

VecTor3:

A. User's Manual

***B. Sample Coupled Thermal and
Structural Analysis***

Fady ELMohandes

Frank J. Vecchio

November 2013

Table of Content

VecTor3	i
Table of Content	ii
A. User's Manual	1
A.1. Introduction.....	2
A.2. Modelling Main Steps.....	4
A.2.1. Starting-up.....	4
A.2.1.1. Display Features.....	6
A.2.2. Material Definition.....	10
A.2.2.1. Base Material Properties Definition.....	10
A.2.2.2. Discrete Reinforcement Material Properties Definition	18
A.2.2.3. Bond Properties Definition	21
A.2.3. Structure Definition.....	24
A.2.3.1. Node Definition.....	25
A.2.3.1.1. Defining Node Coordinates	25
A.2.3.1.2. Applying Node Restraints	27
A.2.3.1.3. Applying Node Constraints (Linking Nodes)	28
A.2.3.2. Element Definition	30
A.2.3.3. Material Type Assignment.....	34
A.2.4. Job Definition.....	36

A.2.4.1. Analysis Job Control Parameters Definition.....	37
A.2.4.2. Structural Analysis Models Definition	46
A.2.4.3. Auxiliary Parameters Definition	53
A.2.4.4. Auxiliary Thermal Parameters Definition.....	57
A.2.5. Load Definition	63
A.2.5.1. Nodal Loads	65
A.2.5.2. Support Displacements (Imposed Displacements)	66
A.2.5.3. Gravity Loads.....	67
A.2.5.4. Elements' Temperatures	68
A.2.5.5. Concrete Elements' Prestrains	69
A.2.5.6. Concrete Elements' Ingress Pressures	70
A.2.5.7. Concrete Elements' Surface Temperatures (Nodal Thermal Loads)	71
A.2.5.8. Lumped Nodal Masses (for Dynamic Analysis).....	74
A.2.5.9. Impulse Forces	75
A.2.5.10. Ground Acceleration	77
A.2.6. Saving the Input Files and Running VecTor3.....	79
B. Sample Coupled Thermal and Structural Analysis	82
B.1. Test Introduction	83
B.2. Modelling Main Steps	87
1 Starting-up.....	88

B.2.1. Material Definition.....	89
B.2.1.1. Base Material Properties Definition.....	89
B.2.1.2. Discrete Reinforcement Material Properties Definition.....	90
B.2.1.3. Bond Properties Definition.....	94
B.2.2. Structure Definition.....	94
B.2.2.1. Node Definition.....	94
B.2.2.1.1. Defining Node Coordinates.....	94
B.2.2.1.2. Applying Node Restraints	95
B.2.2.1.3. Applying Node Constraints (Linking Nodes)	98
B.2.2.2. Element Definition	100
B.2.2.3. Material Type Assignment	104
B.2.3. Job Definition.....	106
B.2.3.1. Analysis Job Control Parameters Definition.....	107
B.2.3.2. Structural Analysis Models Definition.....	112
B.2.3.3. Auxiliary Parameters Definition	115
B.2.3.4. Auxiliary Thermal Parameters Definition.....	116
B.2.4. Load Definition	119
B.2.5. Saving the Input Files and Running VecTor3	129
References.....	133
Appendix A : Sample VECTOR.EQR File	141

VecTor3:

A. User's Manual

F. ElMohandes

November 2013

A.1. Introduction

This user's manual provides a step-by-step procedure for carrying out structural analysis of reinforced concrete structures using VecTor3 with the aid of its pre-processor, Formworks-Plus. This is followed by a step-by-step procedure for a sample coupled thermal and structural analysis in Section B.

First, a brief technical background on how VecTor3 operates needs to be presented. VecTor3 is a finite element computer program for nonlinear analysis of three-dimensional reinforced concrete structures. It was originally created by Selby (1990) at the University of Toronto and was extensively further developed by ElMohandes (2013).

The input of VecTor3 is in the form of ASCII text files, which can be created manually or with the aid of the pre-processor, Formworks-Plus (Sadeghian, 2012; Wong et al., 2013). There are five main input files that are required for any analysis job. The first file is titled 'VECTOR.JOB' and is responsible for defining the analysis job control parameters and the main structural mechanics models to be followed by concrete and steel in the structural analysis.

The second input file, titled 'VT3.AUX', is responsible for defining additional parameters that are specific to special types of structures, loading conditions, and analysis types. The third input file, titled 'VT3T.AUX', is responsible for defining the main parameters, properties, and structural mechanics models that control heat transfer analysis and the structural analysis in coupled thermal and structural analysis.

The fourth input file is the structural input file. It has the extension 'S3R' and it is responsible for defining the geometry and materials of the structure to be analyzed. The fifth input file is the load file. It has the extension 'L3R' and it is responsible for defining the loads to be applied on the structure to be analyzed for a specific loading case. VecTor3 allows for up to five separate loading cases (of similar or different types) to be analyzed simultaneously. Thus, up to five files with the extension 'L3R' can be created for an analysis.

It should be noted that the structural 'S3R' files and the load 'L3R' files can have any arbitrary names, but the name chosen for each file has to be entered in its specific location in the 'VECTOR.JOB' file in order to identify the specific structure and the specific loading cases to be analyzed in the analysis job defined in the 'VECTOR.JOB' file.

Formworks-Plus is a universal pre-processor for the entire VecTor software suite, including VecTor2, VecTor3, VecTor4, VecTor5, and VecTor6. It was created by Wong et al. (2013) and developed further by Sadeghian (2012). A post-processor, Janus, which was developed by Chak (2013) and Jodai (2013), is also available for graphical display of the results of the structural analysis carried out using VecTor3.

Formworks-Plus provides a graphical interface for the user to facilitate the process of creating the input files of the analysis job. It converts the graphical user input into the aforementioned five types of input ASCII text files and provides shortcuts to start the analysis. This user's manual provides the main steps to be followed in Formworks-Plus to create an analysis job of a reinforced concrete structure to be analyzed using VecTor3. These steps can be summarized in the following list:

1. Starting-up
2. Material Definition
3. Structure Definition
4. Job Definition
5. Load Definition
6. Saving the Input Files and Running VecTor3

To facilitate the process of defining the structural analysis job for the user, default values and models have been predefined for most of the entries required for the analysis. The default values and models have been selected based on the most commonly used values and models or based on the recommendations of the developers of VecTor3. These recommendations are based on experience on the values and models that yield the most reasonable agreement between the analyses and the experimental results.

A.2. Modelling Main Steps

A.2.1. Starting-up

As Formworks-Plus is started, Figure A-1 appears as an introductory presentation of the program and its credentials. Since Formworks-Plus is a universal pre-processor for the entire VecTor program suite, when the user clicks on the figure shown in Figure A-1, they are given the option to choose the type of structure to be analyzed from six different available types as shown in Figure A-2. This basically decides which VecTor program will be used. The list includes:

1. Plane Membrane (2-D): using VecTor2 program.
2. Solid (3-D): using VecTor3 program.
3. Shell: using VecTor4 program.
4. Plane Frame (2-D): using VecTor5 program.
5. Axisymmetric Solid: using VecTor6 program.
6. Mixed Type: using a variety of elements available in all VecTor programs linked together using Cyrus program.



Figure A-1

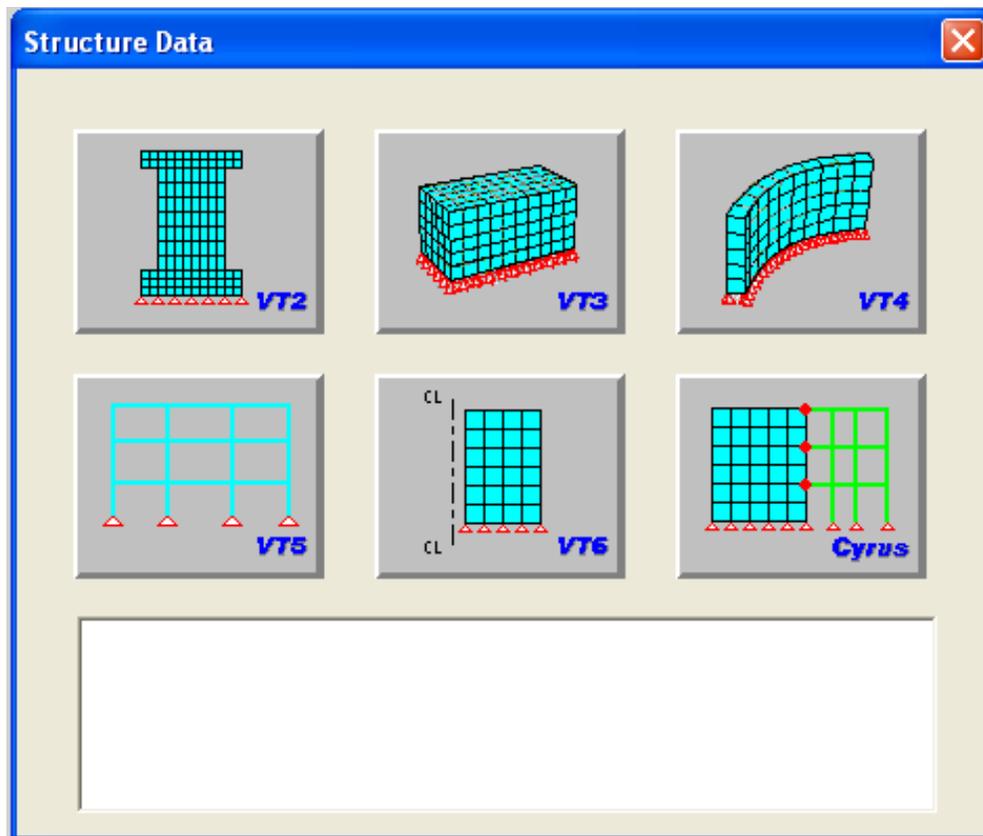


Figure A-2

For analyzing three-dimensional structures using VecTor3, the user is required to choose the second option 'VT3' for 'Solid (3-D)'. This introduces the standard user interface of Formworks-Plus shown in Figure A-3. At this point, the pre-processor is ready for the analysis job definition to be started by the user. The buttons in the toolbar shown in Figure A-4 are responsible for all the user-input data required for the analysis job definition as will be shown in the following steps. The workspace view in the lower part of the screen is responsible for displaying the structure and the loading profiles as they are defined by the user.

At any point, the analysis job definition, including all the structural and loading profiles that have been defined, can be saved by pressing the button shown in Figure A-5. The file to be saved has the extension 'FWP', and it can be reopened and updated at any later time.

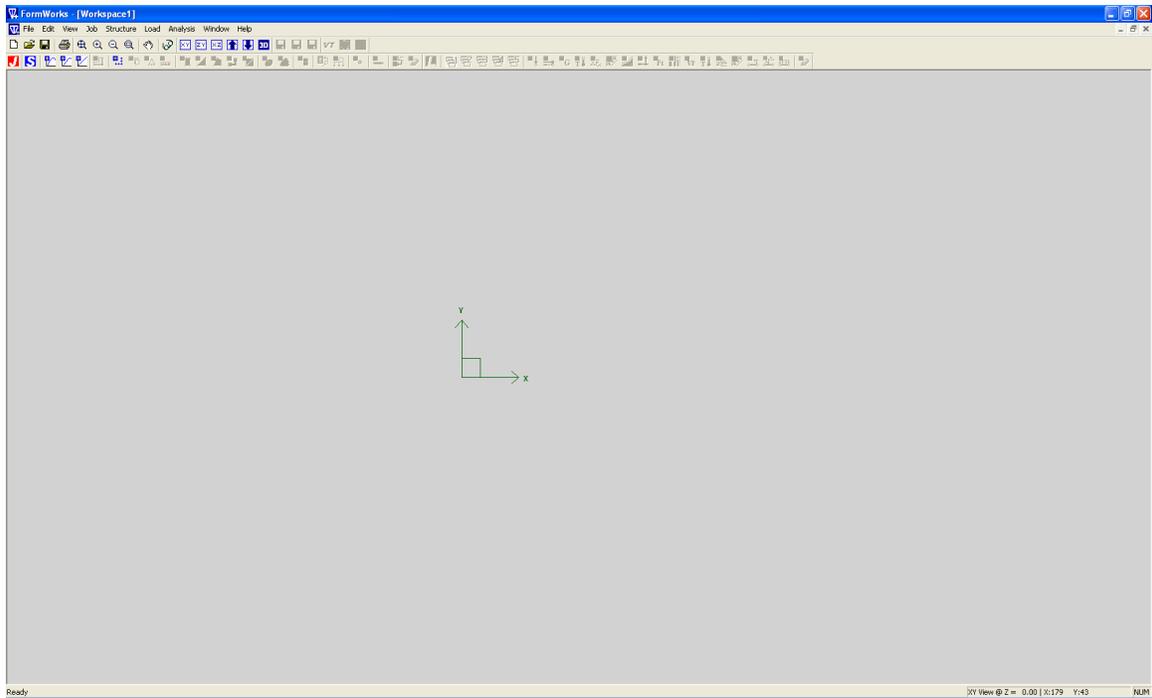


Figure A-3



Figure A-4



Figure A-5

A.2.1.1. Display Features

Formworks-Plus provides two- and three-dimensional graphical representation of the structural model defined by the user. This helps the user to visualize the structure and the loading profiles as they are defined. With the definition of every structural component or loading condition (as will be explained in the following steps), a graphical representation is displayed in the workspace view.

Formworks-Plus also provides many features that facilitate the control of the user over the properties of the graphical representation to be displayed. These features can be accessed at any point through the analysis job definition through the buttons of the toolbar shown in Figure A-6.



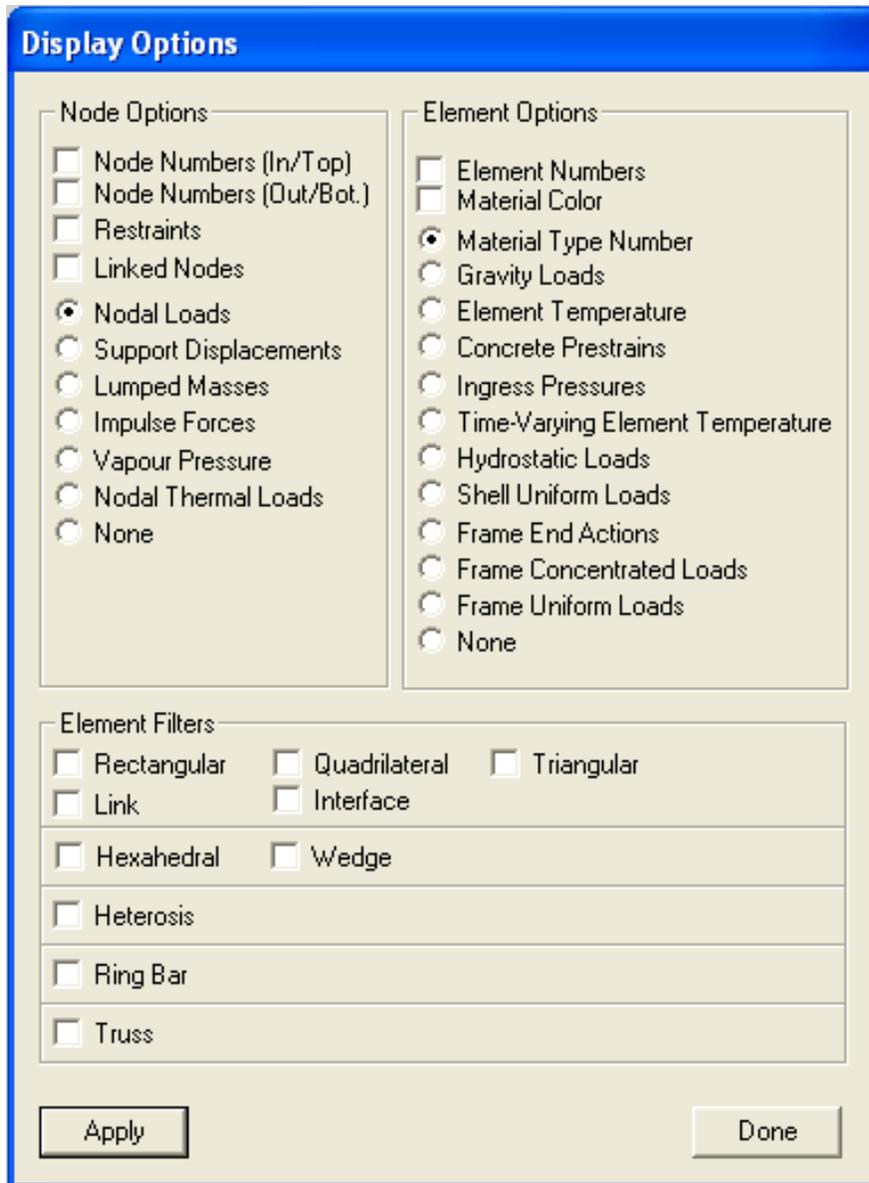
Figure A-6

The first button from the left, titled 'Zoom All', allows the user to view the entire structural model in the workspace view. This can also be done without pressing the 'Zoom All' button by double-clicking the left mouse button. The second and third buttons, , titled 'Zoom In' and 'Zoom Out', respectively, allow the user to incrementally view the structural model in a larger or smaller scale, respectively, with increments of $\pm 10\%$ in size with each press of a button. This can also be done without pressing those buttons by rolling the mouse scroll wheel inwards for a larger display scale or outwards for a smaller display scale. The fourth button, titled 'Zoom Window', allows the user to increase the display scale of a specific part of the structural model by pressing down the left mouse button and dragging a window around the area of interest. The fifth button, titled 'Pan', allows the user to move around in the workspace view by holding down the left mouse button while dragging the mouse in the direction of interest. This can also be done without pressing the 'Pan' button by holding down the mouse scroll wheel while dragging the mouse in the direction of interest.

The sixth button in the toolbar shown in Figure A-6, whose shape is presented in Figure A-7, opens a window titled 'Display Options'. This window is shown in Figure A-8.



Figure A-7

**Figure A-8**

As can be seen in this window, the user is allowed to select what components of the structural model and loading conditions are to be displayed. This can be done for the nodes in the 'Node Options' collective frame on the left-hand side of the window, and for the elements in the 'Element Options' collective frame on the right-hand side of the window. The collective frame in the lower part, titled 'Element Filters', allows the user to select the elements that are required to be excluded from the view to be created. Not all those element types and loading conditions apply to VecTor3, but the ones that are available in VecTor3 will be discussed in detail in the following steps. After selecting the

options of preference for the view to be created, pressing the 'Apply' button creates the desired view and pressing the 'Done' button closes the window.

The next six buttons in the toolbar shown in Figure A-6 are responsible for providing better viewing features for three-dimensional structures. The seventh, eighth, and ninth buttons from the left, titled 'XY', 'YZ', and 'XZ', respectively, allow the user to display a cross section of the structure in the X-Y, Y-Z, or X-Z planes, respectively. A typical view of the window that creates a cross-sectional view of the structural model in the X-Y plane is shown in Figure A-9. The user can specify the elevation in the Z-direction at which the cross-sectional view is required to be displayed. For a cross-sectional view in the Y-Z plane, the elevation in the X-direction should be specified and for a cross-sectional view in the X-Z plane, the elevation in the Y-direction should be specified. Also, the drop list in this window allows the user to view a projection view of the structure from either of the two opposite sides (top and bottom or left and right).

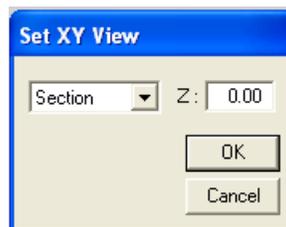


Figure A-9

The next two buttons, shown in Figure A-10, allow the user to navigate through the cross-sectional view created using the 'XY', 'YZ', and 'XZ' buttons, where the button on the left-hand side, featuring the upward arrow, increases the coordinate of the displayed cross-sectional view, and the button on the right-hand side, featuring the downward arrow, decreases the coordinate of the displayed cross-sectional view. Finally, the last button from the left in the toolbar shown in Figure A-6, titled '3D', which is shown in Figure A-11, creates a three-dimensional isometric view of the structural model.

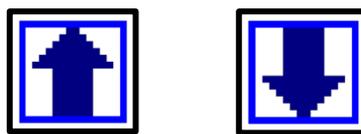


Figure A-10



Figure A-11

A.2.2. Material Definition

This step is required for defining the material properties to be used for the different elements that will be defined afterwards in order to compose the structural model to be analyzed. The user input in this step is saved in the structural 'S3R' input file. Three main material types can be defined in VecTor3, with only the first one (the base material) being mandatory for proceeding with the analysis. Those material types are: the base material, the discrete reinforcement material, and the bond material.

The buttons in the toolbar shown in Figure A-12 are responsible for all user-input data for the 'Material Definition' step.

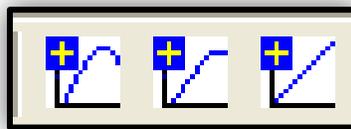


Figure A-12

A.2.2.1. Base Material Properties Definition

This step is required in order to define the base material type of the structural model. The options available in VecTor3 for the base material are concrete or structural steel. Defining at least one base material is mandatory for proceeding with the analysis job.

Pressing the button shown in Figure A-13, which is the third button from the left in the toolbar shown in Figure A-4, the window shown in Figure A-14 opens allowing the user to input the material properties.



Figure A-13

Figure A-14

This window enables the user to select the base material type from a drop list, titled 'Reference Type', from two options: reinforced concrete and structural steel. Figure A-14 shows the input boxes available for the 'Reinforced Concrete' option. As noted in the footnotes in the window, the parameters designated with an asterisk (*) are not mandatory, as VecTor3 imposes default values, should the user opts to leave them vacant or with a value of zero. For concrete material properties, only the 'Cylinder Compressive Strength', f'_c , is mandatory. The following formulae are used to calculate default values for the other properties, depending on the input value of the cylinder compressive strength.

$$f'_t = 0.33\sqrt{f'_c} \quad (1)$$

$$E_{c_i} = 3320\sqrt{f'_c} + 6900 \quad (2)$$

$$\varepsilon'_c = 7.5 \times 10^{-3} f'_c + 1.80 \quad (3)$$

where f'_t is the 'Tensile Strength' (cracking stress); E_{c_i} is the 'Initial Tangent Elasticity Modulus'; and ε'_c is the 'Cylinder Strain at f'_c ' (strain corresponding to the peak compressive stress, f'_c). A typical stress-strain curve for concrete under compression is shown in Figure A-15, showing the main material properties required to compile the curve, where f_c and ε_c are the compressive stress and strain in concrete, respectively.

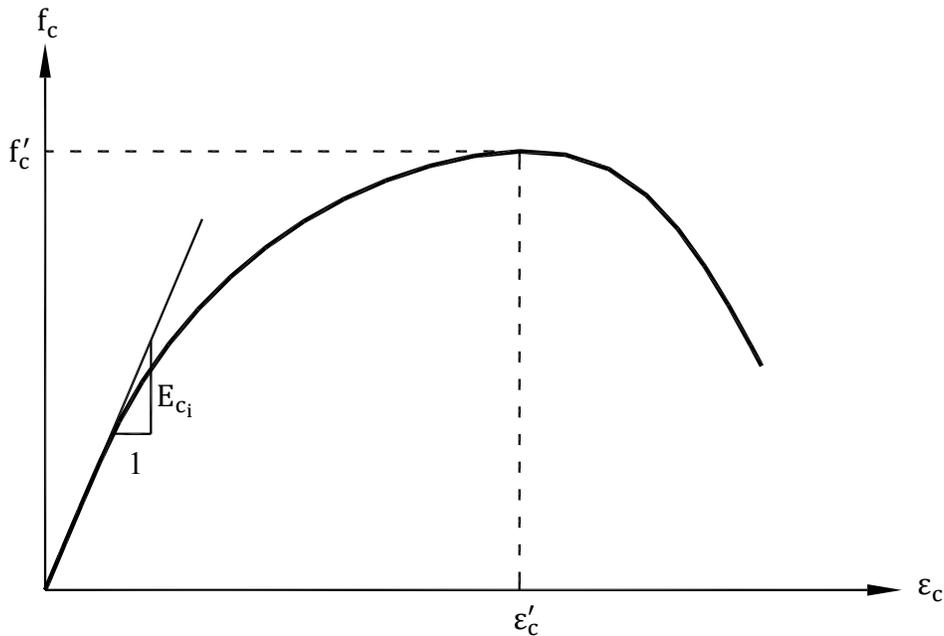


Figure A-15

If left vacant, other properties will be assigned the most commonly used values, independent of the input value of the cylinder compressive strength. 'Poisson's Ratio', μ , and the 'Maximum Aggregate Size', a , are assigned default values of 0.15 and 20 mm, respectively.

The input value for the 'Density' is used to calculate the lumped masses at the nodes comprising the elements modelled with this material for dynamic analysis purposes only. These masses are not used for static dead weight loads, which can be defined using a special load type that will be explained later. Hence, no default value will be assigned to the density of the material. For the average crack spacings in the three directions, S_x , S_y , and S_z , they are calculated, by default, according to the CEB-FIP Model Code (1978), unless their values are explicitly entered by the user in their respective boxes.

It should be noted that the value of the 'Thermal Expansion Coefficient', C_c , entered in this window will not be used in the analysis. It will be overridden by a value that will be calculated according to a specific temperature-dependent model that will be chosen for the thermal expansion coefficient of concrete as will be explained later. Also, the value entered for the 'Thermal Diffusivity', α , will be discarded, since separate temperature-dependent models for the thermal conductivity, specific heat capacity, and density will be used instead. Those models will be defined later.

Should the user choose 'Structural Steel' as the 'Reference Type' in the window shown in Figure A-14, the properties defining the material will change to steel material properties as shown in Figure A-16. The user is then required to enter the various material properties of structural steel, including the 'Yield Strength', f_y , 'Ultimate Strength', f_u , Elastic Modulus (Young's Modulus), E_s , 'Strain-Hardening Strain', ϵ_{sh} , 'Ultimate Strain', ϵ_u , 'Poisson's Ratio', μ , 'Density', ρ , and 'Unsupported Length Ratio', b/t , for buckling consideration. The yield strength is a mandatory input value, but for the other properties, if left vacant or are assigned a value of zero, the following default values and formulae will be used to calculate them: one and half times the value of the yield stress for the ultimate strength, 200,000 MPa for the elastic (Young's) modulus, 5.0 millistrain for the strain-hardening strain, 150.0 millistrain for the ultimate strain, and 0.30 for Poisson's ratio. The density will not be assigned a default value for the same reason that was explained earlier for the 'Reinforced Concrete' material type.

Figure A-17 shows the shape of a typical stress-strain curve for structural steel, showing the main material properties required to compile the curve, where f_s and ϵ_s are the stress and strain in steel, respectively. For structural steel as well, the thermal expansion coefficient entered in this window will be overridden by the value calculated according to the model to be chosen for the temperature-dependent thermal expansion coefficient of steel. Thermal diffusivity of steel entered in this window will also be discarded for the same reason that was explained for the 'Reinforced Concrete' material type.

Define Material Properties

Material Types

Type:

Add
Update
Delete

Material Properties

Reference Type: Structural Steel

Yield Strength, F_y : MPa

Ultimate Strength, F_u : * MPa

Elastic Modulus, E_s : * MPa

Strain-Hardening Strain, ϵ_{sh} : * me

Ultimate Strain, ϵ_u : *

Thermal Expansion Coefficient, C_s : * /°C

Poisson's Ratio, μ_u : *

Density: * kg/m³

Thermal Diffusivity, K_s : mm²/s

Average Crack Spacing...

