundation into plates. Otherwise, you will not have access to foundation results (bearing capacity) anymore.

Press the Static Analysis icon  on Tools toolbar. Then, press the "Analyse" button in the **Static Analysis** dialog box. VisualDesign will calculate the foundation bearing capacity.

Look at Footing Results (Bearing Capacity)

Look at foundation results by selecting **Results** / **Foundations** / **Footings**. If yellow marked lines are present in the spreadsheet, look carefully at results. Yellow lines mean that the design in not OK.

Use the graphic options: Open the **View Options** dialog box. In the **View** tab, activate the "Foundation" option in the *Foundation* section. Then, select the **Results** tab (**View Options**) and activate the "Structural Design Load" option.

It is of no use to open the **Reinforcement** spreadsheet. The design of the footing will be done during the analysis and design process.

If the footing dimensions (bearing capacity) are insufficient, you must generate a new model with appropriate footing dimensions and/or thickness and launch the static analysis again (or design). This is an iterative process.

If the dimensions are OK, save your file under another name to keep results for further consultation. ***If you do not rename the file before you transform the foundation into plates, bearing capacity results will be lost*** because VisualDesign will consider the project as a new one.

Transform the Foundation

VisualDesign will transform the footing into plates and add elastic supports at foundation nodes. For pile foundation, the program will also transform the pile into members and create rigid links between pile head and footing.

To do all of this in one click, activate the Structure mode, select the foundation central support and click on the **Foundation Transformation** function in **Structure** / **Tools** menu.

Calculation of Elastic Supports dialog box:

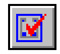
VisualDesign will display the average value of *q ultimate* that were calculated for different load combinations. If no analysis has been carried on, the value will be the one entered in the **Soils** spreadsheet. If there is no value in this field, enter a value different from 0.

See Foundation Transformation

Reinforced Concrete Design

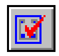
Press the **Analysis and Design** icon  of Tools toolbar to launch the reinforced concrete design for beam, columns and 2-way slabs. Press the "Analyse" button in the **Analysis and Design** dialog box. Close the dialog box when analysis is completed.

Results for slabs (or simple walls)

Activate the Rebar Placement  mode and select a group of plates on Activation toolbar.

Open the **View Options**. Select the **FE Results** tab and activate the layers of rebars you want to visualize. Disable the display of plate surfaces and outline to have a better look. Print the rebar placement and bar list.

Results for Beams and Columns

Activate the Rebar Placement  mode and double-click on a continuous system.

Call up the **View Options** dialog box  and display dimensions, rebar details, force and resistance diagrams. Refer to the ***Reinforced Concrete Design*** chapter of On-line Help.

See also

Static Analysis
Group of Plates - Surfaces
Design of 2-Way Slabs
Foundation Design Module
View Options (Main menu)
Reinforced Concrete Design Module
Rebar Placement window
View Options (Rebar Placement window)