Unsupported Length Ratio, b/t :

perpendicular to y-reinforcement, S_y : * mm

perpendicular to z-reinforcement, S_z : * mm

Color

Smearred Reinforcement Properties

Reference Type: Ductile Steel Reinforcement

Reinforcement Orientation:

k l m

Reinforcement Ratio, ρ : %

Reinforcement Diameter, D_b : mm

Yield Strength, F_y : MPa

Ultimate Strength, F_u : MPa

Elastic Modulus, E_s : MPa

Strain Hardening Strain, ϵ_{sh} : me

Ultimate Strain, ϵ_u : me

Thermal Expansion Coefficient, C_s : * /°C

Prestrain, D_{ep} : me

Unsupported Length Ratio, b/t :

Reinforced concrete material types to be used for solid elements only. * Enter '0' for VT3 default value.

OK Cancel

Figure A-16

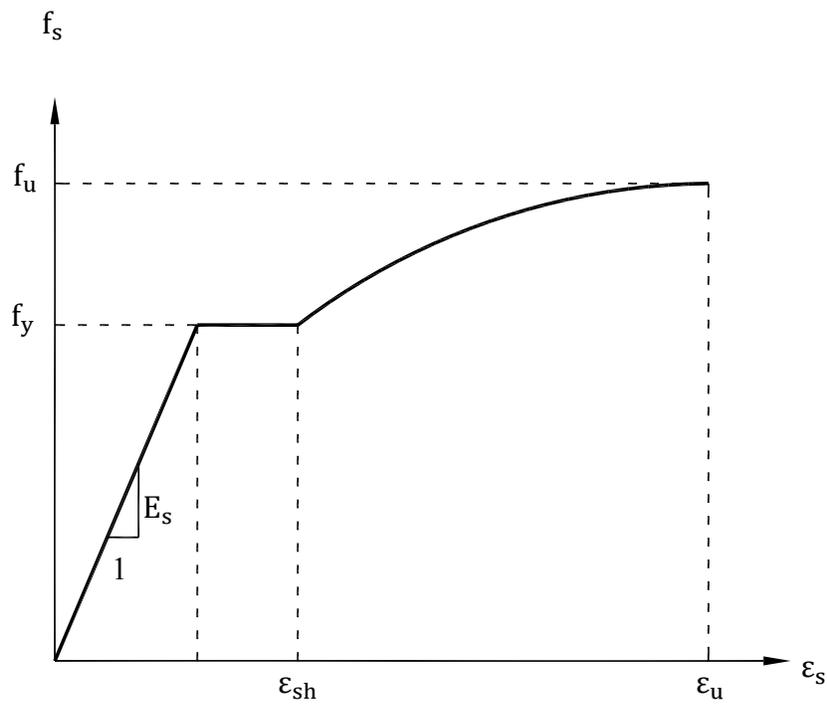


Figure A-17

After the material properties of 'Reinforced Concrete' or 'Structural Steel' are entered in their respective boxes (or left vacant or with zero values for the option to use the default values), the user is required to press the 'Add' button in the upper part on the left-hand side of the window shown in Figure A-14. At this point, the new material type, named 'Material 1', appears in the box titled 'Type' on the left-hand side of the window, as shown in Figure A-18. Should the user need to change the value of any of the properties previously entered, the new value should be entered in the respective box; then, the button titled 'Update' should be pressed. Also, the 'Delete' button allows the user to delete the undesired material types, by selecting them from the box titled 'Type' on the left-hand side of the window and pressing the button. The user is also given the option to choose the colour in which the defined material will be displayed using the button titled 'Colour'.

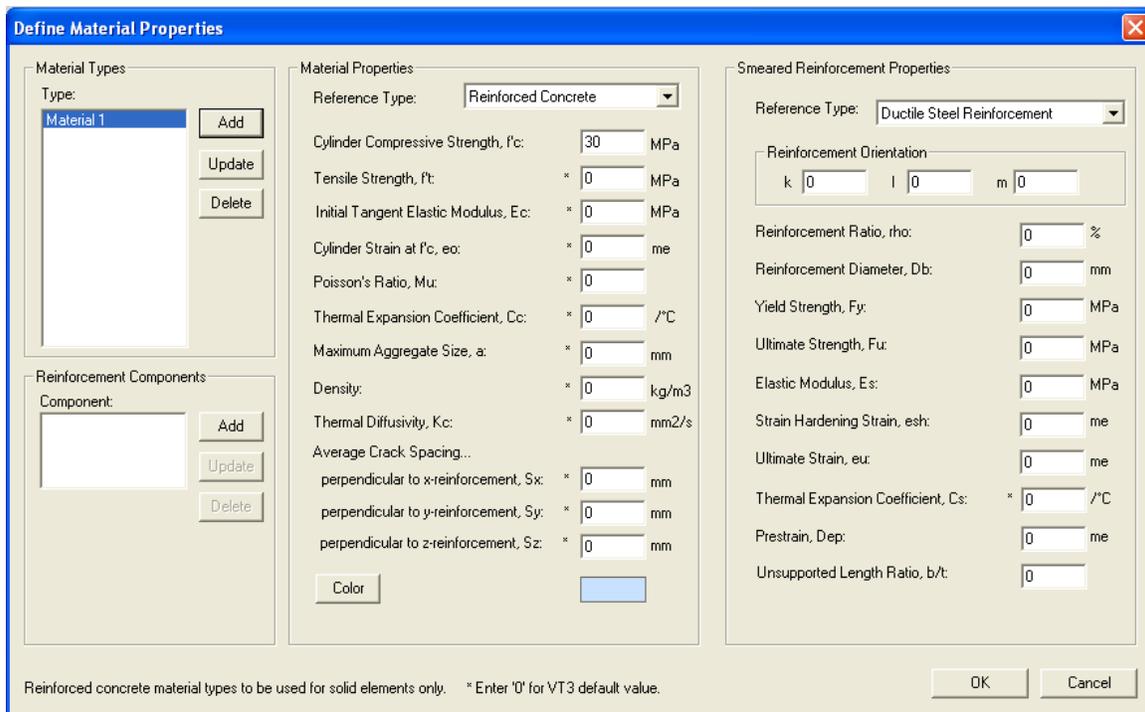


Figure A-18

With the material type added, the previously greyed-out part of the window on the right-hand side, titled 'Smearred Reinforcement Properties', becomes available for user input. In case the 'Reference Type' chosen was 'Reinforced Concrete', smeared steel reinforcement can be added through this part of the window.

The drop list at the top of this part offers the following six types of smeared reinforcement available for use in VecTor3: 'Ductile Steel Reinforcement' (smooth or deformed), 'Prestressing Steel', 'Tension Only Reinforcement', 'Compression Only Reinforcement', 'Steel Fibre - Hooked', and 'Steel Fibre - Straight'. The last two types are used for defining steel fibre-reinforced concrete (SFRC).

For typical reinforced concrete structures modelled using smeared reinforcement, the user should select 'Ductile Steel Reinforcement' from the drop list. Then, the user is required to specify the orientation of the steel reinforcement by entering the values of the cosines of the angles that the reinforcement makes with the three global Cartesian coordinates, X, Y, and Z, namely k, l, and m, respectively. The summation of the squares of k, l, and m should obviously be equal to 1.

The next step is to enter the 'Reinforcement Ratio', ρ , which is the percentage of the cross-sectional area of the reinforcement to the area of the entire section. Alternatively, for the finite element analysis purposes, the reinforcement ratio can be defined as the percentage of the cross-sectional area of the reinforcement to the area of the section of the concrete finite element that is perpendicular to the reinforcement.

The user is then required to enter the various material properties of steel reinforcement, including the 'Reinforcement Diameter', d_b , 'Yield Stress', f_y , 'Ultimate Stress', f_u , 'Elastic Modulus' (Young's modulus), E_s , 'Strain-Hardening Strain', ϵ_{sh} , 'Ultimate Strain', ϵ_u , 'Prestrain' (for prestressed reinforcement), and 'Unsupported Length Ratio' for buckling consideration. Figure A-17 presents the shape of a typical stress-strain curve for steel reinforcement, showing the main material properties of steel required to compile the curve, where f_s and ϵ_s are the compressive stress and strain in steel, respectively. In this part as well, the thermal expansion coefficient entered is overridden by the value calculated according to the model to be chosen for the temperature-dependent thermal expansion coefficient of steel reinforcement.

After the smeared steel reinforcement material properties are entered in their respective boxes, the user is required to press the 'Add' button in the lower part on the left-hand side of the window shown in Figure A-18. At this point, the new reinforcement component type, titled 'Reinforcement 1', appears in the box titled 'Component' on the left-hand

side of the window, as shown in Figure A-19. In this part as well, the 'Update' and 'Delete' buttons can be used to update the material properties of the smeared steel reinforcement or delete the undesired ones, respectively.

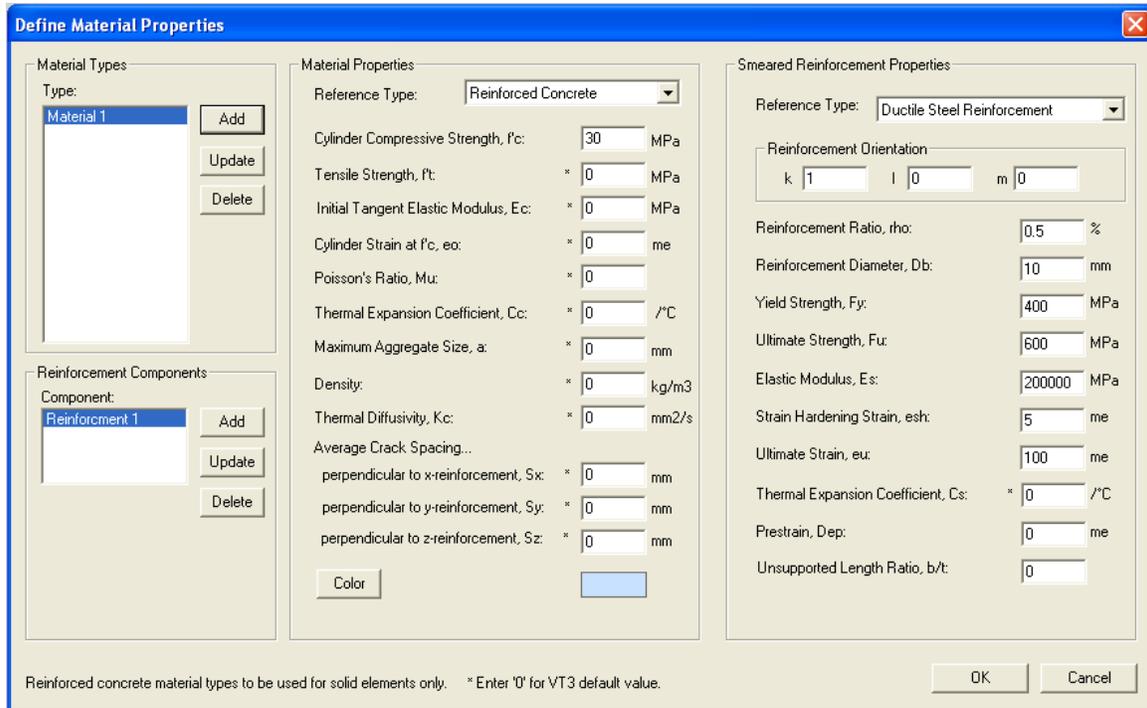
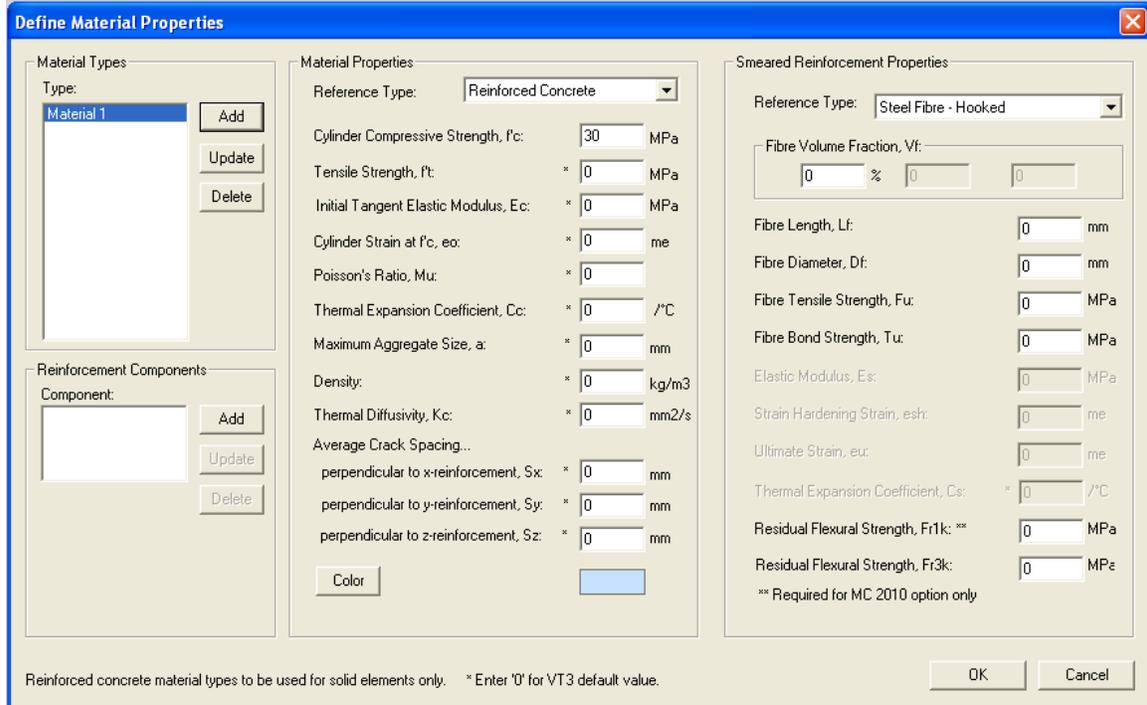


Figure A-19

For defining steel fibre-reinforced concrete, the 'Reference Type' in the 'Smeared Reinforcement Properties' drop list should be chosen as 'Steel Fibre - Straight' or 'Steel Fibre - Hooked' for straight or end-hooked types of fibres, respectively. When either these options is chosen, the window changes to the form shown in Figure A-20, where the properties of SFRC can be entered by the user. The properties to be entered are the 'Fibre Volume Fraction', V_F , 'Fibre Length', l_f , 'Fibre Diameter', d_f , 'Fibre Tensile Strength', f_u , and 'Fibre Bond Strength', t_u . Also for this input, the user is required to press the 'Add' button in the lower part on the left-hand side of the window shown in Figure A-20 and can use the 'Update' and 'Delete' buttons as previously explained.



Define Material Properties

Material Types

Type: Material 1 [Add] [Update] [Delete]

Reinforcement Components

Component: [Add] [Update] [Delete]

Material Properties

Reference Type: Reinforced Concrete

Cylinder Compressive Strength, f_c : 30 MPa

Tensile Strength, f_t : * 0 MPa

Initial Tangent Elastic Modulus, E_c : * 0 MPa

Cylinder Strain at f_c , ϵ_c : * 0 me

Poisson's Ratio, μ_c : * 0

Thermal Expansion Coefficient, C_c : * 0 /°C

Maximum Aggregate Size, a : * 0 mm

Density: * 0 kg/m³

Thermal Diffusivity, K_c : * 0 mm²/s

Average Crack Spacing...

perpendicular to x-reinforcement, S_x : * 0 mm

perpendicular to y-reinforcement, S_y : * 0 mm

perpendicular to z-reinforcement, S_z : * 0 mm

[Color] [Color]

Smearred Reinforcement Properties

Reference Type: Steel Fibre - Hooked

Fibre Volume Fraction, V_f : 0 % 0 0

Fibre Length, L_f : 0 mm

Fibre Diameter, D_f : 0 mm

Fibre Tensile Strength, F_u : 0 MPa

Fibre Bond Strength, T_u : 0 MPa

Elastic Modulus, E_s : 0 MPa

Strain Hardening Strain, ϵ_{sh} : 0 me

Ultimate Strain, ϵ_u : 0 me

Thermal Expansion Coefficient, C_s : * 0 /°C

Residual Flexural Strength, F_{r1k} : ** 0 MPa

Residual Flexural Strength, F_{r3k} : 0 MPa

** Required for MC 2010 option only

[OK] [Cancel]

Reinforced concrete material types to be used for solid elements only. * Enter '0' for VT3 default value.

Figure A-20

It should be noted that VecTor3 allows for defining up to fifty different material types. This includes the base material types in addition to the discrete reinforcement types and the bond types which will be explained in the next two subsections. For the base material types, VecTor3 allows assigning up to four different types of smeared reinforcement (including fibres) per material type.

A.2.2.2. Discrete Reinforcement Material Properties Definition

This step is responsible for defining the various types of discrete (not smeared) reinforcement in the structural model. The types of discrete reinforcement available in VecTor3 are 'Ductile Steel Reinforcement' (smooth or deformed), 'Prestressing Steel', 'Tension Only Reinforcement', and 'Compression Only Reinforcement'. This step is optional, depending on the presence of reinforcement in the structure and the desire of the user to model this reinforcement as discrete uniaxial truss bar elements rather than smearing it evenly within the concrete finite element that contains it.

To start defining the discrete reinforcement material properties, the fourth button from the left in the toolbar shown in Figure A-4 should be pressed. The shape of the button is shown in Figure A-21. This button opens the window shown in Figure A-22.

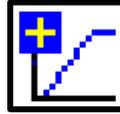
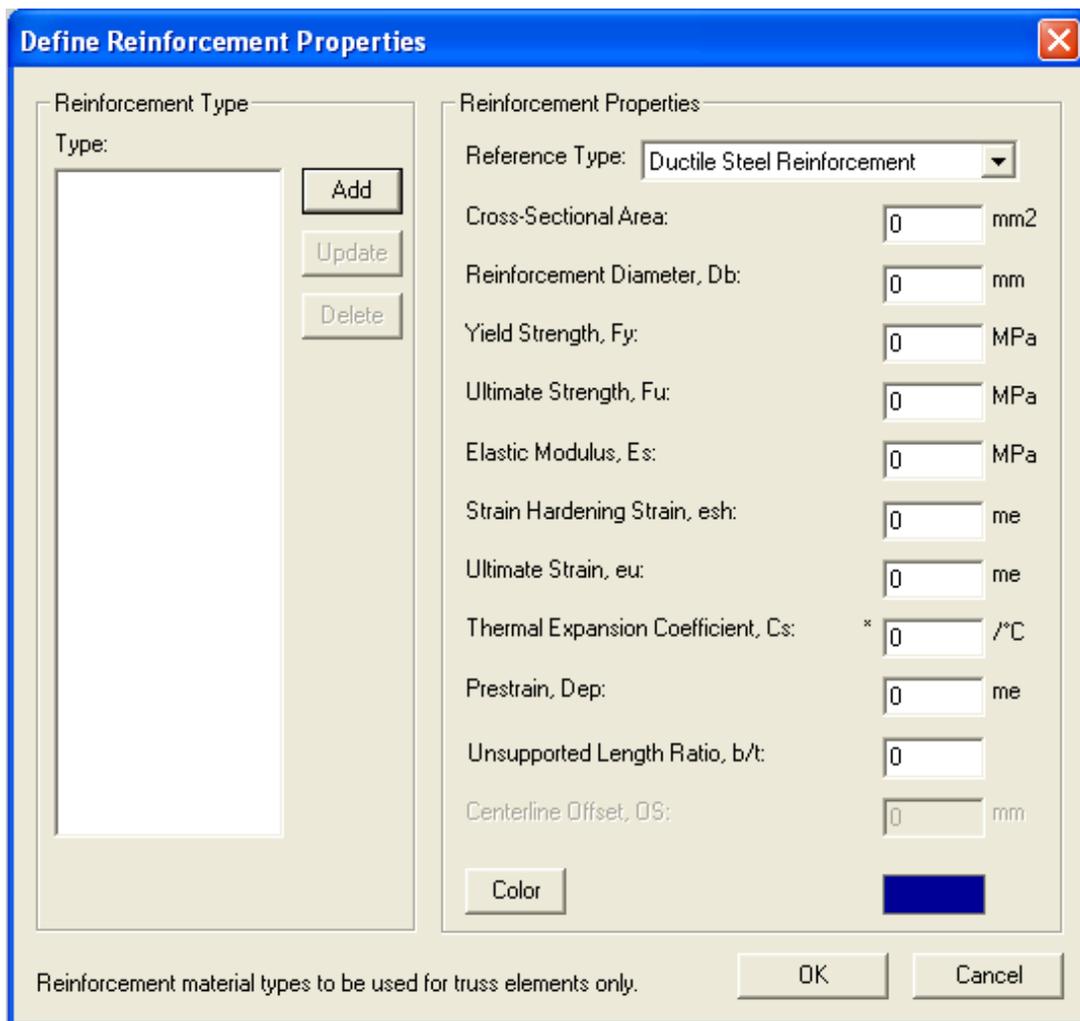


Figure A-21



Define Reinforcement Properties

Reinforcement Type

Type:

Add

Update

Delete

Reinforcement Properties

Reference Type: Ductile Steel Reinforcement

Cross-Sectional Area: 0 mm²

Reinforcement Diameter, D_b : 0 mm

Yield Strength, F_y : 0 MPa

Ultimate Strength, F_u : 0 MPa

Elastic Modulus, E_s : 0 MPa

Strain Hardening Strain, es_h : 0 me

Ultimate Strain, eu : 0 me

Thermal Expansion Coefficient, C_s : * 0 /°C

Prestrain, Dep : 0 me

Unsupported Length Ratio, b/t : 0

Centerline Offset, OS : 0 mm

Color

Reinforcement material types to be used for truss elements only.

OK Cancel

Figure A-22

In the drop list on the right-hand side, the user can choose the type of reinforcement from the four aforementioned types. The properties of steel reinforcement entered in this window also follow the curve shown in Figure A-17 as explained earlier for smeared reinforcement. The single additional property that is unique here is the cross-sectional

area of the reinforcing bar, which substitutes for the reinforcement ratio in the case of the smeared reinforcement.

When the properties are entered, the user is required to press the 'Add' button on the left-hand side of the window shown in Figure A-22, which will add the reinforcement material type to the list box titled 'Type' on the left-hand side, as shown in Figure A-23. The user can use the 'Update', 'Delete', and 'Colour' buttons as required, as explained earlier.

Define Reinforcement Properties

Reinforcement Type

Type:

Reinforcement 1

Add

Update

Delete

Reinforcement Properties

Reference Type: Ductile Steel Reinforcement

Cross-Sectional Area: 500 mm²

Reinforcement Diameter, Db: 25.0 mm

Yield Strength, F_y: 460 MPa

Ultimate Strength, F_u: 640 MPa

Elastic Modulus, E_s: 200000 MPa

Strain Hardening Strain, e_{sh}: 5 me

Ultimate Strain, e_u: 160 me

Thermal Expansion Coefficient, C_s: * 0 /°C

Prestrain, Dep: 0 me

Unsupported Length Ratio, b/t: 0

Centerline Offset, OS: 0 mm

Color

Reinforcement material types to be used for truss elements only.

OK Cancel

Figure A-23

As previously explained, the total number of material types allowed in VecTor3, including base material types, discrete reinforcement types, and bond types, is 50.

A.2.2.3. Bond Properties Definition

This step is responsible for defining the properties of the bond between concrete and discrete reinforcing bars. This allows for the modelling of imperfect bond conditions or totally unbonded conditions for post-tensioned reinforcement cases. This step is accessible through the fifth button from the left in the toolbar shown in Figure A-4. The shape of the button is shown in Figure A-24. This button opens the window shown in Figure A-25.

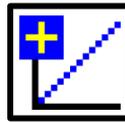
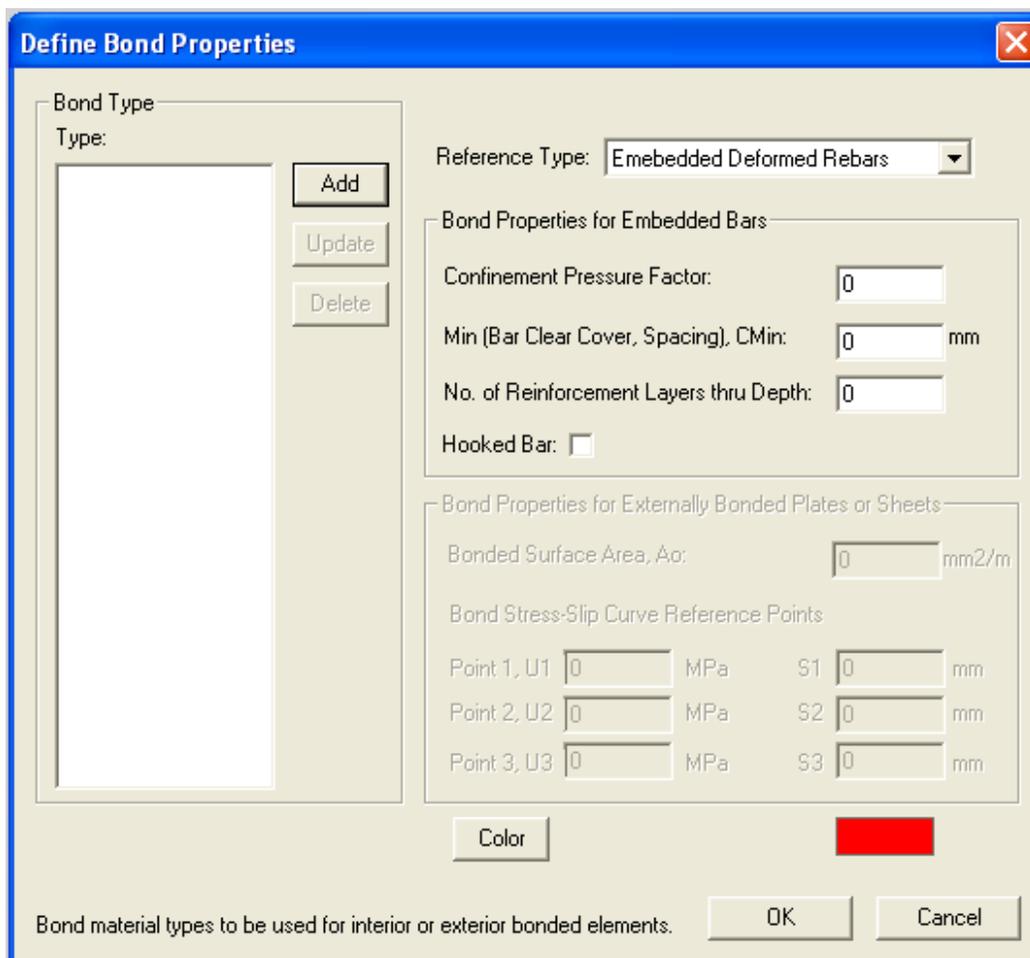


Figure A-24



Define Bond Properties

Bond Type
Type:

Add
Update
Delete

Reference Type: Emebedded Deformed Rebars

Bond Properties for Embedded Bars

Confinement Pressure Factor: 0

Min (Bar Clear Cover, Spacing), CMin: 0 mm

No. of Reinforcement Layers thru Depth: 0

Hooked Bar:

Bond Properties for Externally Bonded Plates or Sheets

Bonded Surface Area, A_o: 0 mm²/m

Bond Stress-Slip Curve Reference Points

Point 1, U1 0 MPa S1 0 mm

Point 2, U2 0 MPa S2 0 mm

Point 3, U3 0 MPa S3 0 mm

Color

Bond material types to be used for interior or exterior bonded elements.

OK Cancel

Figure A-25

The first user input required for modelling the bond type is the type of reinforcing bar from the drop list titled 'Reference Type'. The types available in VecTor3 are the 'Embedded Deformed Rebars', 'Embedded Smooth Rebars', 'Unbonded Bars or Tendons', and 'Unbonded Tendons w/ Friction Losses'. Three other options are available to provide more flexibility for the user. The first two are the 'Interface or Bonded Plates/Fabrics' and the 'Embedded Bars - Custom Input' options, where the user can specify custom bond stress-slip curve using three key points to define the curve. The last option is the 'No Slip' option, where the user can specify perfect bond (no slippage) for specific bond types, which will override the global model that will be selected later for the structural analysis in the 'Structural Analysis Models Definition' step in Section A.2.4.2.

The properties required by the user in this window, besides the type of reinforcing bar, are the 'Confinement Pressure Factor', which defines the level of confinement of concrete in the section, 'Min (Bar Clear Cover, Spacing)' which defines the minimum clear cover or spacing of the reinforcing bar, and the 'Number of Reinforcement Layers Through the Depth' of the concrete section. It should be noted that VecTor3 has the ability to calculate these properties automatically, should the user assigns them a value of zero. Finally, the user is required to tick the 'Hooked Bar' checkbox, if the bars used are hooked-ended.

When the properties are entered, the user is required to press the 'Add' button on the left-hand side of the window shown in Figure A-25, which will add the bond type to the list titled 'Type' on the left-hand side, as shown in Figure A-26. The user can use the 'Update', 'Delete', and 'Colour' buttons as required, as explained earlier.

The greyed-out collective frame at the bottom of the window, titled 'Bond Properties for Externally Bonded Plates or Sheets', becomes available for user input when one of the two options 'Interface or Bonded Plates/Fabrics' and the 'Embedded Bars - Custom Input' is chosen. The user is required to enter the 'Bonded Surface Area' in mm^2/m and three key points to compile the bond stress-slip curve, where 'U1', 'U2', and 'U3' are the bond stresses in MPa and 'S1', 'S2', and 'S3' are the corresponding slips in mm, respectively.

Figure A-26

At this point, all the material types have been entered. For any of these material types (base, discrete reinforcement, and bond), the user can repeat the same steps to define more types of material. For each new material type, the user should enter the material properties in their respective boxes and press the 'Add' button on the left-hand side, which will make the new material type appear in the box titled 'Type' on the left-hand side as well. As previously explained, the user can also change the properties of any material, by selecting it from the list showing in the box titled 'Type' on the left-hand side, making the change, and then pressing the 'Update' button on the left-hand side. Also, the user can delete a previously defined material by selecting it from the list showing in the box titled 'Type' on the left-hand side then pressing the 'Delete' button also on the left-hand side.

A.2.3. Structure Definition

This step is required to define the elements composing the structural model to be analyzed. The user input in this step is saved in the structural 'S3R' input file. This step starts with the definition of the coordinates of the nodes that will be used to define the different element comprising the structural model. Next, those elements can be assigned specific material types from the ones defined in the 'Material Definition' step in Section A.2.2 of the *VecTor3: User's Manual*. The element types available for use in VecTor3 are the eight-noded regular hexahedral element (with right-angled corners), the eight-noded isoparametric hexahedral element, the six-noded regular wedge element (with right-angled corners for the rectangular sides), the two-noded uniaxial truss bar element for modelling discrete reinforcing bars, and the dimensionless two-noded link elements for modelling the bond between concrete and discrete reinforcing bars.

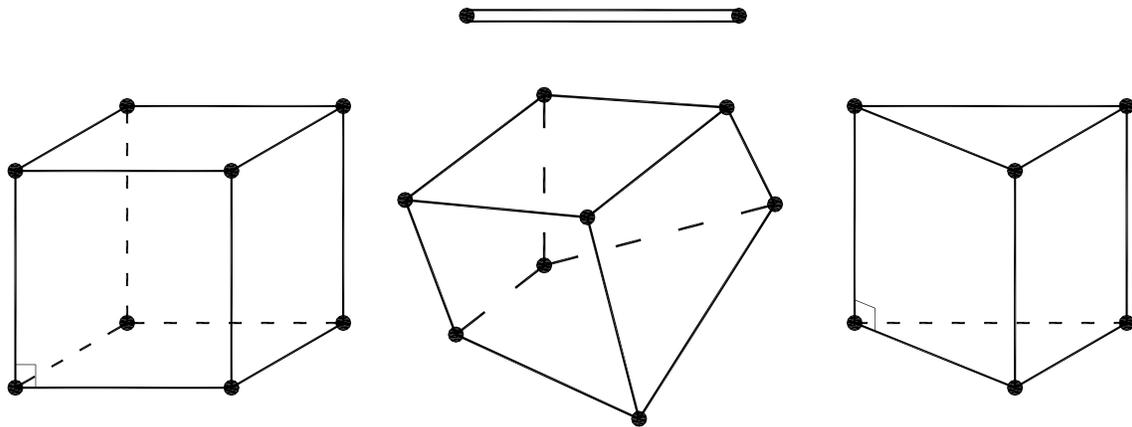


Figure A-27

The buttons in the toolbar shown in Figure A-28 are responsible for all user-input data for the 'Structure Definition' step.



Figure A-28

At any point during this step, the nineteenth button from the left (the last button) in the toolbar shown in Figure A-28, whose shape is shown in Figure A-29, can be used to delete the entire structural model including the nodes, material types, elements, and loads.



Figure A-29

A.2.3.1. Node Definition

A.2.3.1.1. Defining Node Coordinates

For defining the coordinates of the nodes the first button from the left in the toolbar shown in Figure A-28 is used. The shape of the button is shown in Figure A-30. Pressing the button opens the window shown in Figure A-31.

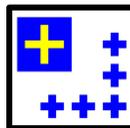


Figure A-30

node	x	y	z	#nodes	d node	d x	d y	d z	#nodes	d node	d x	d y	d z	#nodes	d node	d x	d y	d z	Total
1	0	0	0	1	1	0	0	0	1	1	0	0	0	1	1	0	0	0	0

Figure A-31

In this window, the user can define the nodes by specifying their coordinates in the 'x', 'y', and 'z' boxes, individually, one at a time. Another way to perform this task is to enter the coordinates of the first node in the 'x', 'y', and 'z' boxes, and extrapolate them to define more nodes by specifying the number of nodes to be defined, '# nodes', and the increments of the subsequent nodes in the ID number 'd(node)', the x-coordinate 'd(x)', the y-coordinate 'd(y)', and the z-coordinate 'd(z)'. This extrapolation can be done in

three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-31. After each entry, whether it is for an individual node or an extrapolated group of nodes, the 'Add' button on the right-hand side of the window needs to be pressed. This will cause the entry to show in the larger box at the bottom of the window as shown in Figure A-32.

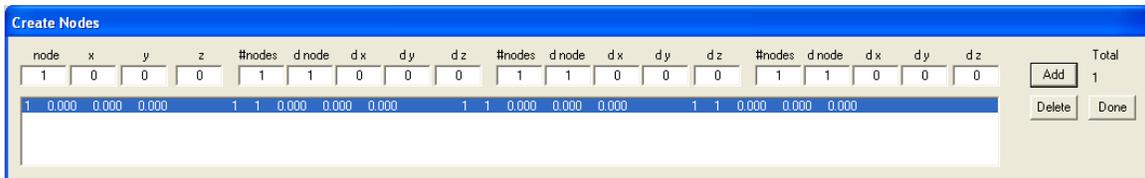


Figure A-32

Unneeded entries can be deleted by selecting them from the list in the larger box at the bottom of the window then pressing the 'Delete' button on the right-hand side of the window. When all the entries of all the required nodes is entered, pressing the 'Done' button on the right-hand side of the window closes the window.

A feature that is available for facilitating the nodes' definition step is the node copying feature. This feature can be accessed through the second button from the left in the toolbar shown in Figure A-28, which is shown in Figure A-33. It enables the user to duplicate certain previously-defined nodes at a certain offset from the existing nodes. The input window shown in Figure A-34 appears when the button is pressed. The input required by the user is the ID number of the node to be duplicated, the offset distance in the global X, Y, and Z directions ('DX', 'DY', and 'DZ'), and the number of duplicate layers to be generated, '# layers'. Finally, for defining a group of nodes to be duplicated, the user is required to input the number of nodes, '# nodes' and the increment in the ID number of the subsequent nodes 'd(node)' for the previously explained extrapolation scheme of nodes selection.

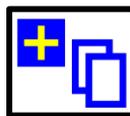


Figure A-33

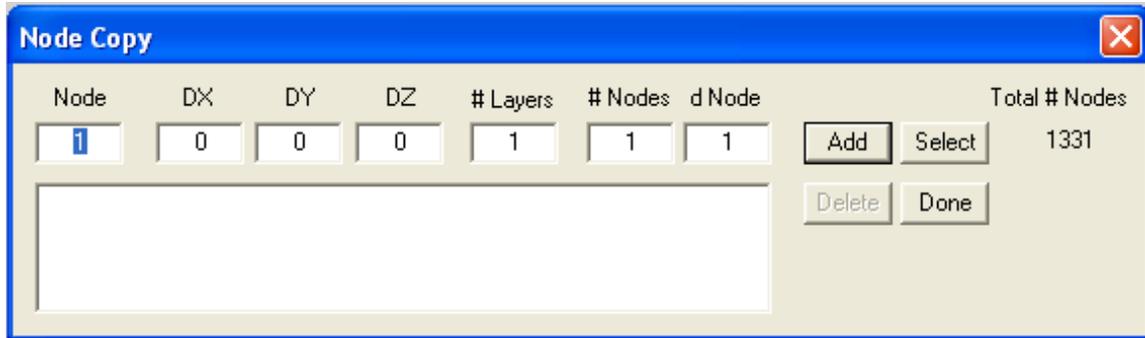


Figure A-34

The button titled 'Select' enables the user to graphically select one node at a time from the nodes showing in the workspace view using the left mouse button. After each entry (of either one or multiple nodes), the 'Add' button on the right-hand side of the window needs to be pressed. This will cause the entry to show in the larger box at the bottom of the window. Unneeded entries can be deleted by selecting them from the list in the larger box at the bottom of the window then pressing the 'Delete' button on the right-hand side of the window. When all the input entries are entered, pressing the 'Done' button on the right-hand side of the window closes the window.

A.2.3.1.2. Applying Node Restraints

This step enables the user to apply certain restraints on specific nodes in the structural model in order to model the support conditions of the structure to be analyzed. Pressing the fourth button from the left in the toolbar shown in Figure A-28, shown in Figure A-35, opens the window shown in Figure A-36. The user can specify the ID number of a node and the directions (or the degrees of freedom (DOFs)) in which the node needs to be restrained in the global X, Y, and Z directions by ticking the boxes titled 'DX', 'DY', and 'DZ', respectively. Ticked directions mean that the node will be restrained in that direction, while unticked ones mean that the node will be free to displace in that direction.



Figure A-35

node	Restrain d.o.f.						# nodes	d node	# nodes	d node	# nodes	d node	Total
1	DX	DY	DZ	R1	R2		1	1	1	1	1	1	0

Figure A-36

As previously explained, also in this window, the user can define the nodes restraints individually, one at a time, or specify the ID number of a certain node with certain restraints and extrapolate the same restraints to other nodes by specifying the number of nodes having the same restraints, ‘# nodes’, and the increment in the ID number of the subsequent nodes, ‘d(node)’. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-36.

A.2.3.1.3. Applying Node Constraints (Linking Nodes)

This step enables the user to apply certain constraints on specific groups of nodes. The nodes can be linked together with specific linear displacement relations throughout the analysis. This allows one or two groups of nodes to have the same displacements in specific directions, or one group to have a displacement of a pre-defined percentage of the displacement of another group.

Pressing the fourth button from the left in the toolbar shown in Figure A-28, shown in Figure A-37, opens the window shown in Figure A-38. The user can specify one group of nodes to be linked together to displace similarly (using the top row of entry boxes) or two groups of nodes to be linked together with a specific relation specified in the ‘Factor’ entry (using the two rows of entry boxes). If only one group is entered, the nodes included in that group will have the same displacement in the specified direction (DOF)

throughout the analysis. If two groups are entered, then the nodes included in each group will have the same displacement in the specified direction (DOF) throughout the analysis, and the displacements of the nodes in the second group will be linked to the displacements of the first group through the specified factor.

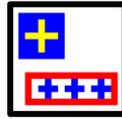


Figure A-37

Group	D.O.F.1	Node1	# nodes	d node	# nodes	d node	# nodes	d node	Total # Linked Nodes
1	X <input checked="" type="radio"/> Y <input type="radio"/> Z <input type="radio"/>	1	1	1	1	1	1	1	0
Factor	D.O.F.2	Node2	# nodes	d node	# nodes	d node	# nodes	d node	
1	X <input checked="" type="radio"/> Y <input type="radio"/> Z <input type="radio"/>	1	1	1	1	1	1	1	

Figure A-38

For a specific entry, the user is required to select the degree of freedom to be linked for the first group (in the global X, Y, or Z-direction) in the part titled 'DOF 1', then the ID number of the first node in the first group is entered in the box titled 'Node 1'. For including more nodes in the group, the user can use the previously explained extrapolation scheme through three sets of boxes for three levels of extrapolation.

If the displacements of the nodes included in the first group are to be linked to the displacements of other nodes through a specific relation, the user is required to enter the input in the second row of boxes in the window shown in Figure A-38. The relation between the two groups is specified through a certain factor entered in the box titled 'Factor'. Then, the degree of freedom of the displacements of the nodes included in the second group which is required to be linked to the degree of freedom of the displacements of the nodes included in the first group should be entered in the part titled 'DOF 2'. Next, the ID number of the first node in the second group is required to be

entered in the box titled 'Node 2'; and more nodes are allowed to be included in that group as well through the three-levelled extrapolation scheme previously explained.

This specification will result in the following relation between the displacements of the nodes included in the first group in the direction 'DOF 1', denoted, D1, and the displacements of the nodes included in the second group in the direction 'DOF 2', denoted, D2:

$$D1 = \text{'Factor'} \times D2$$

A.2.3.2. Element Definition

For defining the elements shown in Figure A-27, the tenth, eleventh, twelfth, and seventeenth buttons from the left in the toolbar shown in Figure A-28 are used for the eight-noded regular hexahedral element, the eight-noded isoparametric hexahedral element, the six-noded regular wedge element, and the two-noded uniaxial truss bar element, respectively. The first three element types can model the base material (reinforced/unreinforced concrete, steel fibre-reinforced concrete, or structural steel), while the last one can model discrete reinforcing bars. The shapes of the buttons are shown in Figure A-39 in order from left to right. Pressing these buttons opens the windows shown in Figure A-40 to Figure A-43, respectively.

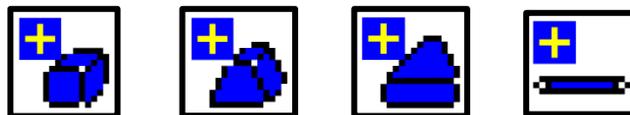


Figure A-39

Create Regular Hexahedral Elements

elmt	node 1	2	3	4	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
		5	6	7										
		1	1	1										

Figure A-40

Create Isoparametric Hexahedral Elements

elmt	node 1	2	3	4	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	1	1	1	1	1	1	1	1	1	1	1	1	1	0
		5	6	7										
		1	1	1										

Figure A-41

Create Wedge Elements

elmt	node 1	2	3	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	1	1	1	1	1	1	1	1	1	1	1	1	0
		4	5	6									
		1	1	1									

Figure A-42

Create Truss Elements

elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	1	1	1	1	1	1	1	1	1	1	1	0

Figure A-43

For the eight-noded regular hexahedral element and the eight-noded isoparametric hexahedral element, the user is required to enter a set of four nodes composing one of the faces of the element for the first four nodes and then enter another set of four nodes composing the opposite face and in the same order. This means that Nodes #1, #2, #3, and #4 form a face of the six-faced element and Nodes #5, #6, #7, and #8 form the opposite face. Node #1 should be opposite to Node #5, Node #2 should be opposite to Node #6, Node #3 should be opposite to Node #7, and Node #4 should be opposite to Node #8.

Similarly, for the six-noded regular wedge element, the user is required to enter a set of three nodes composing one of the faces of the element for the first three nodes and then enter another set of three nodes composing the opposite face and in the same order. This means that Nodes #1, #2, and #3 form a face of the two triangular faces of the element and Nodes #4, #5, and #6 form the opposite triangular face. Node #1 should be opposite to Node #4, Node #2 should be opposite to Node #5, and Node #3 should be opposite to Node #6.

In those windows shown in Figure A-40 to Figure A-43, the user can define the elements by specifying the ID numbers of their composing nodes individually, one at a time. Another way to perform this task is to enter the composing nodes of the first element and extrapolate them to define other elements by specifying the number of elements to be defined, '# elmts', the increment in the ID number of subsequent elements, 'd(elmt)', and the increment in the ID number of the nodes comprising subsequent elements, 'd(node)'. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-40 to Figure A-43. After each entry, whether it is for an individual element or for an extrapolated group of elements, the 'Add' button on the right-hand side of the window needs to be pressed. This will cause the entry to show in the larger box at the bottom of the window as shown in Figure A-44 as a typical example for the regular hexahedral element input.

elmt	node 1	2	3	4	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	5	6	8	7	1	1	1	1	1	1	1	1	1	1
	5	6	7	8										
	1	2	4	3										

1 5 6 8 7 1 2 4 3 1 1 1 1 1 1 1 1

Figure A-44

For defining the dimensionless two-noded link elements required for modelling the bond between concrete and reinforcing bars, the eighth button from the left in the toolbar shown in Figure A-28, which is shown in Figure A-45, needs to be pressed. This opens the window shown in Figure A-46.

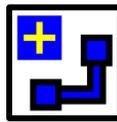


Figure A-45

elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total
1	1	1	1	1	1	1	1	1	1	1	1	1000

Figure A-46

The input entries for link elements have the same format as the other types of elements. However, these elements are dimensionless as previously indicated, meaning that the two nodes comprising the elements must coincide on each other (have the exact same coordinates).

Since link elements model the bond between concrete and reinforcing bars, one of the comprising nodes needs to be a node on a concrete element (regular hexahedral element, isoparametric hexahedral element, or wedge element) and the other node needs to be a node on a truss bar element. For those elements, the user is required to define two

coincident nodes at the location of the element in the 'Nodes Coordinates Definition' step.

For all types of elements, unneeded entries can be deleted by selecting them from the list in the larger box at the bottom of the window then pressing the 'Delete' button on the right-hand side of the window. When the input of all the required elements is entered, pressing the 'Done' button on the right-hand side closes the window.

A.2.3.3. Material Type Assignment

In this step, the elements that were defined in the 'Elements Definition' step (Section A.2.3.2 of the *VecTor3: User's Manual*) are assigned material types from the types that were defined in the 'Material Definition' step (Section A.2.2 of the *VecTor3: User's Manual*). This can be done through the eighteenth button from the left in the toolbar shown in Figure A-28. The shape of the button is shown in Figure A-47.



Figure A-47

Pressing this button opens the window shown in Figure A-48, where the ID number of the element can be entered in the box titled 'elmt' and a material type can be assigned to it from the drop list titled 'Material'. The drop list includes all the material types that were defined in the 'Material Definition' step (Section A.2.2 of the *VecTor3: User's Manual*), including the base material types, the discrete reinforcement material types, and the bond types. The user can choose to deactivate the element in the analysis job by unticking the box titled 'Act'. Deactivated elements are considered to be completely non-existent in the current analysis, but can be activated in subsequent analyses.

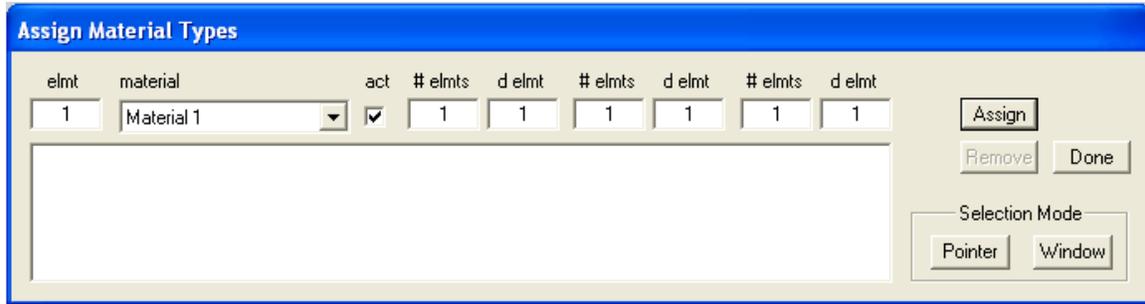


Figure A-48

As previously explained in the ‘Elements Definition’ step (Section A.2.3.2 of the *VecTor3: User's Manual*), the user can assign material types to each element individually, one at a time, or, extrapolate the material assignment to more elements by specifying the number of elements required to be assigned that material type, ‘# elmts’, and the increment in the ID number of the subsequent elements, ‘d(elmt)’. As previously mentioned, this extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-48.

There are also graphical options available for the user to assign material types to the elements. These options can be accessed using the two buttons ‘Pointer’ and ‘Window’ which appear under the collective frame titled ‘Selection Mode’ on the right-hand side of the window shown in Figure A-48. The button ‘Pointer’ allows the user to select one element at a time from the elements displayed in the workspace view, and the button ‘Window’ allows the user to select multiple elements in one step by holding down the left mouse button and dragging a window that contains the desired elements.

Finally, a summary of the ‘Structure Definition’ step can be viewed through the second button from the left in the toolbar shown in Figure A-4, whose shape is shown in Figure A-49. Pressing this button opens the window shown in Figure A-50, where the number of material types, the number of elements of each type, the number of nodes, and the number of restrained nodes can be checked.



Figure A-49

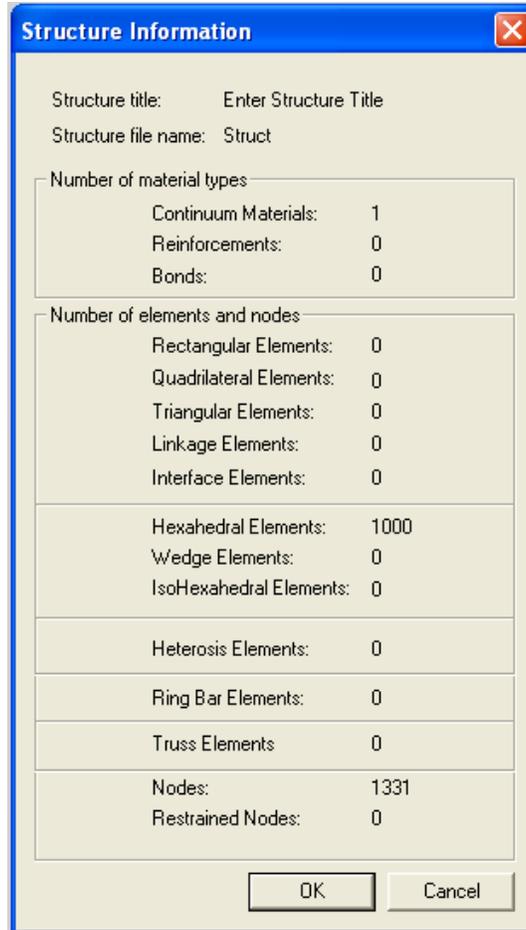


Figure A-50

A.2.4. Job Definition

This step is responsible for defining the analysis job scheme. This can be done before or after the 'Structure Definition' step. Defining the job can be done through the first button from the left in the toolbar shown in Figure A-4, whose shape is shown in Figure A-51.



Figure A-51

Pressing this button opens the window shown in Figure A-52, which contains four main tabs, namely, 'Job Control', 'Models', 'Auxiliary', and 'Auxiliary Thermal'. Each of these tabs is responsible for a certain task in the analysis job definition process.

The 'Define Job' dialog box is shown with the following fields and values:

- Job Data:** Job file name: VecTor; Job title: Enter Job Title; Date: Enter Date.
- Structure Data:** Structure file name: Struct; Structure title: Enter Structure Title; Structure type: Solid (3-D).
- Loading Data:** Load series ID: ID; Starting load stage no.: 1; No. of load stages: 1.
- Case Settings (Cases 1-5):** Each case has fields for Load file name (NULL), Load case title (Enter load case title), Initial factor (0), Final factor (0), Inc. factor (0), Load type (Monotonic), Repetitions (1), Cyclic Inc. factor (0), and Initial Load Stage (1).
- Analysis Parameters:** Seed File Name: NULL; Max. no. of Iterations: 60; Dynamic Averaging Factor: 0.6; Convergence Limit: 1.00001; Convergence Criteria: Displacements - Weighted Average; Analysis Mode: Static Nonlinear - Load Step; Results Files: ASCII Files Only; Modeling Format: Stand Alone Modeling.

Figure A-52

A.2.4.1. Analysis Job Control Parameters Definition

This step is responsible for defining the main framework of the analysis. This is done through the first tab in the 'Job Definition' window, titled 'Job Control'. The user input is composed of four main parts, in four collective frames, namely, 'Job Data', 'Structure Data', 'Loading Data', and 'Analysis Parameters'. The user input in this step is saved in the 'VECTOR.JOB' input file.

The first user-input part is designed for entering the analysis job data. In the collective frame titled 'Job Data', the job file name (the 'JOB' file), job title, and job date can be entered. It should be noted that the job file name should always be left as the default entry 'VecTor'.

The second user-input part is designed for entering the analysis structure model data. In the collective frame titled 'Structural Data', the structure file name (the 'S3R' file), structure title, and structure type can be entered. Since the analysis type described here is for three-dimensional structures using VecTor3, the user should leave the structure type as the default entry 'Solid (3-D)'.

The third user-input part is designed for entering the loading parameters, which is done in the collective frame titled 'Loading Data'. In the top part of this collective frame (as shown in Figure A-52), the user can enter the title of the output files generated by the analysis in the box titled 'Load series ID'. The output files will be titled using the title entered in this box followed by an underscore sign '_', then the load stage number. The output ASCII text files will have the extension 'A3E' and the output binary files will have the extension 'A3R'. The format of output files will be explained later.

The user can also specify the ID number by which the output ASCII text files should start to be titled from. This is done by entering the desired value of the ID number of the first load stage in the box titled 'Starting load stage no.'. This means that if the default output text files title 'ID' is chosen and the ID number of the first load stage is chosen to be 'i', then the title of the first output ASCII text file and binary file will be 'ID_i.A3E' and 'ID_i.A3E', respectively. The last input entry in this box is the total number of load stages to be analyzed in the analysis job, which can be entered in the box titled 'No. of load stages'. As will be explained later, the analysis will run in the load increments to be specified later for a number of increments equal to the number of load stages entered in this box.

The next step in this part is defining the analysis loading scheme for the loading conditions that will be defined later. As previously explained, VecTor3 allows for five different loading cases (of similar or different types) to be analyzed simultaneously. The number of loading cases to be analyzed should be specified in this part by ticking the

checkboxes titled 'Case 1' to 'Case 5' (as shown in Figure A-52) depending on the number of loading cases the user desires to be superimposed.

When the user ticks the checkbox titled 'Case 1' (which is mandatory for an analysis to be started), the greyed-out entry boxes provided for this loading case become available for user input, as shown in Figure A-53. At this point, the user can enter the load file name, which will be assigned the extension 'L3R' as previously explained, and the loading case title, which is only used for referencing. In the next box, titled 'Initial factor', the user is required to enter the factor by which the load values assigned to the current loading case are to be multiplied by (referred to as the "load factor") for the first load stage to be analyzed. The next box, titled 'Final factor', is used to enter the maximum load factor for the current loading case; and the box titled 'Inc. factor' is used to enter the increment value in the load factor for the current loading case, starting from the 'Initial factor' and increasing up to the 'Final factor'. The 'Initial factor', 'Final factor' and 'Inc. factor' can all have either positive or negative values; and the meaning of these factors changes with the different load types and analysis modes as will be explained later.

The next user input is in the form of a drop list titled 'Load type'. In this drop list, the user can select the type of loading to be analyzed, where the options available are: 'Monotonic' for constant or increasing loads, 'Cyclic' for cyclic loads that are repetitive in the same direction, and 'Reverse Cyclic' for cyclic loads that reverse the direction of loading at the end of each cycle. These loading types are further explained in the VecTor2 and FormWorks User's Manual by Wong et al. (2013).

Selecting the 'Cyclic' and the 'Reverse Cyclic' options makes the boxes 'Repetitions' and 'Cyclic Inc. factor' available for user input. For those load types, the 'Final factor' refers to the peak load factor in the loading cycle. 'Cyclic Inc. factor' refers to the increment in the maximum load factor, 'Final Factor', from one cycle to the next; and 'Repetitions' refers to the number of cycles to be run at each specific value of the peak load factor before applying the increment specified in the 'Cyclic Inc. factor'. More about the 'Cyclic' and the 'Reverse Cyclic' load types can be found in the VecTor2 and FormWorks User's Manual by Wong et al. (2013).

The 'Define Job' dialog box contains the following sections and fields:

- Job Data:** Job file name: VecTor; Job title: Enter Job Title; Date: Enter Date.
- Structure Data:** Structure file name: Struct; Structure title: Enter Structure Title; Structure type: Solid (3-D).
- Loading Data:** Load series ID: ID; Starting load stage no.: 1; No. of load stages: 1.
- Case Configuration:**
 - Activate: Case 1, Case 2, Case 3, Case 4, Case 5
 - Load file name: Case1, NULL, NULL, NULL, NULL
 - Load case title: Enter load case title, Enter load case title, Enter load case title, Enter load case title, Enter load case title
 - Initial factor: 0, 0, 0, 0, 0
 - Final factor: 0, 0, 0, 0, 0
 - Inc. factor: 0, 0, 0, 0, 0
 - Load type: Monotonic, Monotonic, Monotonic, Monotonic, Monotonic
 - Repetitions: 1, 1, 1, 1, 1
 - Cyclic Inc. factor: 0, 0, 0, 0, 0
 - Initial Load Stage: 1, 1, 1, 1, 1
- Analysis Parameters:**
 - Seed File Name: NULL
 - Max. no. of Iterations: 60
 - Dynamic Averaging Factor: 0.6
 - Convergence Limit: 1.00001
 - Convergence Criteria: Displacements - Weighted Average
 - Analysis Mode: Static Nonlinear - Load Step
 - Results Files: ASCII Files Only
 - Modeling Format: Stand Alone Modeling

Figure A-53

As will be explained later, there are multiple analysis modes available in VecTor3, which can be selected from the drop list titled 'Analysis Mode' in the lower part of this tab on the right-hand side. The input entries explained above apply to the 'Linear Elastic' and the 'Static Nonlinear - Load Step' modes, both of which will be explained later. For the other analysis modes, which all involve time-stepping procedure, the 'Initial factor' refers to the load factor throughout the analysis, on which no increment will be applied; the entry of the 'Final factor' is discarded; and the 'Inc. factor' refers to the time step (time increment) to be used in the analysis (in seconds).

The 'Repetitions' in this case refers to the number of time sub-steps in each time step. This means that the time step entered in the 'Inc. factor' box will be divided by the number of time sub-steps entered in the 'Repetitions' box, and a separate analysis will be run at time steps equal to the outcome of this calculation. Yet, results output files will

only be generated at the time steps (time increments) entered in the 'Inc. factor' box. The aim of this feature is to enable the user to run the dynamic analysis or heat transfer analysis with smaller time steps to achieve a more accurate and stable analysis, but not have an unnecessary large number of results output files.

The final input box in this collective frame is titled 'Initial load stage'. This entry allows the user to specify the load stage at which the current loading case will be started. Since five loading cases are allowed to be run simultaneously in the same analysis job, the user can choose to start a certain loading case midway or at the end of the application of another loading case. This can be done by specifying the 'Initial load stage' of the loading case. If the user needs all the loading cases to start simultaneously at the beginning of the analysis, the 'Initial load stage' should be left as 1, which is the default value.

The fourth part in this tab is for specifying the analysis parameters for the analysis job. In order to understand the input entries in this part, a brief background of the procedure that VecTor3 follows for structural analysis needs to be presented. VecTor3 performs nonlinear analysis of reinforced concrete structures based on the Modified Compression Field Theory (MCFT) (Vecchio and Collins, 1986) in a total-load secant-stiffness approach. This means that at each load stage, the total load is applied and the secant stiffnesses of the elements are used to calculate the displacements of the nodes comprising the structural model, which are, in turn, used to calculate the strains in the elements. The stresses are then calculated for the elements based on the calculated strains and the constitutive stress-strain models of the elements' materials. Finally, those strains and stresses are used to recalculate the value of the secant stiffnesses of the elements which are to be reused for the solution. This procedure is referred to as an "iteration" in the current load stage. The results of each iteration are compared to those of the previous iteration until certain criteria are met, after which the analysis proceeds to the next load stage.

With this background of the analysis procedure in mind, the entries of the fourth collective frame in this tab, titled 'Analysis Parameters', can be explained. The box titled 'Convergence Criteria' refers to the criteria that need to be met for the solution of the iteration to be deemed final so that the analysis can proceed to the next load stage. The 'Convergence Criteria' compare certain results of each iteration to the results of the

previous iteration. The options for which results are to be compared that are available in VecTor3 can be selected through the drop list and they include:

1. 'Principal Secant Moduli - Weighted Average': A weighted average of the principal secant moduli of all the elements is calculated and used for testing convergence.
2. 'Displacements - Weighted Average': A weighted average of all the displacements of the nodes comprising the structural model is calculated and used for testing convergence.
3. 'Displacements - Maximum Value': The maximum displacement in the nodes comprising the structural model is used for testing convergence.
4. 'Reactions - Weighted Average': A weighted average of all the reactions of the restrained nodes in the structural model is calculated and used for testing convergence.
5. 'Reactions - Maximum Value': The maximum reaction in the restrained nodes in the structural model is used for testing convergence.
6. 'Global Secant Moduli - Weighted Average': A weighted average of the principal secant moduli rotated to the global coordinates according to the principal stress directions of all the elements is calculated and used for testing convergence.
7. 'Secant Moduli, Displacements, & Reactions - Weighted Average': the values of all six previous options are calculated and collectively used for testing convergence.

The 'Weighted Average' is calculated as the square root of the sum of squares of the ratios of the specific result value in the current to previous iteration for all the nodes or elements; and the 'Maximum Value' is calculated based on the ratio of the maximum value of the specific result value in the current iteration to its respective value in the previous iteration.

These values introduce what are called 'convergence factors', which represent the relation between the results of the current iteration to the previous iteration. These convergence factors have a value of one when perfect convergence is achieved, i.e., when the results of the current iteration are exactly equal to those of the previous iteration. For a load stage to be finalized and for the analysis to proceed to the next load stage, one of two criteria needs to be met. The first criterion is that the convergence factor calculated at

the end of a certain iteration is less than a certain predefined limit, which can be entered by the user in the box titled 'Convergence limit', with a value greater than one. The second criterion is that the number of iterations carried out in the current load stage reaches a predefined maximum number that can be entered by the user in the box titled 'Max. no. of iterations'.

The final entry on this topic falls in the entry box titled 'Averaging factor' which specifies a certain factor ranging from 0 to 1. This factor represents the percentage of the value of the secant stiffness of each element that is calculated from the previous iteration to be used in the solution of the current iteration. The importance of this factor lies in the difficulty of reaching convergence for the highly nonlinear procedure involved in the analyses carried out using VecTor3. A smaller averaging factor results in a more stable analysis and smoother convergence from one iteration to the next. A larger averaging factor, on the other hand, results in faster convergence. Therefore, the user has to make a judgement on the value to be used for the averaging factor. The option of ticking the box titled 'Dynamic' allows the user to let VecTor3 choose an appropriate averaging factor according to the response of the structure, where larger averaging factors are used for fast converging elements and smaller averaging factors are used for slow converging elements. This means that with the 'Dynamic' option, each element has a different averaging factor and it keeps changing from one iteration to the next.

It has been found that, in general, flexure-dominant responses do not require a small averaging factor to reach convergence, where a value of 0.50 or more would be acceptable. However, for shear-dominant responses, smaller averaging factors are required. Also, it was found that a small averaging factor is required for the analysis of reinforced concrete members that are subjected to uniform stresses without any weak or overly-stressed part where failure is expected to initiate, such as columns subjected to pure axial loads.

It should be noted that, in VecTor3, if the number of iterations selected by the user is greater than 100, the averaging factor selected is reduced by 5% every 5 iterations. It has been found that this constantly changing averaging factor leads to faster convergence and a more stable analysis. This is because the analysis starts with a large averaging factor

which results in faster convergence, but this averaging factor slowly decreases in order to fine-tune for better convergence.

Also, in this user-input part which is responsible for the analysis parameters of the analysis job, the drop list titled 'Analysis mode' is available for the user to select the type of the analysis required to be carried out. The options available are:

1. 'Linear Elastic', where a static loading procedure that is independent of the loading rate and time is followed. A linear elastic analysis is followed for both concrete and steel material behaviour.
2. 'Static Nonlinear - Load Step', where a static loading procedure that is independent of the loading rate and time is followed. The material behaviour is nonlinear.
3. 'Static Nonlinear - Time Step', where a static time-stepping loading procedure that is independent of the loading rate is followed. The material behaviour is nonlinear. This is the type of analysis that is responsible for carrying out coupled thermal and structural analysis.
4. 'Dynamic Nonlinear – General', where dynamic loading procedure is followed. This is the type of analysis that is responsible for carrying out dynamic analysis based on ground acceleration records defined by a maximum of 50 acceleration-time key points applied over the entire structure or impulse loading records defined by a maximum of three force-time key points applied at any number of nodes in the structural model.
5. 'Dynamic Nonlinear - EQ Record', where dynamic loading procedure is followed. This is the type of analysis that is responsible for carrying out dynamic analysis based on an earthquake record generated by the user in a separate file having the title 'VECTOR.EQR'. This earthquake acceleration-time profile is applied over the entire structure. This is similar to the 'Dynamic Nonlinear – General' for ground acceleration records except for the ability to define the acceleration-time profile with up to five hundred key points and in a separate file. The formatting of the file 'VECTOR.EQR' is shown through a sample file in Appendix A.
6. 'Dynamic Nonlinear - Impulse Record', where dynamic loading procedure is followed. This is the type of analysis that is responsible for carrying out dynamic analysis based on impulse loading records defined by a maximum of 50 force-time

key points applied at any number of nodes in the structural model. This is similar to the 'Dynamic Nonlinear – General' for impulse loads analysis except for the ability to define the force-time profile with up to fifty key points rather than three key points.

Also in this user-input part, the drop box titled 'Results files' enables the user to select the types of results' output files required to be generated by VecTor3. Two types of files can be generated for each load stage: text ASCII files with the extension 'A3E' and binary files with the extension 'A3R'. The text ASCII files are the ones that can be read using the post-processor, Janus, for graphical display of the results, or can be read as plain text files by the user. The binary files, on the other hand, have a special purpose, where they save the status of the structure at the end of the load stage, enabling the user to start a subsequent analysis based on this status which acts as a 'seed'. There are five options in this drop list available for the user as follows:

1. 'ASCII and Binary Files', where both types of files are generated for every load stage.
2. 'ASCII Files Only and Binary File for Last Load Stage', where text ASCII files are generated for every load stage and a binary file is generated for the last load stage analyzed only.
3. 'Binary Files Only', where only binary files are generated for every load stage.
4. 'Last Load Stage Only', where both types of files are generated for the last load stage analyzed only.
5. 'ASCII Files Only', where only text ASCII files are generated for every load stage.

As explained above, the user can use any binary file as a 'seed' to start an analysis job starting with the status of the structure that is saved in the binary file. The binary file in this case is called the 'seed file' of the new analysis job. If the analysis job being defined is new, i.e., starting from the initial unloaded state of the structure, the entry in the box titled 'Seed File Name' should be set to 'NULL', which is the default entry. If the analysis being defined is starting from a 'seed file', then the title of the binary file containing the desired initial status should be entered in the box titled 'Seed file name' without the extension 'A3R'.

Finally, the entry in the box titled 'Modelling format' should be left to the default entry, 'Stand Alone Modelling', if the entire structural model will be generated using the element types of VecTor3 only. If the user desires to create a model that combines different element types across the different VecTor programs, the option "Cyrus Sub-Structure' should be selected. The procedure of such analysis will be presented in a separate user's manual.

This concludes the entries required by the user in the 'Job Control' tab in the 'Job Definition' window.

A.2.4.2. Structural Analysis Models Definition

The next tab in the 'Job Definition' window is titled 'Models', and it can be seen in Figure A-54.

This tab allows the user to select the models of the advanced structural mechanisms to be followed in the structural analysis for concrete, steel reinforcing bars, and bond between concrete and steel reinforcing bars. This is in addition to several general analysis models.

The tab is composed of four collective frames, one for each material type with a total of three and one for the analysis models. Those collective frames are titled 'Concrete Models', 'Reinforcement Models', 'Bond Models', and 'Analysis Models'.

The screenshot shows the 'Define Job' dialog box with the following settings:

- Concrete Models:**
 - Compression Pre-Peak: Parabola - Hognestad
 - Compression Post-Peak: Modified Park-Kent
 - Compression Softening: Vecchio 1992-A (e1/e2-Form)
 - Tension Stiffening: Modified Bentz 2003
 - Tension Softening: Linear
 - FRC Tension: SDEM - Monotonic
 - Confined Strength: Kupfer / Richart
 - Dilation: Variable - Kupfer
 - Cracking Criterion: Mohr-Coulomb (Stress)
 - Crack Stress Calc: Basic (DSFM/MCFT)
 - Crack Width Check: Agg/2.5 Max Crack Width
 - Crack Slip Calc: Walraven (Monotonic)
 - Creep and Relaxation: Not Available
 - Hysteretic Response: Nonlinear w/ Plastic Offsets
- Reinforcement Models:**
 - Hysteretic Response: Bauschinger Effect (Seckin)
 - Dowel Action: Tassios (Crack Slip)
 - Buckling: Akkaya 2012 (Refined Dhakal-Ma)
- Bond Models:**
 - Concrete Bond: Eligehausen
- Analysis Models:**
 - Strain History: Previous Loading Considered
 - Strain Rate Effects (Conc./Steel): n/c ; n/c
 - Structural Damping: Not Considered
 - Geometric Nonlinearity: Considered
 - Crack Process: Uniform
- Reset Options:** Basic, Advanced

Figure A-54

The following are lists of the models available for user selection for each advanced structural mechanism and analysis model. More about these models can be found in Wong et al. (2013) and ElMohandes (2013). The user input in this step is saved in the 'VECTOR.JOB' input file.

Concrete Models:

Compression Pre-Peak Response:

- Linear
- Parabola - Hognestad (Hognestad, 1951)
- NSC - Popovics (Popovics, 1973)
- HSC - Popovics (Collins and Porasz, 1989)
- Hoshikuma 1997 (Hoshikuma et al., 1997)

- Smith - Young 1956 (Smith and Young, 1956)
- FRC - Lee 2011
- Attard - Setunge (Attard and Setunge, 1996)
- Elastic - Plastic
- Fire - Lie and Lin + ASCE (Lie and Lin, 1985; ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992))
- Fire - The Eurocode (EN 1992-1-2:2004, 2005)
- Fire - Aslani and Bastami (Aslani and Bastami, 2011)

Compression Post-Peak Response:

- Base Curve
- Modified Park-Kent (Kent and Park, 1971; Park et al., 1982)
- Montoya 2003 (Montoya, 2003)
- Popovics / Mander (Popovics, 1973; Mander et al., 1988)
- Hoshikuma 1997 (Hoshikuma et al., 1997)
- Saenz - Spacone (Saenz, 1964; Kwon and Spacone, 2002)
- FRC - Lee 2011
- Attard - Setunge (Attard and Setunge, 1996)
- Elastic - Plastic

Compression Softening Model:

- No Compression Softening
- Vecchio 1992-A (e_1/e_2 -Form) (Vecchio and Collins, 1993)
- Vecchio 1992-B (e_1/e_0 -Form) (Vecchio and Collins, 1993)
- Vecchio-Collins 1982 (Vecchio, 1982)
- Vecchio-Collins 1986 (Vecchio and Collins, 1986)
- Vecchio 1992-C
- Maekawa 1978 (Miyahara et al., 1987, 1988)
- Mehlhorn 1990 (Kollegger and Mehlhorn, 1990)
- Noguchi 1989 (Shirai and Noguchi, 1989)

Tension Stiffening Model:

- No Tension Stiffening
- Modified Bentz 2003 (Sato and Vecchio, 2003)
- Vecchio 1982 (Vecchio, 1982)
- Collins-Mitchell 1987 (Collins and Mitchell, 1987)
- Bentz 1999 (Bentz, 2000)
- Izumo 1991 (Izumo et al., 1991)
- Lee 2010 (w/ Post-Yield)

Tension Softening Model:

- Not Considered
- Linear
- Bilinear
- Nonlinear - Yamamoto (Yamamoto, 1999)
- Nonlinear - Hordijk (Hordijk, 1991)
- Spinella (Spinella et al., 2010)
- Custom Input (Strain Based) (user-defined stress-strain curve in the auxiliary tab)
- Custom Input (Crack Based) (user-defined stress-strain curve in the auxiliary tab)
- FRC - Lee 2011 Exponential Model
- FRC (fib Model Code 2010)

FRC Post-Crack Tension Model:

- Not Considered
- SDEM - Monotonic (Lee et al., 2013)
- SDEM - Cyclic
- DEM (Lee et al., 2011a, 2011b)
- VEM (Voo and Foster, 2003)
- fib Model Code 2010

Confined Strength Model:

- Strength Enhancement Neglected

- Kupfer - Richart (Richart et al., 1928; Kupfer et al., 1969)
- Montoya - Ottosen (Ottosen, 1979; Montoya, 2003)
- Selby (Selby and Vecchio, 1997)
- Cui & Sheikh (Cui and Sheikh, 2010)

Dilatation (Lateral Expansion):

- Constant Poisson's Ratio
- Variable - Kupfer (Kupfer et al., 1969)
- Variable - Montoya 2003 (Montoya, 2003)
- Variable - Montoya w/ Limit (Montoya, 2003)

Cracking Criterion:

- Uniaxial Cracking Stress
- Mohr - Coulomb (Stress) (Coulomb, 1776; Mohr, 1900)
- Mohr - Coulomb (Strain) (Coulomb, 1776; Mohr, 1900)
- CEB-FIP Model
- Gupta 1998 (Gupta, 1998)
- Mohr-Coulomb (w/ Strain History) (Coulomb, 1776; Mohr, 1900)

Crack Slip Check (for MCFT):

- Crack Stress Calcs Omitted
- Basic (DSFM/MCFT)
- Advanced (Lee 2009)

Crack Width Check:

- Stability Check Omitted
- Agg/2.5 Max Crack Width (maximum aggregate size divided by 2.5)
- Agg/5.0 Max Crack Width (maximum aggregate size divided by 5.0)
- 10 mm Max Crack Width
- 5 mm Max Crack Width
- 2 mm Max Crack Width
- 1 mm Max Crack Width

Crack Slip Calculation (Element Slip Distortion for DSFM):

- Not Considered
- Walraven (Monotonic) (Walraven and Reinhardt, 1981)
- Vecchio-Lai (Cyclic) (Vecchio and Lai, 2004)
- Maekawa (Monotonic) (Okamura and Maekawa, 1991)
- Hybrid I (Walraven) (Walraven and Reinhardt, 1981; with preset minimum lag)
- Hybrid II (Vecchio-Lai) (Vecchio and Lai, 2004; with preset minimum lag)
- Hybrid III (Maekawa) (Okamura and Maekawa, 1991; with preset minimum lag)

Creep and Relaxation:

- Not Available (The input of concrete creep parameters is disabled in this tab but will be considered in the next tab which is titled 'Auxiliary')

Hysteretic Response:

- Linear w/ No Plastic Offsets
- Linear w/ Plastic Offsets
- Nonlinear w/ Plastic Offsets
- Palermo 2002 (w/ Decay) (Palermo, 2002)

Reinforcement Models:

Hysteretic Response:

- Linear
- Bauschinger Effect (Seckin) (Bauschinger, 1886; Seckin, 1981)
- Elastic-Hardening (Curvilinear)
- Elastic-Hardening (Trilinear)
- Elastic-Plastic (Bilinear)
- Seckin (w/ Local Accumulation) (Bauschinger, 1886; Seckin, 1981)

Dowel Action:

- Not Considered

- Tassios (Crack Slip) (Vintzeleou and Tassios, 1987)
- Tassios (Strength) (Vintzeleou and Tassios, 1987)

Buckling:

- Not Considered
- Akkaya 2012 (Refined Dhakal-Maekawa) (Dhakal and Maekawa, 2002)
- Dhakal-Maekawa 2002 (Dhakal and Maekawa, 2002)
- Asatsu Model (Asatsu (Asazu) et al., 2001)

Bond Models:

Concrete Bond:

- Perfect Bond
- Eligehausen (Eligehausen et al., 1983)
- Gan-Vecchio (Gan, 2000)
- Harajli (Harajli and Mukaddam, 1988)
- Fujii (Morita and Fujii, 1982)
- Eligehausen (No Cyclic Damage) (Eligehausen et al., 1983)
- Gan (No Cyclic Damage) (Gan, 2000)

Analysis Models:

Strain History:

- Previous Loading Neglected
- Previous Loading Considered

Strain Rate Effects (the first entry is for concrete and the second is for steel):

- n/c ; n/c (not considered)
- n/c ; Malvar-Crawford (Malvar and Crawford, 1998)
- n/c ; CEB-FIP 1988 (CEB, 1988)
- fib MC 2010 ; n/c (fib Model Code 2010, 2012)

- fib MC 2010 ; Malvar-Crawford (fib Model Code 2010, 2012; Malvar and Crawford, 1998)
- fib MC 2010 ; CEB-FIP 1988 (fib Model Code 2010, 2012; CEB, 1988)
- fib MC 1990 ; n/c (CEB-FIP Model Code 1990, 1993)
- fib MC 1990 ; Malvar-Crawford (CEB-FIP Model Code 1990, 1993; Malvar and Crawford, 1998)
- fib MC 1990 ; CEB-FIP 1988 (CEB-FIP Model Code 1990, 1993; CEB, 1988)

Structural Damping:

- Not Considered
- Rayleigh Damping (Rayleigh, 1877)
- Alternative Damping

Geometric Nonlinearity:

- Not Considered
- Considered

Crack Process:

- Uniform

For VecTor3, it is required that the drop list of the 'Crack Process' be left as its default and only option 'Uniform'. More about those models is available in the in the VecTor2 and FormWorks User's Manual by Wong et al. (2013) and in ElMohandes (2013). Besides the ability of the user to select the most preferable models according to the conditions of the structure to be analyzed, there is a shortcut available in the lower part of the right-hand side of this tab in a box titled 'Reset Options'. The user can select 'Basic' or 'Advanced' for two available preset groups of models that are preferred by the program's developers.

A.2.4.3. Auxiliary Parameters Definition

The next tab in the 'Job Definition' window is titled 'Auxiliary', and it can be seen in Figure A-55. This tab allows the user to define several additional parameters that are

required for special types of structures, loading conditions, and analysis types. The user input in this step is saved in the 'VT3.AUX' input file.

The 'Define Job' dialog box is shown with the 'Auxiliary Thermal' tab selected. The 'General' section includes: Stiffness Matrix Solver Type (Solver 2), Quadrilateral Element Type (Isoparametric), Concrete Aggregate Type (Carbonate), Shear Protection, Concrete Fracture Energy (0 kN/m), Concrete Thermal Conductivity (2.19 W/mK), Prestressing Friction Coefficient (0.3), Prestressing Wobble Coefficient (0.0025), and Thermal Time Stepping Factor (0.6666667). The 'Dynamic Analysis' section includes: Reference Mode #1 (1), Reference Mode #2 (2), Damping Factor #1 (%) (0), Damping Factor #2 (%) (0), Mass Factor due to Self-Weight (1), Time Integration Method (Newmark Constant), and Ground Acceleration in x, y, and z directions (all Not Considered). The 'Tension Softening' section has four points with Strain and Stress values. The 'Masonry Structures' section includes Principal Direction, Masonry Joint Thickness, Joint Shear Strength Ratio, Comp Strength Ratio, Tensile Strength Ratio, Elastic Modulus Ratio, Friction Angle, and Strength Reduction Factor. The 'Material Resistance / Creep Factors' section includes Resistance Factors for Concrete, Rebar Steel, P/S Steel, Structural Steel, Masonry/Mortar, and Wood/Ortho, as well as Concrete Creep and P/S Relaxation Coefficients. A 'Reset Default' button is located at the bottom right of the dialog area. 'OK', 'Cancel', and 'Apply' buttons are at the bottom of the window.

Figure A-55

In the first collective frame, titled 'General', the user can select between two solvers available for the solution of the equations of the displacements of the present degrees of freedom in the finite element procedure. 'Solver 1' relies on the creation of scratch files to store the values involved in the solution, while 'Solver 2' relies on storing those numbers in the virtual memory. This makes 'Solver 2' much faster than 'Solver 1', which is why it is the default selection.

Since there is a separate tab, the 'Auxiliary Thermal' tab, for the input of the properties of concrete related to the thermal analysis, the input entered by the user in this tab for those properties will be overridden by the input entered in the 'Auxiliary Thermal' tab. Those

properties include the 'Concrete Aggregate Type', 'Concrete Thermal Conductivity', and 'Thermal Time Stepping Factor'.

The value of the 'Concrete Fracture Energy' is required in case the model selected in the 'Tension Softening' drop list in the 'Models' tab discussed in the previous section (Section A.2.4.2 of the *VecTor3: User's Manual*) depends on it. This only applies to the 'Nonlinear - Hordijk' and the 'FRC - Lee 2011 Exponential Model' options. However, should this entry be left at the default value, '0', and one of those two options is selected for the 'Tension Softening' model, the 'Concrete Fracture Energy' value is calculated by VecTor3 according to the model proposed by Bažant and Becq-Giraudon (2002).

The 'Prestressing Friction Coefficient' and the 'Prestressing Wobble Coefficient' are required for estimating the friction losses in unbonded tendons. Therefore, these entries are only used when the 'unbonded tendons w/ friction losses' option is selected in the 'Bond Properties Definition' step (Section A.2.2.3 of the *VecTor3: User's Manual*).

Another collective frame, titled 'Tension Softening', allows the user to enter four key points to define a custom stress-strain curve for unreinforced concrete in tension. The boxes titled 'Tension Softening Pt.1: Strain' and 'Tension Softening Pt.1: Stress' are designated for the first key point on the curve, where the former box is designated for the strain entry and the latter is designated for the corresponding stress. This applies for the next six boxes with the 'Pt.2', 'Pt.3', and 'Pt.4' for the second, third, and fourth key points, respectively, where the first box of each key point is designated for the strain and the next box is designated for the corresponding stress. This curve will only be used when the option 'Custom Input (Strain Based)' or the option 'Custom Input (Crack Based)' is selected in the 'Tension Softening' drop list in the 'Models' tab as discussed in the previous section (Section A.2.4.2 of the *VecTor3: User's Manual*).

The collective frame titled 'Dynamic Analysis' in the upper part of the right-hand side of this tab is responsible for defining several additional parameters related to dynamic analysis. 'Reference Mode #1' and 'Reference Mode #2' refer to the first and second modes of vibration for Rayleigh damping procedure (Rayleigh, 1877), respectively. 'Damping Factor #1' and 'Damping Factor #2' are the corresponding damping factors for these first and second modes of vibration, respectively. It should be noted that VecTor3

only considers the first 100 modes of vibration in order to save time and computational resources.

VecTor3 uses Newmark's time integration method for the dynamic analysis solution (Newmark, 1959). Two options are available for the use of the method, which can be selected through the 'Time Integration Method' drop list, namely, 'Newmark Constant' and 'Newmark Linear'. The constant factors γ and β , which determine the variation of acceleration within the time step, are set according to this selection. 'Newmark Constant' refers to constant acceleration within the time step based on an average of its value at the beginning and the end of the time step. For this option, γ is set to $1/2$ and β is set to $1/4$. 'Newmark Linear' refers to linear variation of the acceleration within the time step. For this option, γ is set to $1/2$ and β is set to $1/6$.

The next three entries, titled 'Ground Acceleration in X-Direction', 'Ground Acceleration in Y-Direction', and 'Ground Acceleration in Z-Direction', are responsible for determining the directions in which the ground acceleration record is to be applied. This type of structural analysis involving a ground acceleration record requires the selection of a special analysis type as explained earlier and a special loading input as will be explained later.

Finally, the lower part on the right-hand side of the tab presents a collective frame titled 'Material Resistance / Creep Factors'. The first four entries in this collective frame aim at facilitating the structural design and structural assessment processes, where a resistance factor can be assigned to the material properties of the various types of material that were defined in the 'Material Definition' step, including concrete, steel reinforcing bars, prestressing tendons, and structural steel. The strengths of those material types will be multiplied by their respective resistance factors throughout the analysis. This enables the user to assign the desired material factor of safety to each of the material types that have been defined.

Also, the second last entry in this collective frame allows the user to account for the creep of concrete with time by assigning a 'Concrete Creep Coefficient'. The initial stiffness of concrete will be divided, throughout the analysis, by a value of one plus this value entered for this coefficient, so will the strain corresponding to the peak compressive

stress. Finally, the last input box allows the user to account for the relaxation of prestressing tendons with time by assigning a 'P/S Relaxation Coefficient'. Young's modulus of the prestressing tendons will be divided, throughout the analysis, by a value of one plus the value entered for this coefficient.

A.2.4.4. Auxiliary Thermal Parameters Definition

This tab in the 'Job Definition' window is titled 'Auxiliary Thermal', and it can be seen in Figure A-56. This tab allows the user to define several additional parameters that are required for the coupled thermal and structural analysis. The entries in this tab involve both the heat transfer analysis through concrete and the coupled thermal and structural analysis. The user input in this step is saved in the 'VT3T.AUX' input file. The first major entry in this tab is the 'Thermal Time Stepping Factor'. This factor varies from 0 to 1 and it is required for the time discretization in the finite difference solution scheme of the heat transfer analysis. For more information about this factor, refer to ElMohandes (2013).

The user is required to specify the type of aggregate used in the concrete mix through the drop list titled 'Concrete Aggregate Type', where three options are available: 'Calcareous', 'Siliceous', and 'Lightweight'. The user is also allowed to specify the 'Convective Heat Transfer Coefficient' and 'Emissivity' of concrete for the convective heat transfer analysis cases. Also, the 'Initial Density' of concrete and 'Initial Moisture Content' can be specified by the user.

For coupled heat and moisture transfer analysis, additional properties of concrete need to be specified by the user, namely, the 'Initial Porosity' and the 'Initial Permeability'.

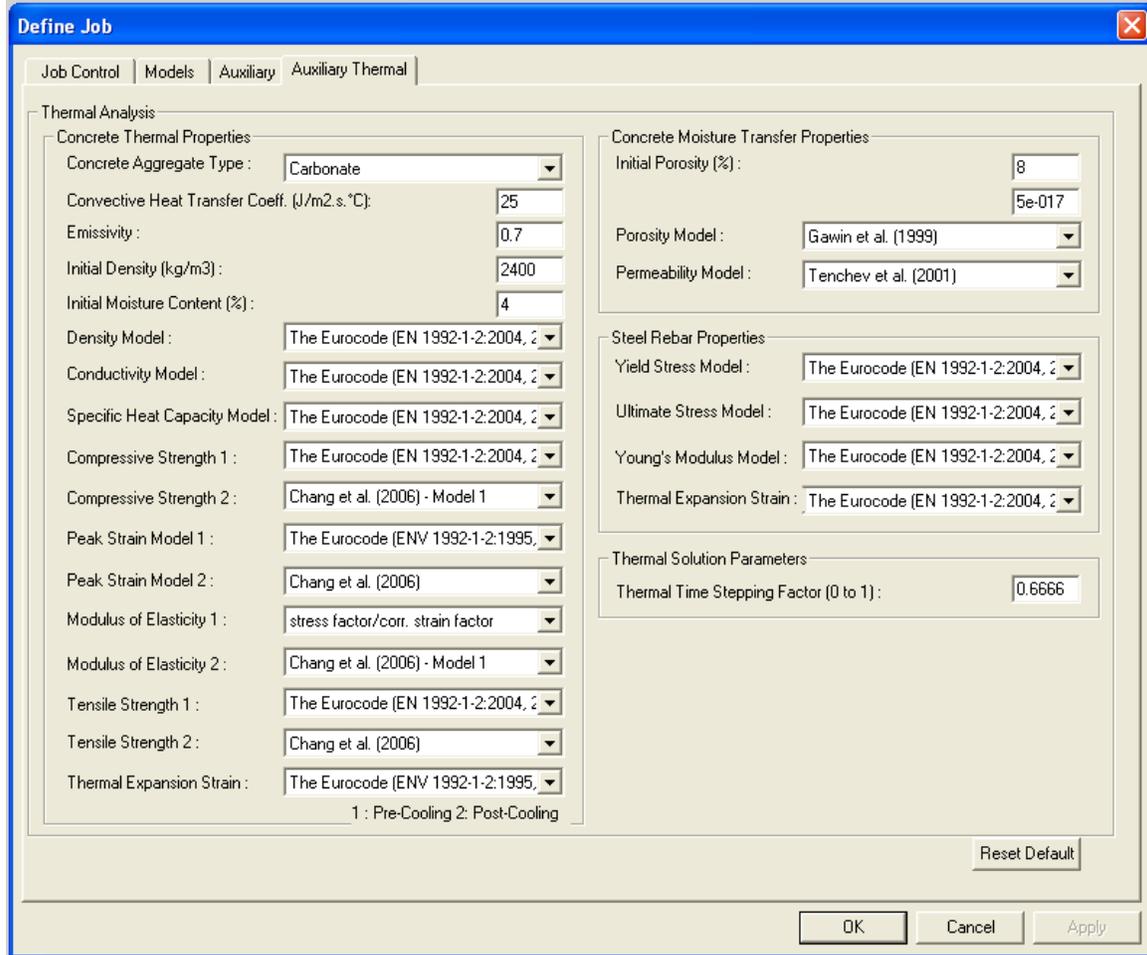


Figure A-56

For the thermal and structural properties of concrete and steel that are affected by the rise in temperature, the following are lists of the models available for user selection for those properties. More about these models can be found in ElMohandes (2013).

For concrete:

Density:

- Lie and Kodur (1996) - for SFRC
- Shin et al. (2002)
- Kodur and Sultan (2003) - for HSC
- Kodur and Sultan (2003) - for SFR - HSC
- The Eurocode (EN 1992-1-2:2004, 2005)

Thermal conductivity:

- The ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- Lie and Kodur (1996) - for SFRC
- Shin et al. (2002)
- Kodur and Sultan (2003) - for HSC
- Kodur and Sultan (2003) - for SFR - HSC
- Kodur et al. (2004) - for HSC
- The Eurocode (EN 1992-1-2:2004, 2005)
- Kodur and Khaliq (2011) - for HSC

Specific heat capacity:

- The ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- Shin et al. (2002)
- Kodur and Sultan (2003) - for HSC
- Kodur and Sultan (2003) - for SFR - HSC
- The Eurocode (EN 1992-1-2:2004, 2005)
- Kodur and Khaliq (2011) - for HSC

Porosity:

- Gawin et al. (1999)
- Toumi and Resheidat (2010)

Permeability:

- Gawin et al. (1999)
- Tenchev et al. (2001)

Compressive strength at elevated temperatures:

- No modification factor
- Lie and Lin (1985) - the ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)

- Bažant and Chern (1987)
- Li and Guo (1993)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- Lie and Kodur (1996) - for SFRC
- Chang and Jau (2001) - for siliceous aggregates
- Cheng et al. (2004) - for HSC
- The Eurocode (EN 1992-1-2:2004, 2005)
- Li and Purkiss (2005)
- Hertz (2005)
- Aslani and Bastami (2011)

Residual compressive strength:

- No modification factor
- Lie et al. (1986)
- Hertz (2005)
- Chang et al. (2006) - Model 1
- Chang et al. (2006) - Model 2

Strain corresponding to peak compressive stress:

- No modification factor
- Lie and Lin (1985) - the ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- Bažant and Chern (1987)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- Lie and Kodur (1996) - for SFRC
- Cheng et al. (2004) - for HSC
- The Eurocode (EN 1992-1-2:2004, 2005)
- Li and Purkiss (2005)

Residual strain corresponding to peak compressive stress:

- No modification factor

- Chang et al. (2006)

Initial modulus of elasticity:

- No modification factor
- Modification factor of peak compressive stress / Modification factor of strain corresponding to peak compressive stress
- Bažant and Chern (1987)
- Lu (1989)
- Li and Guo (1993)
- Li and Purkiss (2005)
- Aslani and Bastami (2011)

Residual initial modulus of elasticity strength:

- No modification factor
- Chang et al. (2006) - Model 1
- Chang et al. (2006) - Model 2

Tensile strength:

- No modification factor
- Bažant and Chern (1987)
- Li and Guo (1993)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- The Eurocode (EN 1992-1-2:2004, 2005)
- Song et al. (2007)
- Moftah (2008)
- Dwaikat and Kodur (2009)
- Aslani and Bastami (2011)

Residual tensile strength:

- No modification factor
- Xie and Qian (1998) - Model 1
- Xie and Qian (1998) - Model 2

- Chang et al. (2006)

Thermal expansion strain:

- The ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- Lie and Kodur (1996) - for SFRC
- The Eurocode (ENV 1992-1-2:1995, 1996)
- The Eurocode (EN 1992-1-2:2004, 2005)
- Kodur and Sultan (2003) - for HSC
- Kodur and Sultan (2003) - for SFR-HSC
- Kodur and Khaliq (2011) - for HSC

For reinforcement:

Yield stress:

- No modification factor
- Brockenbrough (1970)
- The Centre Technique Industriel de la Construction Métallique (CTICM) (1982) - the ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- The European Convention for Constructional Steelwork (ECCS) (1983)
- Saab (1990)
- The Eurocode (ENV 1992-1-2:1995, 1996) - loading based
- The Eurocode (ENV 1992-1-2:1995, 1996) - tension model
- The Eurocode (ENV 1992-1-2:1995, 1996) - compression model
- The Eurocode (EN 1992-1-2:2004, 2005)

Ultimate stress:

- No modification factor
- The Eurocode (EN 1992-1-2:2004, 2005)

Young's modulus:

- No modification factor

- Brockenbrough (1970)
- The Centre Technique Industriel de la Construction Métallique (CTICM) (1982) - the ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- The European Convention for Constructional Steelwork (ECCS) (1983)
- Saab (1990)
- The Eurocode (EN 1992-1-2:2004, 2005)

Thermal expansion strain:

- The European Convention for Constructional Steelwork (ECCS) (1983)
- The ASCE Manual of Practice (Structural Fire Protection by T. T. Lie, 1992)
- The Eurocode (ENV 1992-1-2:1995, 1996)
- The Eurocode (EN 1992-1-2:2004, 2005)

A.2.5. Load Definition

This step is responsible for defining the loads applied on the structural model to be analyzed. After the loading cases are defined in the 'Job Definition' step, the load input toolbar becomes available for user input. The toolbar is shown in Figure A-57.



Figure A-57

The second to sixth buttons from the left in the toolbar shown in Figure A-57, which can be seen in Figure A-58, allow the user to define the various loads to be included in the five separate loading cases allowed by VecTor3. When the button with the number '1' is pressed, all the loads to be defined will be included in 'Case 1' (check the 'Job Control' tab in the 'Job Definition' window shown in Figure A-52). This applies to all five loading cases.



Figure A-58

The user input in this step is saved in the five possible 'L3R' input files, depending on the number of loading cases defined in the 'Job Definition' step, with one file for each loading case.

The different load types that can be analyzed using VecTor3, going through the buttons in the 'Load Definition' toolbar shown in Figure A-57 from left to right, can be summarized in the following list:

1. Nodal Loads
2. Support Displacements (Imposed Displacements)
3. Gravity Loads
4. Elements' Temperatures
5. Concrete Elements' Prestrains
6. Concrete Elements' Ingress Pressures
7. Concrete Elements' Surface Temperatures (Nodal Thermal Loads)
8. Lumped Nodal Masses (for Dynamic Analysis)
9. Impulse Forces
10. Ground Acceleration

Pressing these buttons opens input windows that allow the user to define the loads applied on the structural model by specifying a node's ID number or an element's ID number and the load applied on it. Also, for all those load types (except ground acceleration), there are graphical options available for the user to assign the loads to the nodes and elements without the need to specify their ID number. Those options can be accessed using the two buttons 'Pointer' and 'Window' which appear in the collective frame titled 'Selection Mode' on the right-hand side of the window of every load type. The button 'Pointer' allows the user to select one node or element at a time from the nodes and elements displayed in the workspace view. The button 'Window' allows the

user to select multiple nodes or elements in one step by holding down the left mouse button and dragging a window that contains the desired nodes or elements.

Also for all those load types, after each entry, the 'Apply' button on the right-hand side of the input window should be pressed. Once all the entries are entered, the 'Done' button should be pressed to close the window. Unneeded entries can be deleted by selecting them from the list in the larger box at the bottom of the window then pressing the 'Delete' button on the right-hand side of the window.

At any point, the last button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-59, can be used to delete the loads applied on the entire structure for the loading case selected in the toolbar shown in Figure A-58 (from 1 to 5).



Figure A-59

The following subsections introduce each of the load types available in VecTor3 and the input entries required by the user to define them.

A.2.5.1. Nodal Loads

The seventh button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-60, allows the user to define concentrated point loads on specific nodes in the structural model. Pressing the button opens the window shown in Figure A-61. The user is required to specify the ID number of the node and the concentrated point loads applied on the node in the global X, Y, and Z-directions in kN.

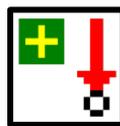


Figure A-60

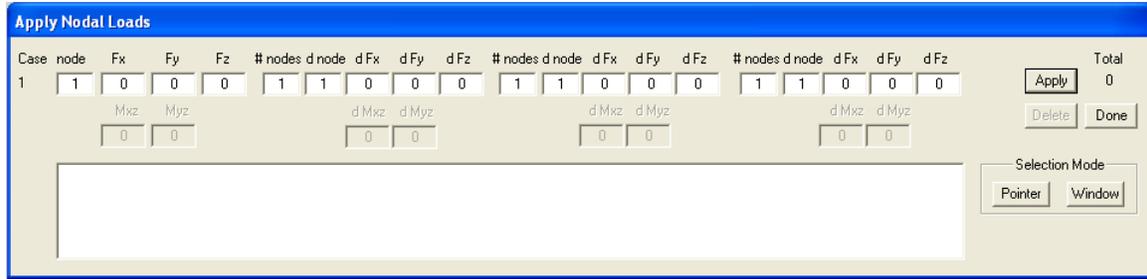


Figure A-61

The user can define the concentrated point loads on the nodes individually, one at a time, or specify the ID number of a certain node with specific concentrated point loads and extrapolate the loads to other nodes by specifying the number of loaded nodes, ‘# nodes’, the increment in the ID number of the subsequent nodes, ‘d(node)’, and the increments in the concentrated load value from one node to the next in the X-direction, ‘d(Fx)’, the Y-direction, ‘d(Fy)’, and the Z-direction, ‘d(Fz)’. This increment in the load value can be beneficial in the cases of triangular and trapezoidal distributed loads. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-61. The ‘Pointer’ and ‘Window’ selection options that were explained earlier are also available.

A.2.5.2. Support Displacements (Imposed Displacements)

The eighth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-62, allows the user to define support displacements (or imposed displacements) on specific nodes in the structural model. Pressing the button opens the window shown in Figure A-63. The user is required to specify the ID number of the node, the direction of displacement in the X, Y, or Z-direction (DOF), and the imposed displacement value in mm.



Figure A-62

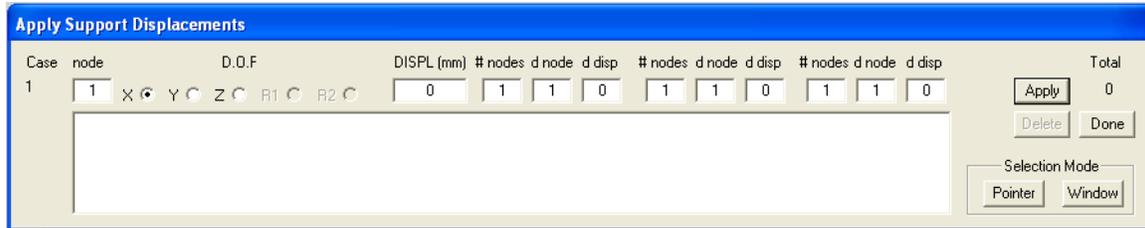


Figure A-63

The user can define the imposed displacements on the nodes individually, one at a time, or specify the ID number of a certain node with specific imposed displacement and extrapolate the displacement to other nodes by specifying the number of nodes with imposed displacement, ‘# nodes’, the increment in the ID number of the subsequent nodes, ‘d(node)’, and the increments in the imposed displacement value from one node to the next, ‘d(displ)’. It should be noted that the direction (DOF) of imposed displacement will be the same for all those nodes whose imposed displacements were defined together. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-63. The ‘Pointer’ and ‘Window’ selection options that were explained earlier are also available.

A.2.5.3. Gravity Loads

The ninth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-64, allows the user to define elements whose gravity loads are to be considered in the analysis. Pressing the button opens the window shown in Figure A-65. The user is required to specify the ID number of the element, the density of the element in kg/m^3 , and the factors by which the element mass contributes to generating weight (force) in the X, Y, or Z-direction (‘GX’, ‘GY’, and ‘GZ’). It should be noted that a positive value for ‘GX’, ‘GY’, and ‘GZ’ means that the gravity force will be applied in the positive direction of the respective axis.



Figure A-64

Case	elmt	DENSITY	GX	GY	GZ	# elmts	d elmt	# elmts	d elmt	# elmts	d elmt	Total
1	1	0	0	1	0	1	1	1	1	1	1	0

* Enter positive value for gravity force acting in the positive direction of coordinate axis

Figure A-65

The user can define the gravity loads of the elements individually, one at a time, or specify the ID number of a certain element whose gravity load is to be considered in the analysis and extrapolate the input to other elements by specifying the number of elements with similar density, ‘# elmts’ and the increment in the ID number of the subsequent elements, ‘d(elmt)’. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-65. The ‘Pointer’ and ‘Window’ selection options that were explained earlier are also available.

A.2.5.4. Elements' Temperatures

The tenth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-66, allows the user to define the temperatures of specific elements in the structural model. Pressing the button opens the window shown in Figure A-67. The user is required to specify the ID number of the element and its temperature in °C.



Figure A-66

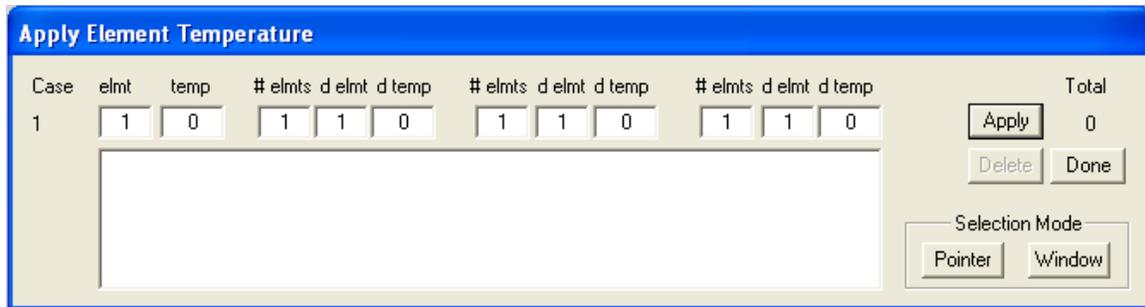


Figure A-67

The user can define the temperatures of the elements individually, one at a time, or specify the ID number of a certain element with a specific temperature and extrapolate the input to other elements by specifying the number of elements whose temperatures need to be specified, ‘# elmts’, the increment in the ID number of the subsequent elements, ‘d(elmt)’, and the increment in the temperature from one element to the next, ‘d(temp)’. This increment in the temperature value can be beneficial in the cases where the user needs to apply previously-known non-uniform temperature distribution through a section. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-67. The ‘Pointer’ and ‘Window’ selection options that were explained earlier are also available.

It should be noted that this type of temperature definition does not involve any heat transfer between the elements. However, the thermal expansion strains and the modified mechanical properties of the materials will be calculated based on the temperatures assigned to the elements.

A.2.5.5. Concrete Elements’ Prestrains

The eleventh button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-68, allows the user to define prestrains in specific concrete elements in the structural model. Pressing the button opens the window shown in Figure A-69. While the option to apply non-uniform prestrains through the elements are planned to be added to VecTor3, they are still under development. At this moment, only a uniform prestrain through the element can be defined. The user is required to specify the ID

number of the element, the type of prestrain, which should be left as the default 'Uniform', and the prestrain value in millistrain.



Figure A-68

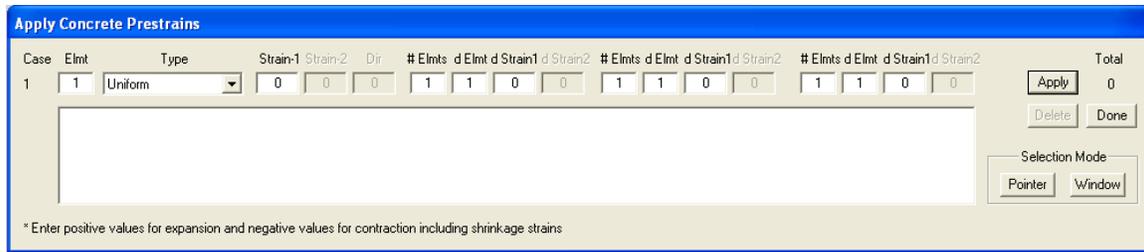


Figure A-69

The user can define the prestains of the elements individually, one at a time, or specify the ID number of a certain element with a specific prestrain and extrapolate the input to other elements by specifying the number of elements with prestains, '# elmts', the increment in the ID number of the subsequent elements, 'd(elmt)', and the increment in the prestrain value from one element to the next, 'd(Strain1)'. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-69. The 'Pointer' and 'Window' selection options that were explained earlier are also available.

A.2.5.6. Concrete Elements' Ingress Pressures

The twelfth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-70, allows the user to define internal pressure in specific elements in the structural model. Pressing the button opens the window shown in Figure A-71. The user is required to specify the ID number of the element and the value of the internal pressure in MPa.



Figure A-70

Case	elmt	pressure	# elmts	d elmt	d pressure	# elmts	d elmt	d pressure	# elmts	d elmt	d pressure	Total
1	1	0	1	1	0	1	1	0	1	1	0	0

Figure A-71

The user can define the internal pressure of the elements individually, one at a time, or specify the ID number of a certain element with a specific internal pressure and extrapolate the input to other elements by specifying the number of elements having internal pressure, ‘# elmts’, the increment in the ID number of the subsequent elements, ‘d(elmt)’, and the increment in the value of the internal pressure from one element to the next, ‘d(pressure)’. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-71. The ‘Pointer’ and ‘Window’ selection options that were explained earlier are also available.

A.2.5.7. Concrete Elements’ Surface Temperatures (Nodal Thermal Loads)

The thirteenth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-72, allows the user to define specific nodes in the structural model that are subjected to elevated temperatures for carrying out the heat transfer analysis through the section. Pressing the button opens the window shown in Figure A-73.



Figure A-72

Case	node	Surface	Pore Pres	Fire Curve Type	Tm1	Tp1	Tm2	Tp2	Tm3	Tp3	Tcool	# nodes d node	# nodesd node	# nodesd node	# nodesd node	Total	
1	1	Atmosphere	<input type="checkbox"/>	Steady-State	0	0	0	0	0	0	0	1	1	1	1	1	0

Figure A-73

The first user input is the ID number of the node. Next, in the input drop list titled ‘Surface’, the user is allowed to select between two options: ‘Element Surface’ or ‘Atmosphere’. ‘Element Surface’ refers to the heat transfer through conduction, where the temperatures of the specified nodes are required to be entered. ‘Atmosphere’ refers to the heat transfer through convection, where the specified nodes are subjected to a surrounding environment with elevated temperatures (through fire, for example); hence, their temperatures are unknown and need to be determined through the analysis. The convection heat transfer analysis carried out using VecTo3 considers both convection and radiation.

The checkbox, titled ‘Pore Pres’, which stands for pore pressure, provides the user with the option whether to consider heat transfer only, where the checkbox should be left blank, or to consider coupled heat and moisture transfer, where the checkbox should be ticked. While the first option is less time-costly and less computationally expensive, the second option provides a calculation of the increase in the pressure in the pores of concrete subjected to elevated temperatures. The value of this pressure is used by VecTor3 to estimate the occurrence of a phenomenon known as ‘explosive spalling’, which involves a forcible ejection of large fragments of concrete separating violently and abruptly when subjected to fire, accompanied by a loud explosive noise. VecTor3 constantly deactivates the elements that are estimated to be spalled as the analysis progresses. This enables VecTor3 to better estimate the heat transfer process as layers of concrete no longer exist and steel reinforcing bars become directly exposed to fire. It also enables VecTor3 to better estimate the structural response as the area of the load-bearing cross section is reduced.

The next drop list, titled 'Fire Curve Type', provides the user with several options for the temperature-time curve that controls the progress of fire (or temperature) with time. The drop list includes the following options:

1. 'Steady State', where a constant temperature (in °C) is assigned to the nodes for steady-state heat transfer analysis. This constant temperature is required to be entered by the user in the box titled 'Tp1'. Other input entries are not required.
2. 'Linear Model', where the user can input a custom temperature-time curve using three key points of the time (in seconds) at 'Tm1', 'Tm2', and 'Tm3' and the corresponding temperature (in °C) at each of these times at 'Tp1', 'Tp2', and 'Tp3', respectively.
3. 'ASTM E119 – 12a (2012) model', where the standard temperature-time curve provided by the ASTM E119 – 12a (2012) will be followed. Other input entries are not required.
4. 'ISO 834-1:1999 (1999) model', where the standard temperature-time curve provided by the ISO 834-1:1999 (1999) model will be followed. Other input entries are not required.

The box titled 'Tcool' allows the user to specify the time at which the fire is extinguished and the nodes start to cool naturally to room temperature in air. This should only be used when one of the standard temperature-time curves is selected. The fire decay model that is followed by VecTor3 is the ISO 834-1:1999 (1999) model. For the 'Linear Model' option, the user can specify the temperature decay using the key points; and for the steady state analysis, an option for temperature decay is unnecessary.

The user can specify the nodes with specific temperature-time profiles individually, one at a time, or specify the ID number of a certain node with a specific temperature-time profile and extrapolate the input to other nodes with the same temperature-time profile by specifying the number of nodes, '# nodes', and the increment in the ID number of the subsequent nodes, 'd(node)'. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-73. The 'Pointer' and 'Window' selection options that were explained earlier are also available.

A.2.5.8. Lumped Nodal Masses (for Dynamic Analysis)

The fourteenth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-74, allows the user to define additional masses lumped at specific nodes in the structural model, which can be used by the user for dynamic analysis purposes. Pressing the button opens the window shown in Figure A-75. The user is required to input the ID number of the node, the directions (DOFs) in which the lumped mass contributes in the dynamic analysis (in the X, Y, and Z-directions), and the mass value in kg.

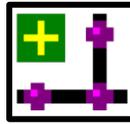


Figure A-74

Case	Node	DOF	Mass (kg)	GF-X	GF-Y	GF-Z	Vo-X	Vo-Y	Vo-Z	# nodes	d node	# nodes	d node	# nodes	d node	Total
1	1	X <input type="checkbox"/> Y <input type="checkbox"/> Z <input type="checkbox"/>	0	0	0	0	0	0	0	1	1	1	1	1	1	0

Buttons: Apply, Delete, Done

Selection Mode: Pointer, Window

Figure A-75

The boxes 'GF-X', 'GF-Y', and 'GF-Z' allow the user to specify the contribution of the defined lumped mass in the X, Y, and Z-directions, respectively. These entries are entered as factors that will be multiplied by the gravitational acceleration, g , in the analysis. This means that a factor of +1 in the 'GF-Z' box, for example, means that the real weight of the lumped mass will be applied in the positive Z-direction, and a factor of -1 means that the real weight of the lumped mass will be applied in the negative Z-direction. The boxes 'Vo-X', 'Vo-Y', and 'Vo-Z' allow the user to specify an initial velocity for the lumped mass in the X, Y, and Z-directions in m/sec, respectively, where these initial velocities will act as the excitation for the dynamic analysis.

These lumped masses are used in all the dynamic analysis types, whether initial velocities have been assigned to them or not. Also, if a density value has been assigned to the material of an element at the 'Material Definition' step (Section A.2.2 of the *VecTor3*:

User's Manual), a mass is calculated for the element based on that density. These masses are assigned to the nodes comprising that element and are added to the values of the lumped masses entered in this input window.

The user can specify the nodes with lumped masses individually, one at a time, or specify the ID number of a certain node with a specific lumped mass and extrapolate the input to other nodes with the same lumped mass, directions of application, factors of contribution in the different directions, and initial velocity. This can be done by specifying the number of nodes, '# nodes' and the increment in the ID number of the subsequent nodes, 'd(node)'. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-75. The 'Pointer' and 'Window' selection options that were explained earlier are also available.

A.2.5.9. Impulse Forces

The fifteenth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-76, allows the user to define impulse loading records applied on specific nodes in the structural model. Pressing the button opens the window shown in Figure A-77. The user is first required to enter the ID number of the node, the direction of the impulse force (DOF) in the X, Y, or Z-direction, and the force-time profile.



Figure A-76

Case	node	# nodes	d node	# nodes	d node	# nodes	d node	d.o.f.	T	F	Total
1	1	1	1	1	1	1	1	X <input type="radio"/> Y <input type="radio"/> Z <input type="radio"/>	0	0	0

Figure A-77

As previously explained, there are two ways to define impulse loading records on specific nodes. The first one is associated with the 'Analysis Mode' titled 'Dynamic Nonlinear – General', which can be selected in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*). This 'Analysis Mode' allows the user to define the impulse loading record at each node using a maximum of three force-time key points. However, this type of input is not available through Formworks-Plus, and can only be entered in the text load file that has the extension 'L3R' as previously explained.

The second way to define impulse loading records, and also the one available through Formworks-Plus, is associated with the 'Analysis Mode' titled 'Dynamic Nonlinear - Impulse Record', which can be selected in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*). This 'Analysis Mode' allows the user to define the impulse loading record of specific nodes using a maximum of 50 force-time key points entered one at a time for each node. For each node, the time, 'T', and the corresponding force, 'F' represent one force-time key point on the impulse force-time loading record. The direction, 'DOF', should be maintained the same for each sequence of key points' entries for each node. Should the same node be subjected to impulse loads in more than one direction, the record of the first direction should be entered in full for the entire time domain, then the entries of the next record can be entered.

To allow the user to check the input for possible input errors, the user can select the entry of a certain node in the box in the lower part of the window then press the 'Graph' button on the right-hand side of the window. This will generate a graphical representation of the force-time impulse loading record at that node.

The user can specify the nodes with impulse loading records individually, one at a time, or specify the ID number of a certain node with an impulse loading record and extrapolate the input to other nodes with the same record. This can be done by specifying the number of nodes, '# nodes' and the increment in the ID number of the subsequent nodes, 'd(node)'. This extrapolation can be done in three levels simultaneously using the three sets of boxes available for user input as shown in Figure A-77. The 'Pointer' and 'Window' selection options that were explained earlier are also available.

A.2.5.10. Ground Acceleration

The sixteenth button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-78, allows the user to define ground acceleration applied on the structural model. Pressing the button opens the window shown in Figure A-79.

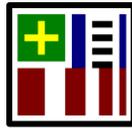


Figure A-78

Case	Time (s)	Acc X (G)	Acc Y (G)	Acc Z (G)
1	0	0	0	0

Buttons: Apply, Delete, Insert, Done

Summary:

X direction	Time (s)	Acc X (G)	Acc Y (G)	Acc Z (G)
Max. PGA	0	0	0	0
Min. PGA	0	0	0	0

Total Duration (s): 0
Number of Data: 0

Figure A-79

As previously explained, there are two ways to define ground acceleration records on the structural model. The first one is associated with the 'Analysis Mode' titled 'Dynamic Nonlinear – General', which can be selected in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*). This 'Analysis Mode' allows the user to define the ground acceleration record using a maximum of 50 acceleration-time key points applied over the entire structure. This can be done through the input window shown in Figure A-79. The user is required to enter the time, 'Time (s)', and the corresponding acceleration as a factor of the ground acceleration, g, in the X, Y, and Z-directions in the boxes titled 'Acc-X (g)', 'Acc-Y (g)', 'Acc-Z (g)', respectively.

Each entry represents one point in time. The collective frame titled 'Summary' on the right-hand side of the input window shows a summary of the input, including the total duration of the record, 'Total Duration', in seconds and the total number of acceleration-time key points, 'Number of Data'. In the drop list at the top, the user can select a certain direction (X, Y, or Z) for which the maximum and minimum ground accelerations will be

displayed in the boxes titled 'Max. PGA' and 'Min. PGA', respectively, together with the corresponding time and the corresponding acceleration values in the other two directions.

The second way to define ground acceleration records is associated with the 'Analysis Mode' titled 'Dynamic Nonlinear - EQ Record', which can be selected in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*). This 'Analysis Mode' allows the user to define the impulse loading record using an earthquake record saved in a separate file with the title 'VECTOR.EQR'. This file cannot be generated using Formworks-Plus, but existing files can be read by Formworks-Plus to enable the user to check the input for possible errors. In order to read the file, the user is required to press the 'Insert' button, which opens a window allowing the user to select the 'VECTOR.EQR' file that needs to be read. The format of a typical 'VECTOR.EQR' file is presented in Appendix A. The earthquake acceleration-time record read from the 'VECTOR.EQR' file is also applied over the entire structure.

With all the load types for all the five available loading cases defined, the user can view a summary of each of those loading case. This can be done by selecting the loading case from the buttons shown in Figure A-58, then pressing the first button from the left in the toolbar shown in Figure A-57, whose shape is shown in Figure A-80.

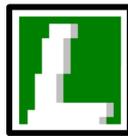
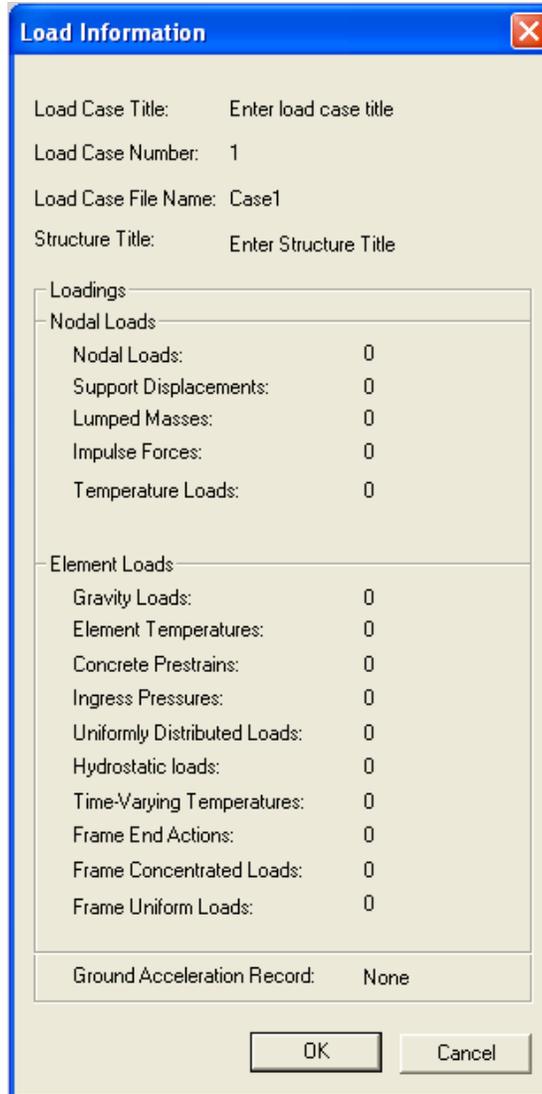


Figure A-80

Pressing this button opens the window shown in Figure A-81, where the 'Load Case Title', 'Load Case Number', 'Load Case File Name', and 'Structure Title' are shown, together with the number of loads of each type that are applied on the structural model.



Load Information	
Load Case Title:	Enter load case title
Load Case Number:	1
Load Case File Name:	Case1
Structure Title:	Enter Structure Title
Loadings	
Nodal Loads	
Nodal Loads:	0
Support Displacements:	0
Lumped Masses:	0
Impulse Forces:	0
Temperature Loads:	0
Element Loads	
Gravity Loads:	0
Element Temperatures:	0
Concrete Prestraints:	0
Ingress Pressures:	0
Uniformly Distributed Loads:	0
Hydrostatic loads:	0
Time-Varying Temperatures:	0
Frame End Actions:	0
Frame Concentrated Loads:	0
Frame Uniform Loads:	0
Ground Acceleration Record:	None

Figure A-81

A.2.6. Saving the Input Files and Running VecTor3

After defining all the parts of the analysis job by following the previous steps, the structural model becomes ready to be analyzed. This can be done using the toolbar shown in Figure A-82. But, the first step is to save the analysis job (if it has not been saved yet) by pressing the button shown in Figure A-5.

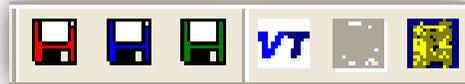


Figure A-82

As explained in the Introduction section (Section A.1 of the *VecTor3: User's Manual*), there are five main input files that are required for defining the analysis job to be carried out. Pressing the first button from the left in the toolbar shown in Figure A-82, featuring the red floppy disc, creates and saves the 'VECTOR.JOB', 'VT3.AUX', and 'VT3T.AUX' files. Pressing the second button, featuring the blue floppy disc, creates and saves the structure file with the name specified in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*) and the extension 'S3R'. Finally, the third button, featuring the green floppy disc, creates and saves the load files (up to five depending on the number of the loading cases defined) with the names specified in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*) and the extension 'L3R'.

At this point, VecTor3 has all the input files it needs and is ready to run. Pressing the fourth button from the left in the toolbar shown in Figure A-82, titled 'VT', starts VecTor3. Also, the user will automatically be prompted to save all input files if the 'VT' button is pressed directly. The start of the analysis will appear in the form of an MS-DOS window that displays the progress of the analysis. For each load stage, the window will show the load stage number and a list of the loading cases and their respective load factors. For time-stepping analyses, the time step number and the time incident being analyzed (in seconds) will also be shown.

Next, the progress of convergence will be displayed for each iteration being analyzed. First, if thermal analysis (nodal thermal loads) is included in one of the loading cases, the convergence progress of the successive iterations of the heat transfer analysis or the coupled heat and moisture transfer analysis will be displayed, depending on whether the user ticked the 'Pore Pres' checkbox in the 'Concrete Elements Surface Temperatures' in the 'Load Definition' step or not (Section A.2.5.7 of the *VecTor3: User's Manual*).

After the convergence of the thermal analysis is achieved, the convergence progress of the successive iterations of the structural analysis will be displayed. This will carry on until convergence is achieved or the maximum number of iterations allowed per load stage that was defined in the 'Job Control' tab in the 'Job Definition' step (Section A.2.4.1 of the *VecTor3: User's Manual*) is reached. Then, a new load stage will start.

For future reference, all the data displayed in the MS-DOS window while the analysis is running are also saved in a text file with the title 'VECTOR.TXT'.

After the analysis is completed, the user can use the post-processor, Janus, for a graphical display of the results. This can be done by starting Janus by pressing the sixth button from the left (the last button) in the toolbar shown in Figure A-82, whose shape is shown in Figure A-83, then opening the 'VECTOR.JOB' file in Janus.



Figure A-83

VecTor3:

***B. Sample Coupled Thermal and
Structural Analysis***

Fady ElMohandes

Frank J. Vecchio

November 2013

B.1. Test Introduction

In this section, a step-by-step procedure for a sample of a coupled thermal and structural analysis using VecTor3 is presented. For this purpose, Column 1582, tested by the National Research Council Canada (NRC), has been chosen. This test, carried out by Mostafaei et al. (2012), is the first of its kind. It involved testing a full-scale column specimen for fire resistance assessment under both axial and lateral loading.

The column had a 305 mm square cross section, and a height of 3760 mm with two 25 mm thick steel loading plates at the top and the bottom. It was reinforced using four 25 mm diameter longitudinal steel reinforcing bars at the four corners with a clear cover of about 48 mm. The bars were tied using 10 mm diameter ties at a spacing of 305 mm. Figure B-1 shows the cross section of the column and Figure B-2 shows the reinforcement of the column in an isometric view.

The concrete was batched using calcareous aggregates and had a compressive strength of 55 MPa, based on three cylinder compression tests carried out on the day of testing. For the longitudinal steel reinforcing bars, the yield stress and the ultimate strength were reported as 444 MPa and 730 MPa, respectively. The yield stress and the ultimate strength of the ties were reported as 427 MPa and 671 MPa, respectively.

The column was loaded to an axial load of 1590 kN, applied from the bottom, prior to the start of fire. Rotation was restrained at both ends of the column. With the start of fire, the lateral displacement was applied to the top of the column following the profile shown in Figure B-3. The lateral displacement was kept at its maximum value of 50 mm (which was reached at 120 minutes from the start of fire) until the failure of the column.

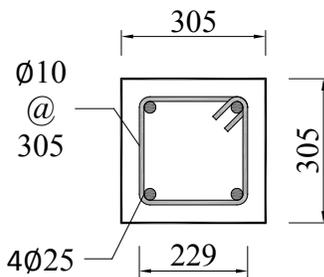


Figure B-1

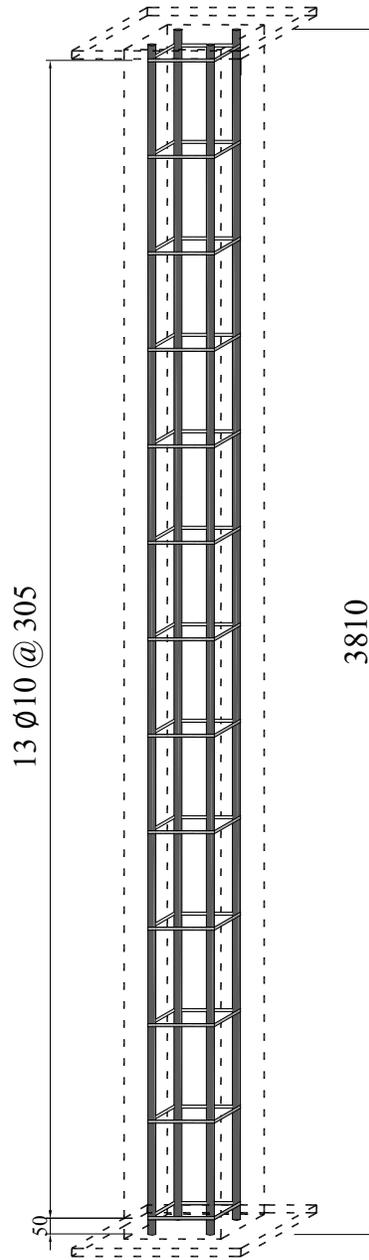


Figure B-2

The simulation of fire conditions was done in the NRC Column Furnace Facility, shown in Figure B-4, following the CAN/ULC S101-07 (2007) standard temperature-fire curve. The setup of the test required the column's top and bottom edges to be covered by insulation, thus only a 3175 mm length of the 3760 mm long column was subjected to fire. The mechanical and thermal loading setup of Column 1582 is shown in Figure B-5.

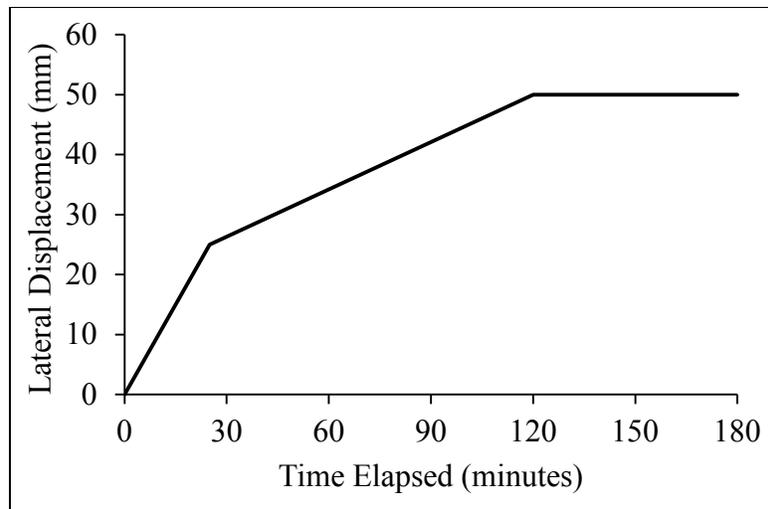


Figure B-3



Figure B-4

For the VecTor3 analysis, only half of the cross section needs to be modelled, taking advantage of the geometric and loading symmetry of the column. The finite element discretization, shown in Figure B-6, will be used to create the model, where the black dots mark the locations of the longitudinal reinforcing bars. Longitudinally, the column will be discretized into 75 elements with a height of 50.13 mm each, to achieve a total height of 3760 mm for the model. The 13 ties will be modelled at 300 mm spacings, instead of the actual 305 mm, with appropriate compensation in cross-sectional area, in

order to fit in the selected finite element discretization. This means that a tie will be located at every sixth element along the height.

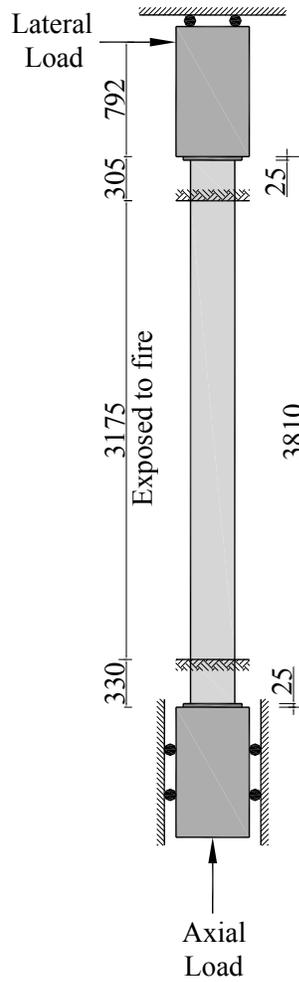
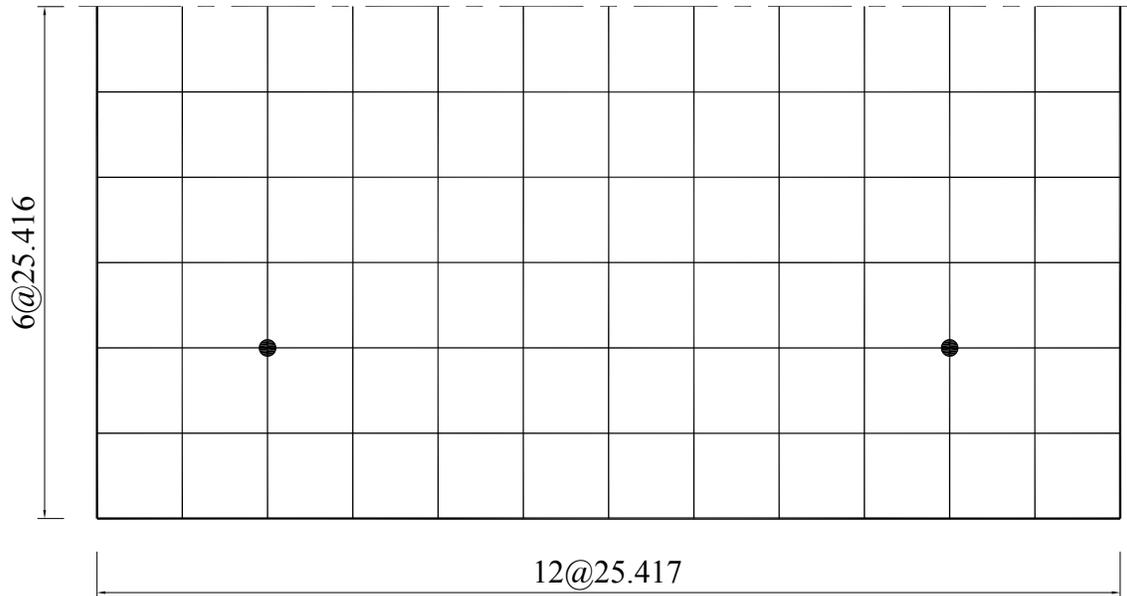


Figure B-5

**Figure B-6**

A time step of 60 seconds will be used for the analysis. The axial load of 1590 kN will be applied prior to the start of fire at the top of the column as nodal loads in a load control regime. Only half the load will be applied in order to achieve the same stress condition on the half model used in the analysis. The models provided by the current version of the Eurocode (EN 1992-1-2:2004, 2005) will be used for the thermal analysis. The exact same experimental lateral loading profile will be applied at the top of the column in a displacement control loading regime until failure. The thermal loading of the fire and the lateral loading will be started at the same time, resembling the experimental conditions.

B.2. Modelling Main Steps

In order to simplify the explanation of the process of creating the finite element model and running the analysis, the main steps explained in the *VecTor3: User's Manual* will be followed and in the same order as follows:

1 Starting-up

Formworks-Plus is started and the figure shown in Figure B-7 appears. When figure is clicked, the window shown in Figure B-8 appears. The second option ‘Solid (3-D)’ should be selected in order to use VecTor3.



Figure B-7

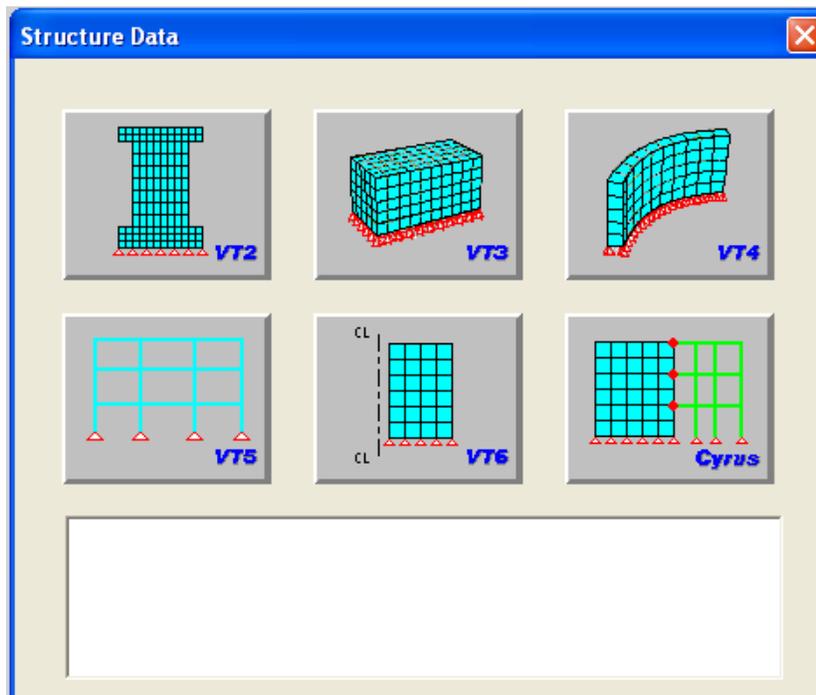


Figure B-8

B.2.1. Material Definition

B.2.1.1. Base Material Properties Definition

The button shown in Figure B-9 is pressed in order to define the material properties of concrete. Pressing the button opens the window shown in Figure B-10.



Figure B-9

Figure B-10

To specify a compressive strength of 55 MPa for the concrete of the column, ‘Reinforced Concrete’ is selected in the ‘Reference Type’ drop list at the top and a value of ‘55’ is inserted in the box titled ‘Cylinder Compressive Strength’, f'_c . If more properties of the concrete used in this column were known, their values should be inserted in their respective boxes in this window. After this is done, the ‘Add’ button is pressed and ‘Material 1’ appears in the list of ‘Material Types’ on the left-hand side of the window, as shown in Figure B-11. No smeared reinforcement will be used in the concrete material

used in this model; otherwise, this would be done through the part on the right-hand side of this window as explained in Section A.2.2.1 of the *VecTor3: User's Manual*. Only one material type is used for this model as only one type of concrete was used in the column. If more types were used, they would be added to the model in the same way. Once all the material types are added, the 'OK' button is pressed to close the window.

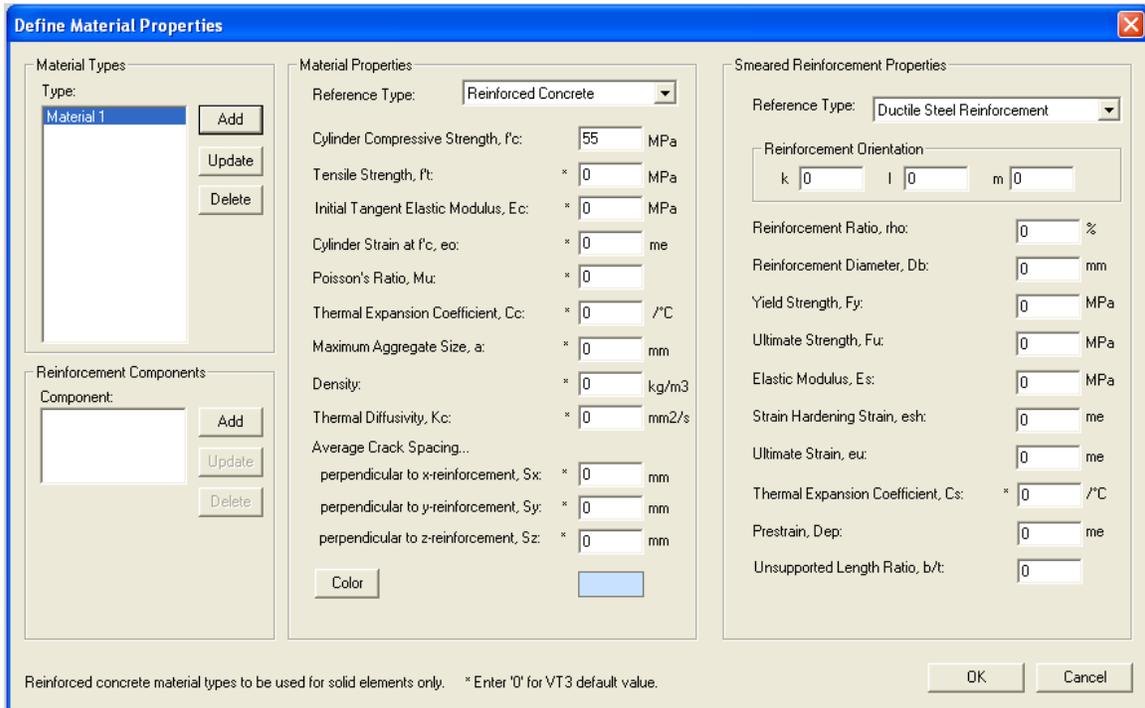


Figure B-11

B.2.1.2. Discrete Reinforcement Material Properties Definition

Using the button shown in Figure B-12, the properties of the discrete reinforcement materials can be added. Pressing the button opens the window shown in Figure B-13. Two types of discrete reinforcement material should be added in this model: one for the longitudinal bars and another for the ties.



Figure B-12

Define Reinforcement Properties

Reinforcement Type

Type:

Add

Update

Delete

Reinforcement Properties

Reference Type: Ductile Steel Reinforcement

Cross-Sectional Area: 0 mm²

Reinforcement Diameter, Db: 0 mm

Yield Strength, Fy: 0 MPa

Ultimate Strength, Fu: 0 MPa

Elastic Modulus, Es: 0 MPa

Strain Hardening Strain, esh: 0 me

Ultimate Strain, eu: 0 me

Thermal Expansion Coefficient, Cs: * 0 /°C

Prestrain, Dep: 0 me

Unsupported Length Ratio, b/t: 0

Centerline Offset, OS: 0 mm

Color

Reinforcement material types to be used for truss elements only.

OK Cancel

Figure B-13

For the longitudinal bars, ‘Ductile Steel Reinforcement’ is selected in the ‘Reference Type’ drop list at the top. Since the bars have a diameter of 25 mm, a value of 490 mm² and a value of 25 mm are inserted in the boxes titled ‘Cross-Sectional Area’ and ‘Reinforcement Diameter, d_b ’, respectively. Similarly, a value of 444 MPa and a value of 730 MPa are inserted in the boxes titled ‘Yield Stress, f_y ’ and ‘Ultimate Stress, f_u ’, respectively.

Since the actual stress-strain curve of the longitudinal reinforcing bars is not provided, default values will be used for the other material properties. In the ‘Elastic Modulus, E_s ’ box, a value of 200,000 MPa will be inserted. For the ‘Strain-Hardening Strain, ϵ_{sh} ’ and the ‘Ultimate Strain, ϵ_u ’, a value of 5 and a value of 200 will be used, respectively. The value entered for the ‘Thermal Expansion Coefficient, C_s ’, in this window will not be

used in the analysis as explained in the *VecTor3: User's Manual*. It will be overridden by a value that will be calculated according to a specific temperature-dependent model that will be chosen for the thermal expansion coefficient of concrete. For the 'Prestrain' and the 'Unsupported Length Ratio', they will both be assigned a value of zero.

Once all the properties are inserted in their respective boxes, the 'Add' button is pressed. 'Reinforcement 1' appears in the 'Reinforcement Type' list on the left-hand side of the window, as shown in Figure B-14.

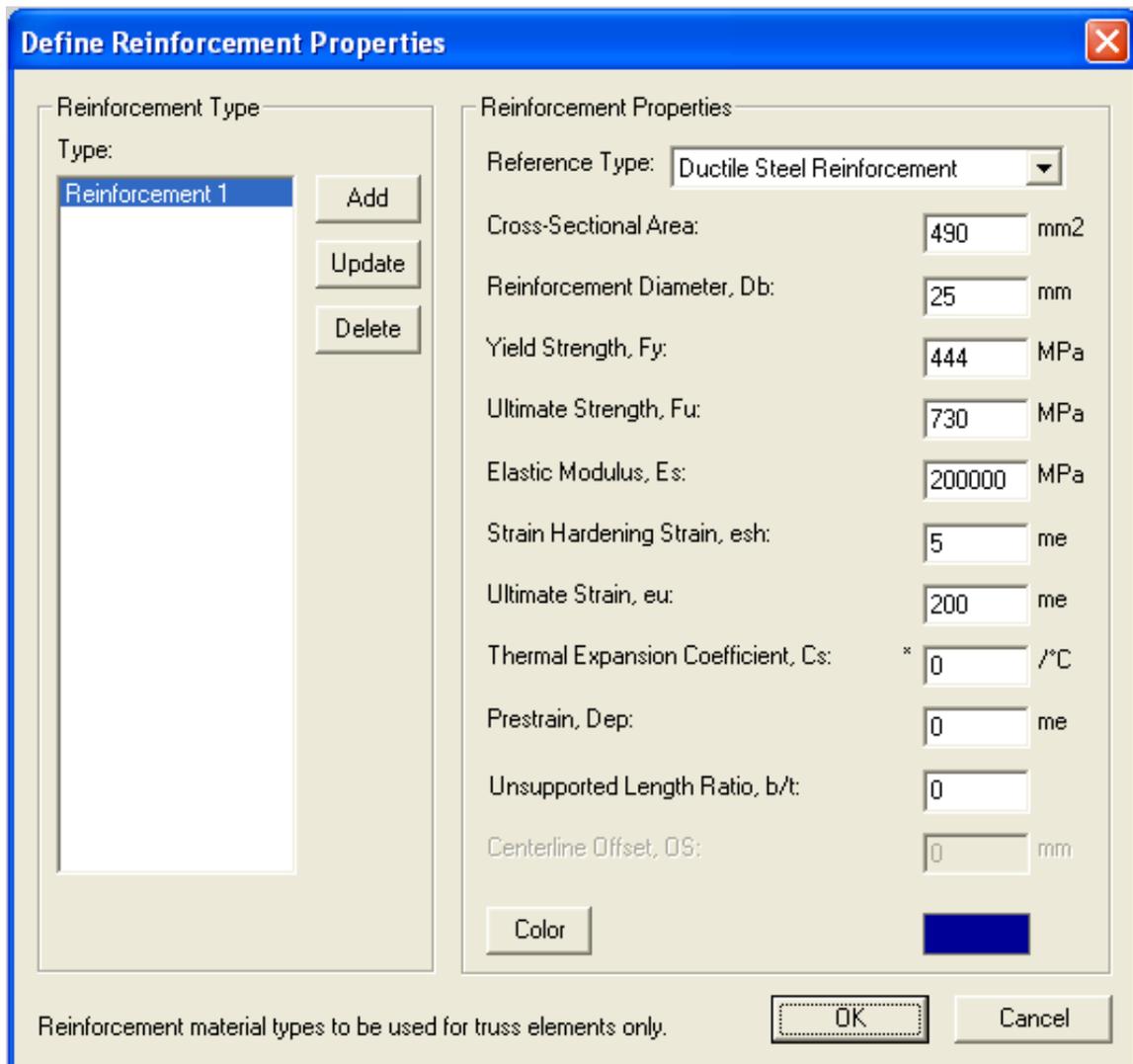


Figure B-14

For the ties, 'Ductile Steel Reinforcement' is selected in the 'Reference Type' drop list at the top. Since they have a diameter of 10 mm, a value of 78 mm² and a value of 10 mm

are inserted in the boxes titled ‘Cross-Sectional Area’ and ‘Reinforcement Diameter, d_b ’, respectively. Similarly, a value of 427 MPa and a value of 671 MPa are inserted in the boxes titled ‘Yield Stress, f_y ’ and ‘Ultimate Stress, f_u ’, respectively. For the other properties, same default values are used as explained for the longitudinal bars’ material type. Again, the ‘Add’ button is pressed and ‘Reinforcement 2’ appears in the ‘Reinforcement Type’ list on the left-hand side of the window, as shown in Figure B-15. Once all the material types are added, the ‘OK’ button is pressed to close the window.

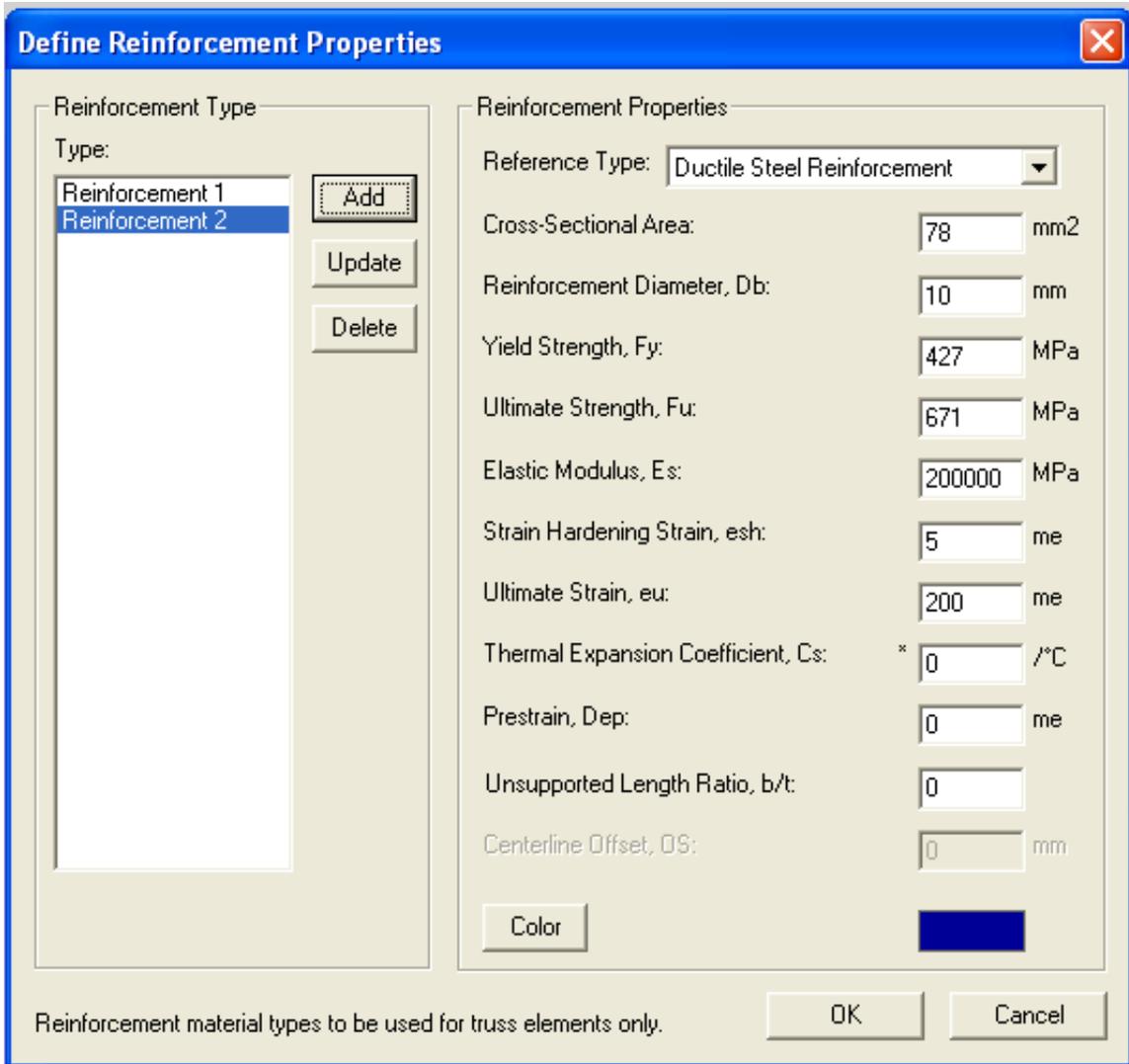


Figure B-15

B.2.1.3. Bond Properties Definition

Since, bond slip is not expected to be significant in this analysis, bond between concrete and discrete reinforcing bars will not be discretely modelled. Therefore, this step will be skipped.

B.2.2. Structure Definition

B.2.2.1. Node Definition

B.2.2.1.1. Defining Node Coordinates

For defining the coordinates of the nodes, the button shown in Figure B-16 is pressed, and the window shown in Figure B-17 opens.

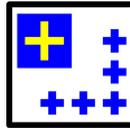


Figure B-16

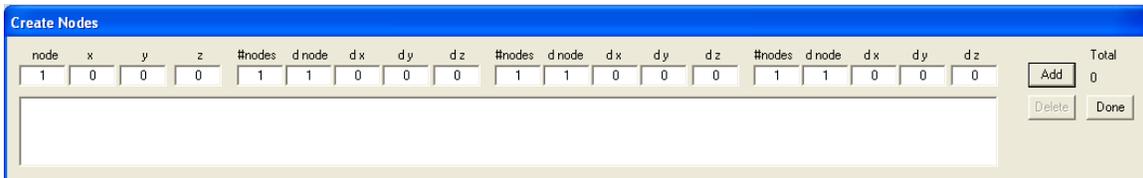


Figure B-17

For the finite element discretization suggested in Figure B-6, the values shown in the boxes in Figure B-18 should be added. The numbers of nodes are 13, 7, and 76 in the X-, Y-, and Z-direction, respectively. The coordinates of the first node are (0,0,0). The distances between the 13 nodes in the X-direction are 25.4167 mm, adding up to a total of 305 mm, with a numbering increment, d(node), of 1 from one node to the next. In the Y-direction, the 7 nodes will be 25.4167 mm apart, having a numbering increment, d(node), of 13 from one node to the next. The spacings between the nodes in that direction add up to a total of 152.5 mm, which is half the depth of the column. This means that a grid of

7×13 nodes will be created at this level, with a total of 91 nodes. Extrapolating those nodes in the Z-direction, 76 nodes will be created at distances of 50.1333 mm, with a numbering increment, d(node), of 91 from one node to the next.

Pressing the ‘Add’ button adds the nodes to the structure. The nodes can be seen in the X-Y plane in Figure B-19. More views can be obtained using the display features explained in Section A.2.1.1 of the *VecTor3: User’s Manual*.

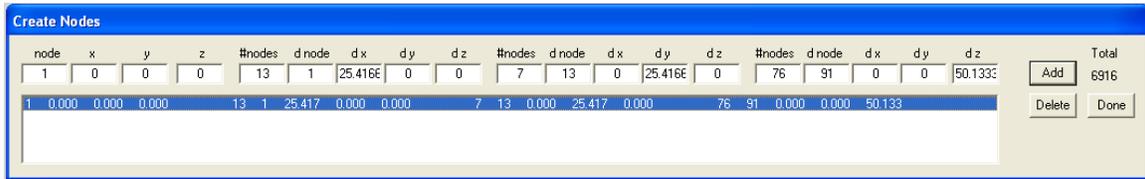


Figure B-18

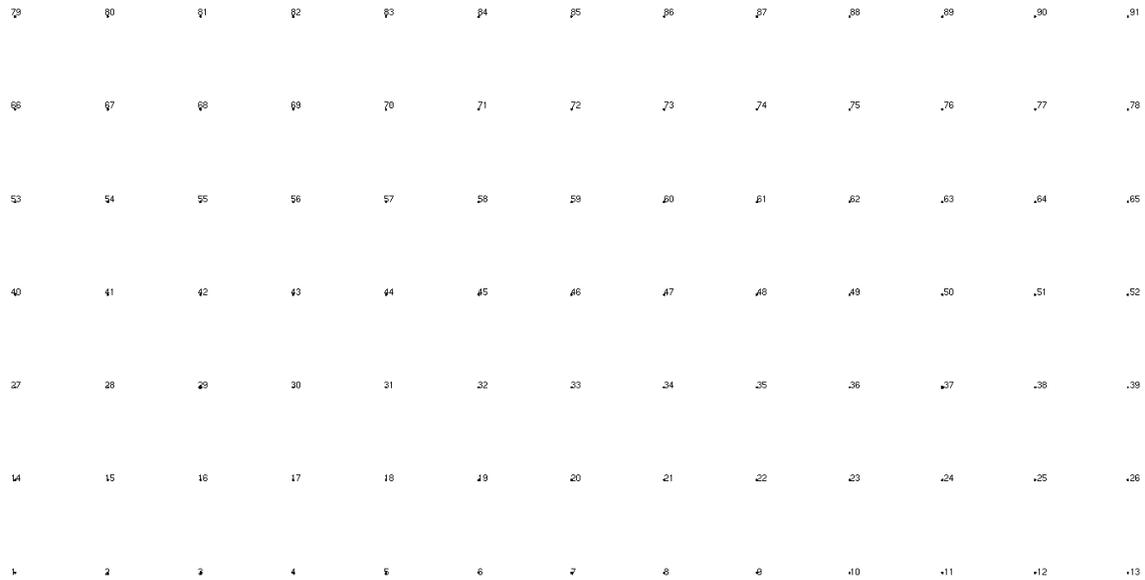


Figure B-19

B.2.2.1.2. Applying Node Restraints

Pressing the button shown in Figure B-20 opens the window shown in Figure B-21, where nodes’ restraints can be defined. In this model, all the nodes at the plane of the base of the column (at Z=0) should be fully restrained in all directions to resemble the experimental fully clamped condition.



Figure B-20

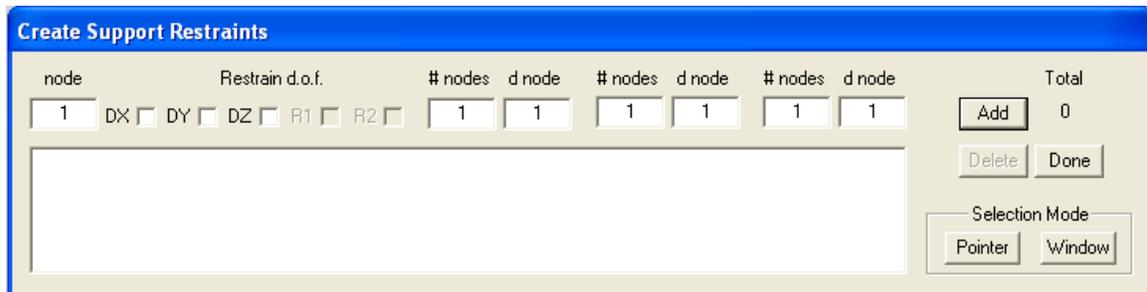


Figure B-21

Therefore, all the nodes at the plane of the base of the column will be restrained in the vertical direction (the Z-direction according to the selected orientation). However, in order to avoid additional stresses resulting from the expansion of the column at the plane of the base when the column is axially loaded, only the nodes lying on the axes of symmetry will be restrained in the X- and Y- directions. This means that, at the base of the column, the nodes lying on the axis of symmetry in the Y-direction at X=152.5 mm will be restrained in the X-direction and the nodes lying on the axis of symmetry in the X-direction at Y=152.5 mm will be restrained in the Y-direction.

Also, the plane of symmetry needs to be restrained in the perpendicular direction along the height of the column so that the response of the half model resembles the response of the full model.

Therefore, in order to achieve the aforementioned restraints, the lines shown in Figure B-22, Figure B-23, and Figure B-24 will be entered, pressing the 'Add' button after each line, then pressing the 'Done' button to close the window at the end. Figure B-25 shows the restraining conditions at the base of the model of the column (X-Y plane at Z=0).

Create Support Restraints

node	Restrain d.o.f.			# nodes	d node	# nodes	d node	# nodes	d node	Total	
1	<input type="checkbox"/> DX	<input type="checkbox"/> DY	<input type="checkbox"/> DZ	<input checked="" type="checkbox"/> R1	<input type="checkbox"/> R2	91	1	1	1	1	0

Selection Mode

Figure B-22

Create Support Restraints

node	Restrain d.o.f.			# nodes	d node	# nodes	d node	# nodes	d node	Total	
7	<input checked="" type="checkbox"/> DX	<input type="checkbox"/> DY	<input type="checkbox"/> DZ	<input type="checkbox"/> R1	<input type="checkbox"/> R2	7	13	1	1	1	0

Selection Mode

Figure B-23

Create Support Restraints

node	Restrain d.o.f.			# nodes	d node	# nodes	d node	# nodes	d node	Total	
79	<input type="checkbox"/> DX	<input type="checkbox"/> DY	<input checked="" type="checkbox"/> DZ	<input type="checkbox"/> R1	<input type="checkbox"/> R2	13	1	76	91	1	0

Selection Mode

Figure B-24



Figure B-25

B.2.2.1.3. Applying Node Constraints (Linking Nodes)

In the test, the rotation was restrained at both ends of the column. The restraints applied at the base of the model in the previous step (Section B.2.2.1.2 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*) achieve this rotation restraint for the bottom horizontal plane of the model. However, the top plane cannot be vertically restrained since the model will be axially loaded and should be free to deform along its height. Therefore, in order to achieve the rotation restraint for the top horizontal plane of the model, the nodes at this plane will have to be linked (constrained) to displace together vertically (in the Z-direction according to the selected orientation).

This will be done by pressing the button shown in Figure B-26, opening the window shown in Figure B-27.

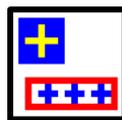


Figure B-26

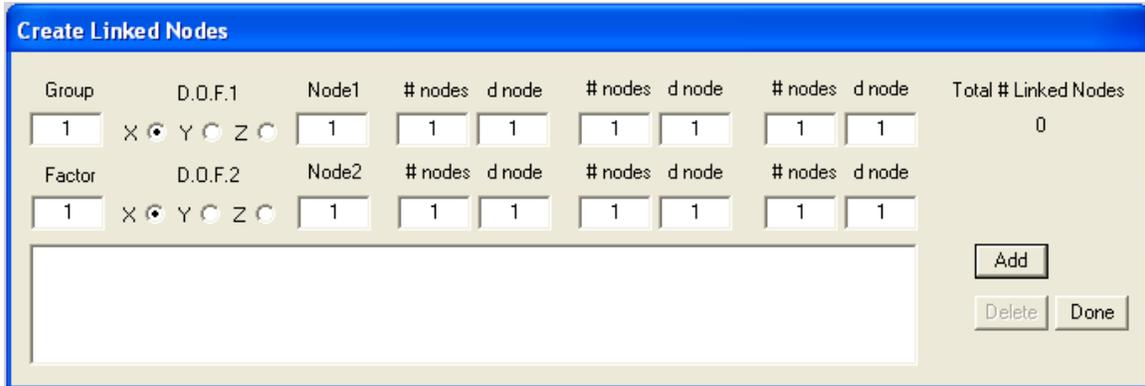


Figure B-27

The line shown in Figure B-28 will be entered, pressing the ‘Add’ button, then pressing the ‘Done’ button to close the window. A view of the top plane of the model (X-Y plane at Z=3760 mm) is shown in Figure B-29, where the constraints are shown as red dots. The restraints perpendicular to the axis of symmetry can also be seen in the figure.

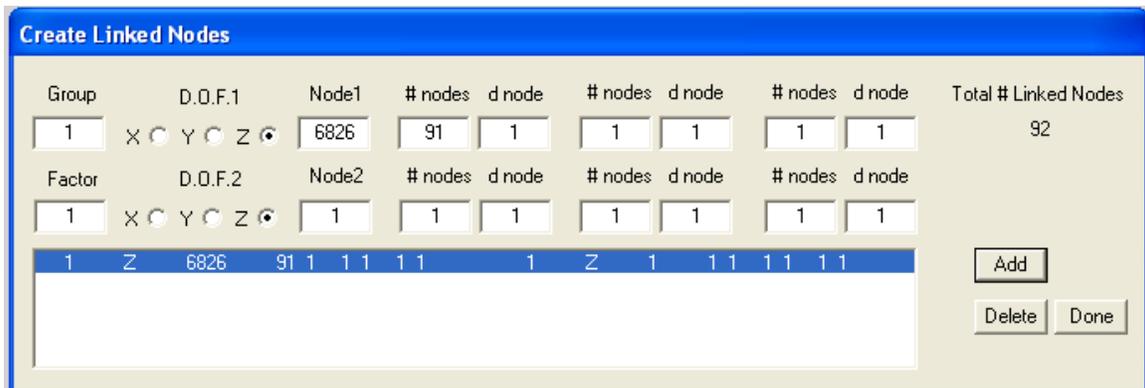


Figure B-28

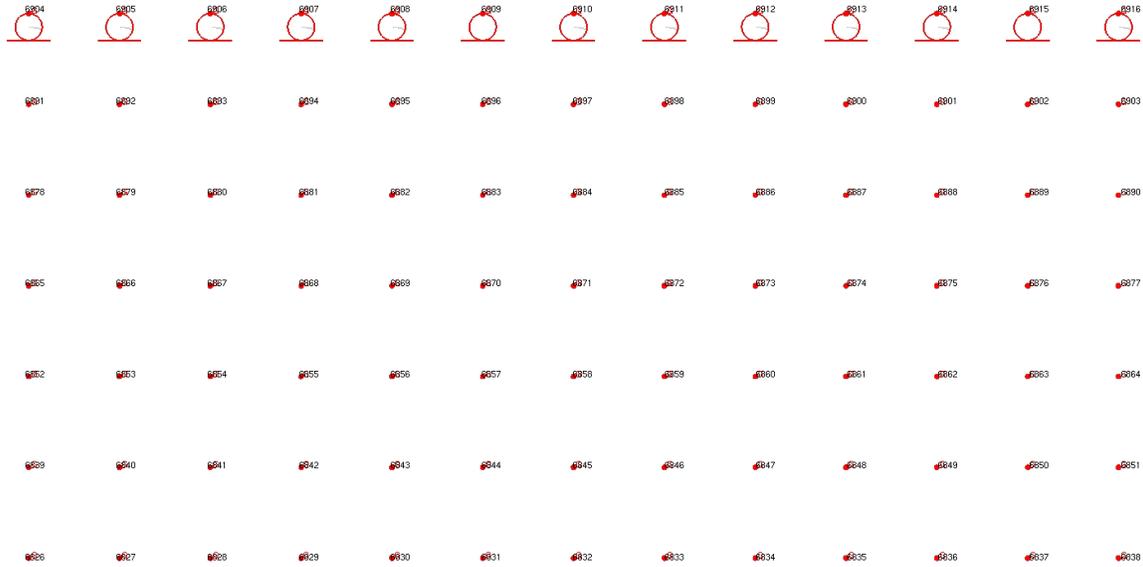


Figure B-29

B.2.2.2. Element Definition

In this test, the geometric nonlinearity effects have significant influence on the overall behaviour of the column as it is laterally loaded. Therefore, the eight-noded isoparametric hexahedral element type is chosen for creating the model to allow for considering those effects. To define the nodes comprising the elements, the button shown in Figure B-30 is pressed, which opens the window shown in Figure B-31.



Figure B-30

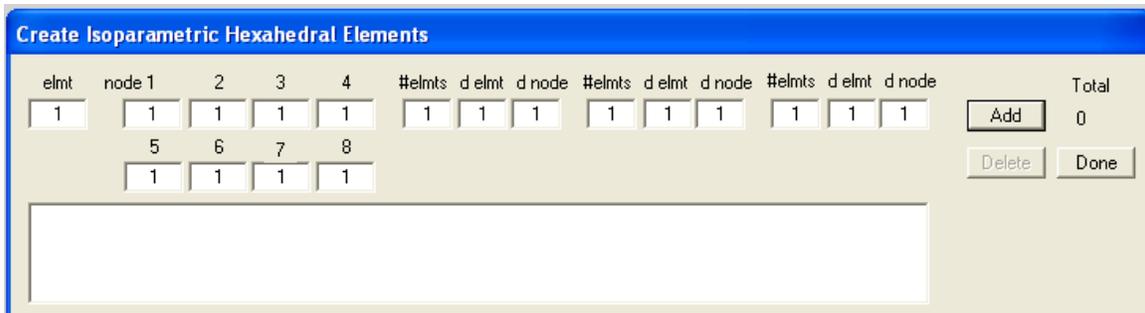


Figure B-31

Adding the line shown in Figure B-32 creates all the elements of the model, where 12 elements exist in the X-direction, 6 in the Y-direction, and 75 in the Z-direction. Returning to the nodes' numbering scheme defined in Section B.2.2.1.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*, the ID numbers of the nodes comprising the first element can be found. After all the elements are defined, pressing the 'Done' button closes the window. The three-dimensional view of the model with the created elements is shown in Figure B-33. More views can be obtained using the display features explained in Section A.2.1.1 of the *VecTor3: User's Manual*.

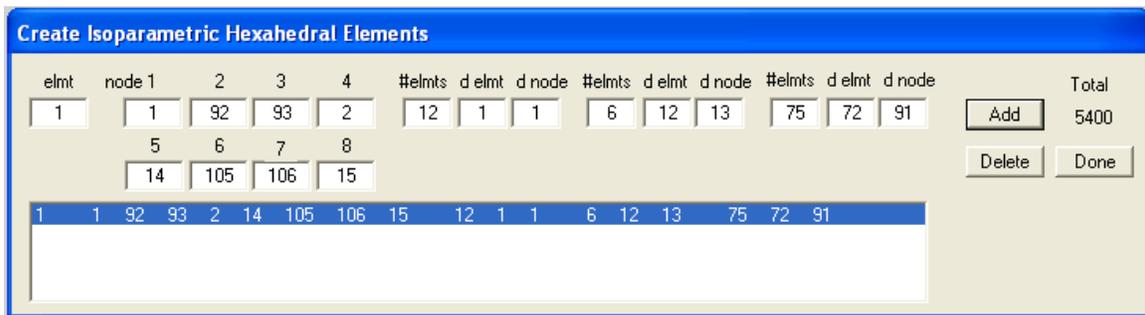


Figure B-32

The next step is to model the reinforcing bars using the two-noded uniaxial truss bar element type. This can be done using the button shown in Figure B-34, which opens the window shown in Figure B-35.

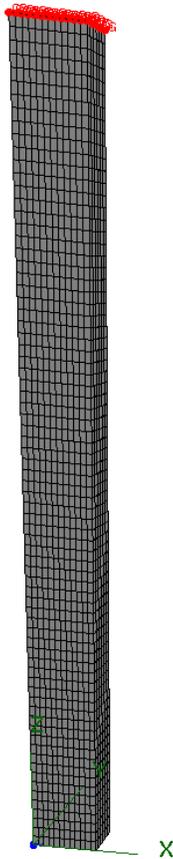


Figure B-33

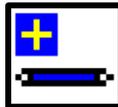


Figure B-34

Create Truss Elements													
elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total	
1	1	1	1	1	1	1	1	1	1	1	1	0	Add
												Delete	Done

Figure B-35

The line shown in Figure B-36 creates the longitudinal truss bar elements and the lines shown in Figure B-37 and Figure B-38 create the ties truss bar elements. When all elements are defined, pressing the 'Done' button closes the window. The view shown in

Figure B-39 shows the layout of the longitudinal reinforcement and the ties in the model. The coordinates of the nodes comprising the elements can be found using the various display features explained in Section A.2.1.1 of the *VecTor3: User's Manual*.

Create Truss Elements													
elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total	
5401	29	120	75	2	91	2	1	8	1	1	1	5758	
5401	29	120	75	2	91	2	1	8	1	1	1		
5551	120	121	8	1	1	13	16	546	1	1	1		
5559	120	133	4	2	13	2	1	8	13	16	546		

Figure B-36

Create Truss Elements													
elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total	
5551	120	121	8	1	1	13	16	546	1	1	1	5758	
5401	29	120	75	2	91	2	1	8	1	1	1		
5551	120	121	8	1	1	13	16	546	1	1	1		
5559	120	133	4	2	13	2	1	8	13	16	546		

Figure B-37

Create Truss Elements													
elmt	node 1	2	#elmts	d elmt	d node	#elmts	d elmt	d node	#elmts	d elmt	d node	Total	
5559	120	133	4	2	13	2	1	8	13	16	546	5758	
5401	29	120	75	2	91	2	1	8	1	1	1		
5551	120	121	8	1	1	13	16	546	1	1	1		
5559	120	133	4	2	13	2	1	8	13	16	546		

Figure B-38

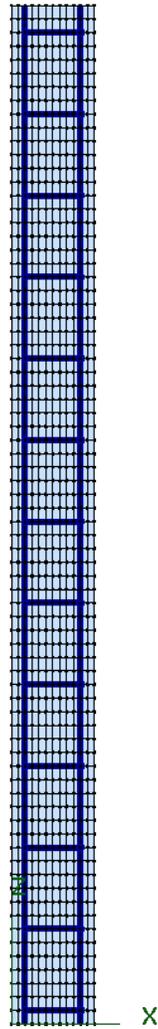


Figure B-39

B.2.2.3. Material Type Assignment

The created elements should be assigned specific materials from the material types that were previously defined. This is done by pressing the button shown in Figure B-40, opening the window shown in Figure B-41.

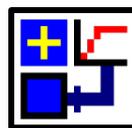


Figure B-40

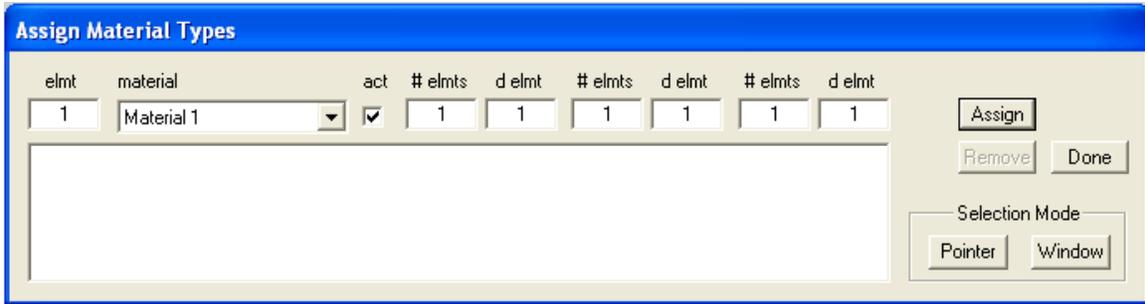


Figure B-41

The concrete elements are assigned ‘Material 1’ through the line shown in Figure B-42, the longitudinal bars are assigned ‘Reinforcement 1’ through the line shown in Figure B-43, and the ties are assigned ‘Reinforcement 2’ through the line shown in Figure B-44. When all the elements are assigned their respective materials, pressing the ‘Done’ button closes the window.

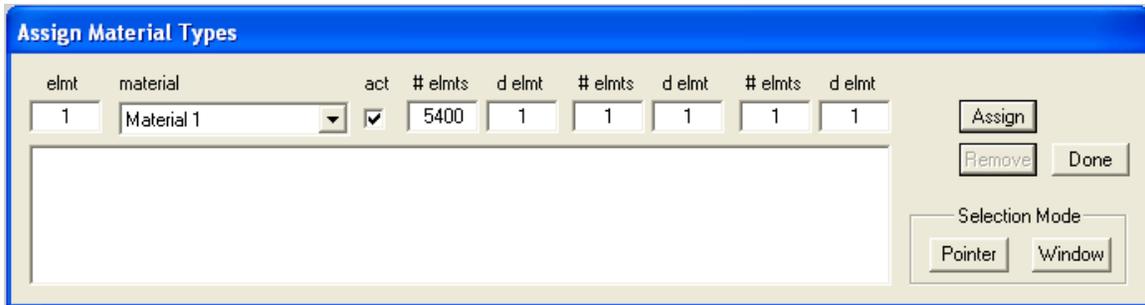


Figure B-42

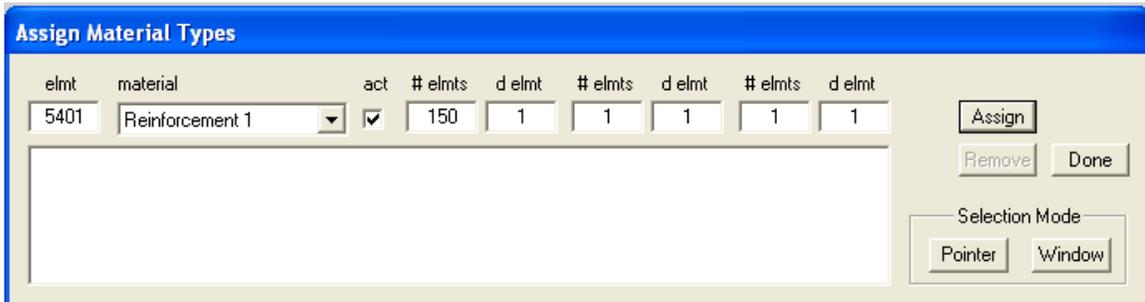


Figure B-43

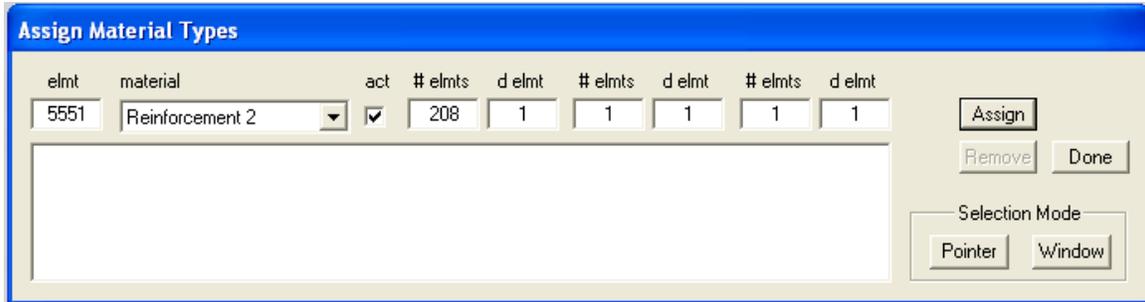


Figure B-44

B.2.3. Job Definition

This step is responsible for defining the analysis job scheme. Defining the job is done through the button shown in Figure B-45. Pressing this button opens the window shown in Figure B-46.



Figure B-45

Figure B-46

B.2.3.1. Analysis Job Control Parameters Definition

In the first tab in the ‘Job Definition’ window, titled ‘Job Control’, four main parts need to be defined, namely, ‘Job Data’, ‘Structure Data’, ‘Loading Data’, and ‘Analysis Parameters’.

In the collective frame titled ‘Job Data’, the job file name (the ‘JOB’ file) name should always be left as the default entry ‘VecTor’. The job title can be chosen as ‘Column 1582’, and the job date can be entered in its box for referencing purposes. In the collective frame titled ‘Structural Data’, the structure file name (the ‘S3R’ file) can be left unchanged as the default entry ‘Struct’, the structure title can be selected as ‘Column 1582’, and the structure type needs to be left as the default entry ‘Solid (3-D)’.

It should be noted that all the selected titles in those two collective frames are arbitrary and can be selected according to the user's preference, except for the job file name 'VecTor'.

In the third collective frame titled, 'Loading Data', the title of the output files generated by the analysis can be entered in the box titled 'Load series ID'. The default entry 'ID' can be left unchanged. In the box titled 'Starting load stage no.', the default value of '1' can be left unchanged as well. For the last box in this line, titled 'No. of load stages', a value of 500 will be selected. This value depends on the load increment applied in each load stage. Since the time increment will be selected as 60 seconds (1 minute) and the column is expected to fail in less than eight hours, 500 load stages should be enough for the analysis.

For defining the analysis loading scheme for the loading conditions that will be defined later, four loading cases will be defined. Technically, there are only three distinct loading cases involved in this analysis, namely, axial, lateral, and thermal loading. However, since the rate of the lateral loading changes through the test (after 25 minutes from the start of loading as shown in the lateral loading profile shown in Figure B-3), the loading case of the lateral loading will have to be divided into two separate loading cases, one for each loading rate. This means that a lateral loading case will be defined for the first 25 minutes (first 25 load stages) of the test and another one will be defined for the next 95 minutes. Then, they will both be left with constant values to achieve the constant lateral displacement of 50 mm that was reached at 120 minutes from the start of the test. Those constant values will be applied until the end of the analysis (the failure of the column). The final input is shown in Figure B-47.

The first loading case to be defined will be the thermal loading case simulating the fire conditions. 'Case1' box in the collective frame 'Loading Data' in the window shown in Figure B-46 is ticked in order to define the loading case. Arbitrarily, the 'Load file name' will be set to 'Thermal' and the 'Load case title' will be 'Fire Load'. Since this is a time-stepping analysis, the 'Initial factor' refers to the load factor throughout the analysis, on which no increment will be applied. Therefore, it will be set to '1'. The entry of the 'Final factor' will be discarded in the analysis, so it can be arbitrarily set to '1' as well. The

‘Inc. factor’ refers to the time step (time increment) to be used in the analysis (in seconds). This will be set to ‘60’, which means that the analysis will run at increments of one minute.

The ‘Load Type’ can be left as the default entry ‘Monotonic’. The ‘Repetitions’ will be set to ‘1’ as well. This is because, for thermal loading, the time step is not as small as that used for dynamic loading; so, there is no need to use time sub-steps. For more information on this part, refer to Section A.2.4.1 of the *VecTor3: User’s Manual*. For the ‘Cyclic Inc. factor’ entry, it will be discarded for this type of loading. Finally, the ‘Initial load stage’ will be set to ‘1’, meaning that the fire conditions will be applied starting from the first load stage (where the temperature is 20°C at the first load stage when the time elapsed is 0 seconds).

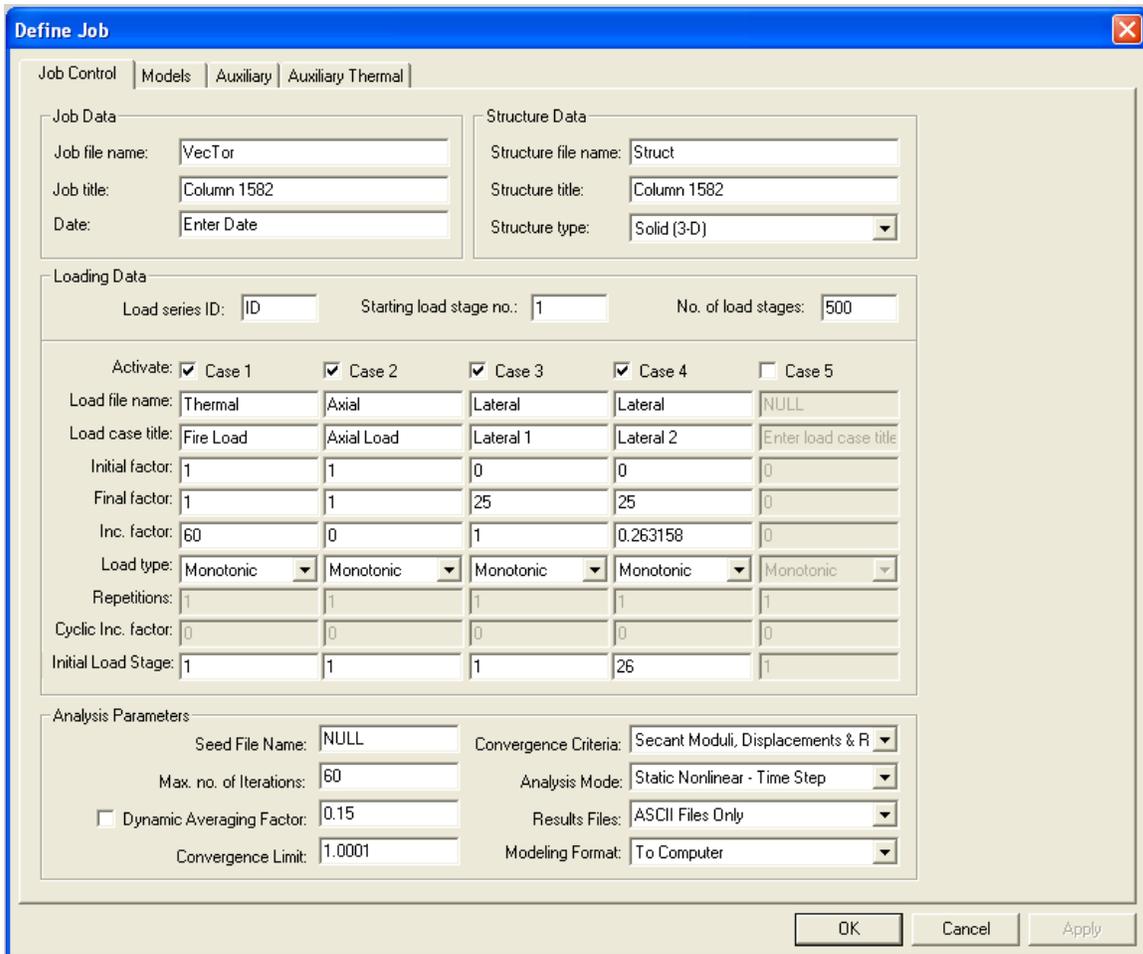


Figure B-47

The second loading case to be defined will be the axial loading case. The axial load will be applied in a load control regime to resemble the constant axial load applied during the test. The 'Case 2' box will be ticked to define it. The 'Load file name' can be arbitrarily set to 'Axial' and the 'Load case title' as 'Axial Load'. Since the axial load is kept constant throughout the test, both the 'Initial factor' and the 'Final factor' will be set to '1', with the 'Inc. factor' set to zero. The 'Load Type' should be set to 'Monotonic'. 'Repetitions' and 'Cyclic Inc. factor' entries will be discarded for this type of loading. Therefore, they can be left as their default entries of '1' and '0', respectively. Finally, the 'Initial load stage' will be set to '1', meaning that the axial load will be applied starting from the first load stage.

The third and fourth loading cases will be dedicated to the lateral loading. A displacement control regime will be used for the lateral loading to resemble the test conditions. The 'Case 3' and 'Case 4' boxes will be ticked. Since both loading cases involve the same type of loading and at the same nodes, they can both be defined by the same 'Load file name', which will be set to 'Lateral' as shown in Figure B-47. To distinguish between the two lateral loading cases, each will be assigned a different 'Load case title'. Arbitrarily, 'Case 3' will be assigned 'Lateral 1' and 'Case 4' will be assigned 'Lateral 2'.

Starting with 'Case 3', it will be assigned an 'Initial factor' of '0' and a 'Final factor' of '25', with an 'Inc. factor' of '1'. The increment factor ('Inc. factor') is selected based on the 60-second time step selected previously for the thermal loading case and a 1 mm displacement intended to be defined at the 'Load Definition' step later on (Section B.2.4). Returning to Figure B-3, it can be seen that the rate of lateral loading for the first 25 minutes of the test was 1 mm/min. Thus, at each load step of 60 seconds, an additional 1 mm of lateral loading should be applied.

The 'Load Type' should be set to 'Monotonic'. 'Repetitions' and 'Cyclic Inc. factor' entries will be discarded for this type of loading. Therefore, they can be left as their default entries of '1' and '0', respectively. Finally, the 'Initial load stage' will be set to '1', meaning that this lateral loading case will be applied starting from the first load stage.

The next 25 minutes of the test with the different rate of lateral loading will be defined in 'Case 4'. This explains why the 'Initial load stage' will be set to '26'. This is because at

load stage #1, the load factor of the lateral loading case (Case 3), will be 0, increasing up to a value of 25 at load stage #26. Therefore, at Load Stage #26, the new lateral loading case (Case 4) should start with a load factor of 0 ('Initial factor' of '0'), which, in addition to the load factor of 25 of 'Case 3', adds up to a total of 25 mm. The load factor of 'Case 4' should then increase to a value of 25 ('Final factor' of '25') within the next 95 load stages (95 minutes after the first 25 minutes to an elapsed time of 120 minutes from the start of the test) to simulate the lateral loading profile shown in Figure B-3). This means that the 'Inc. factor' for 'Case 4' should be set to a value of $25 / 95$, which is equal to '0.263158' (mm/min), as can be seen in Figure B-47. Again, the 'Load Type' should be set to 'Monotonic'; and the default entries of '1' and '0' should be left for the 'Repetitions' and the 'Cyclic Inc. factor', respectively, as they will be discarded for this type of loading.

In the fourth collective frame titled, 'Analysis Parameters', the 'Seed file name' should be left as the default entry 'NULL' as the analysis required to be run is a new analysis. The 'Max. no. of iterations' can also be left as the default entry '60', although a stable and accurate analysis can be achieved with a smaller number of iterations. The 'Averaging factor' should be set to a small value as in compression members, such as columns, a small averaging factor is required for a stable analysis. In this analysis, an averaging factor of '0.15' will be chosen. The 'Convergence limit' criterion can also be slackened to a higher value than the default value of '1.00001'. This is because, for thermal analysis, achieving such a strict convergence factor can be difficult. In this analysis, a value of '1.0001' will be used.

For the 'Convergence criteria', the option 'Secant Moduli, Displacements, & Reactions - Weighted Average' will be chosen to ensure reasonable convergence. The 'Analysis mode' for this type of analysis is 'Static Nonlinear - Time Step' as it involves a static (not dynamic) analysis in a time-stepping procedure. For the 'Results files', the 'ASCII files only' option can be used. There is no intention to start an analysis from a seed file generated from this analysis; hence, there is no need to generate binary files.

Finally, the entry in the box titled 'Modelling format' should be left to the default entry 'Stand Alone Modelling', since the entire structural model will be generated using the element types of VecTor3 only.

This concludes the entries required in the 'Job Control' tab in the 'Job Definition' window.

B.2.3.2. Structural Analysis Models Definition

The next tab in the 'Job Definition' window, titled 'Models', is shown in Figure B-48 with the default entries. This tab allows the user to select the models of the advanced structural mechanisms to be followed in the structural analysis for concrete, steel reinforcing bars, and bond between concrete and steel reinforcing bars. This is in addition to several general analysis models.

The screenshot shows the 'Define Job' dialog box with the 'Models' tab selected. The dialog is divided into several sections:

- Concrete Models:**
 - Compression Pre-Peak: Parabola - Hognestad
 - Compression Post-Peak: Modified Park-Kent
 - Compression Softening: Vecchio 1992-A (e1/e2-Form)
 - Tension Stiffening: Modified Bentz 2003
 - Tension Softening: Linear
 - FRC Tension: SDEM - Monotonic
 - Confined Strength: Kupfer / Richart
 - Dilation: Variable - Kupfer
 - Cracking Criterion: Mohr-Coulomb (Stress)
 - Crack Stress Calc: Basic (DSFM/MCFT)
 - Crack Width Check: Agg/2.5 Max Crack Width
 - Crack Slip Calc: Walraven (Monotonic)
 - Creep and Relaxation: Not Available
 - Hysteretic Response: Nonlinear w/ Plastic Offsets
- Reinforcement Models:**
 - Hysteretic Response: Bauschinger Effect (Seckin)
 - Dowel Action: Tassios (Crack Slip)
 - Buckling: Akkaya 2012 (Refined Dhakal-Ma)
- Bond Models:**
 - Concrete Bond: Elgehausen
- Analysis Models:**
 - Strain History: Previous Loading Considered
 - Strain Rate Effects (Conc/Steel): n/c : n/c
 - Structural Damping: Not Considered
 - Geometric Nonlinearity: Considered
 - Crack Process: Uniform

At the bottom right, there is a 'Reset Options' section with 'Basic' and 'Advanced' buttons. At the bottom of the dialog are 'OK', 'Cancel', and 'Apply' buttons.

Figure B-48

Figure B-49 shows the optimum options for the current analysis as follows:

Concrete Models:

Compression Pre-Peak	HSC - Popovics
Compression Post-Peak	Modified Park-Kent
Compression Softening	Vecchio 1992-A (e1/e2-Form)
Tension Stiffening	Modified Bentz 2003
Tension Softening	Linear
FRC Tension	SDEM - Monotonic
Confined Strength	Kupfer - Richart
Dilatation	Variable - Kupfer
Cracking Criterion	Mohr - Coulomb (Stress)
Crack Stress Calc	Basic (DSFM/MCFT)
Crack Width Check	Agg/2.5 Max Crack Width
Crack Slip Calc	Walraven (Monotonic)
Concrete Creep and Relaxation	Not Available
Concrete Hysteretic Response	Linear w/ No Plastic Offsets

Reinforcement Models:

Reinforcement Hysteretic Response	Elastic-Hardening (Curvilinear)
Reinforcement Dowel Action	Tassios (Crack Slip)
Reinforcement Buckling	Akkaya 2012 (Refined Dhakal-Maekawa)

Bond Models:

Concrete Bond	Eligehausen
---------------	-------------

Analysis Models:

Element Strain Histories	Previous Loading Considered
Strain Rate Effects:	n/c ; n/c
Structural Damping	Not Considered
Geometric Nonlinearity	Considered
Uniform	Uniform

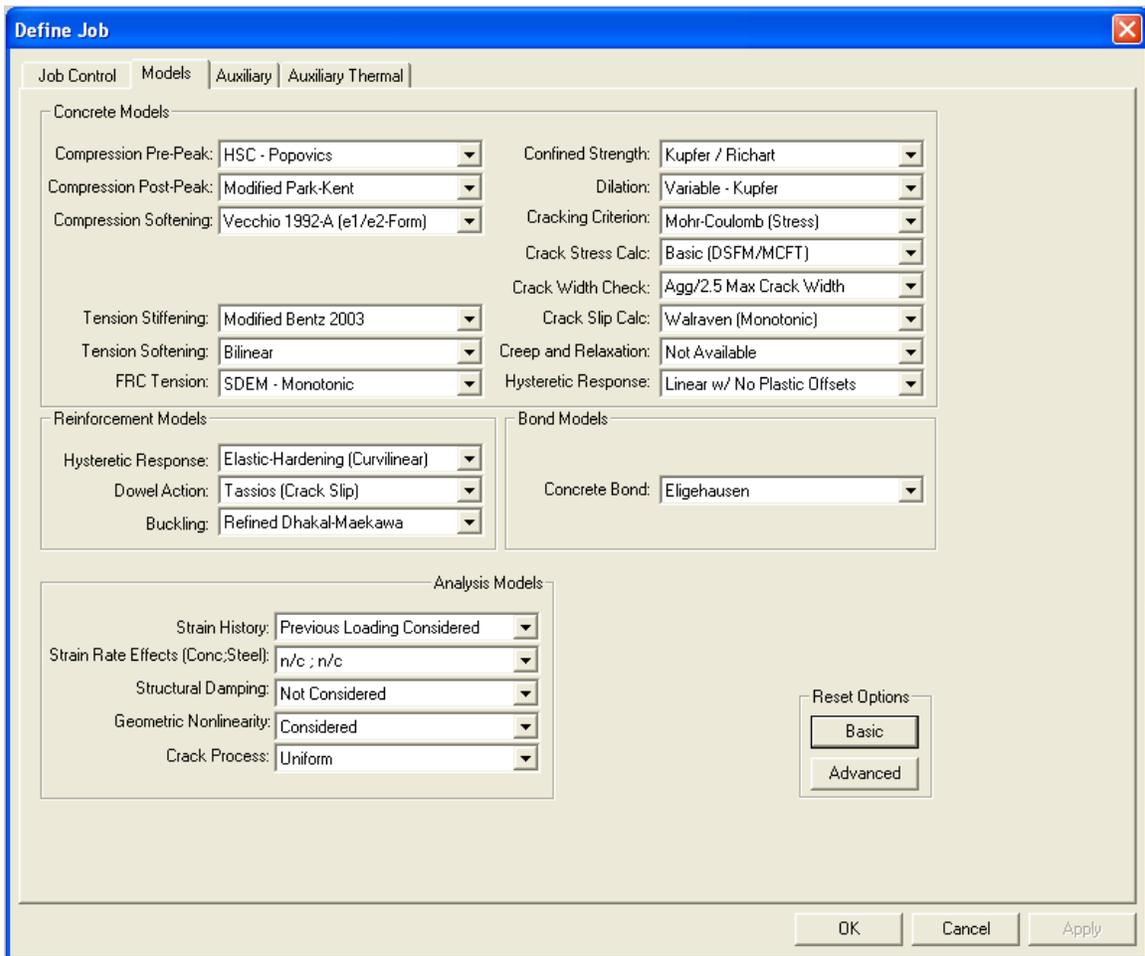


Figure B-49

While most of the models selected are the default ones, some were changed to better suit the current structure and analysis conditions. Since the concrete used had a compressive strength of 55 MPa, it is more appropriate to use the model of high-strength concrete ‘HSC - Popovics’ for the ‘Compression Pre-Peak’ model. Since no reverse loading conditions are applied in the analysis, the concrete ‘Hysteretic Response’ can be changed

to ‘Linear w/ No Plastic Offsets’. While this is not necessary for the accuracy of the analysis, it leads to a more stable analysis. Therefore, whenever the nonlinear response and the plastic offsets are not expected to have a major influence on any analysis, it may be a good idea not to use them. This also applies to the reinforcement ‘Hysteretic Response’ which is recommended to be set to ‘Elastic-Hardening (Curvilinear)’, neglecting the Bauschinger effect.

B.2.3.3. Auxiliary Parameters Definition

The next tab in the ‘Job Definition’ window, titled ‘Auxiliary’, is shown in Figure B-50. In the ‘General’ collective frame, the default ‘Stiffness Matrix Solver Type’ should be left as the default entry ‘Solver 2’ as this solver is much faster than ‘Solver 1’.

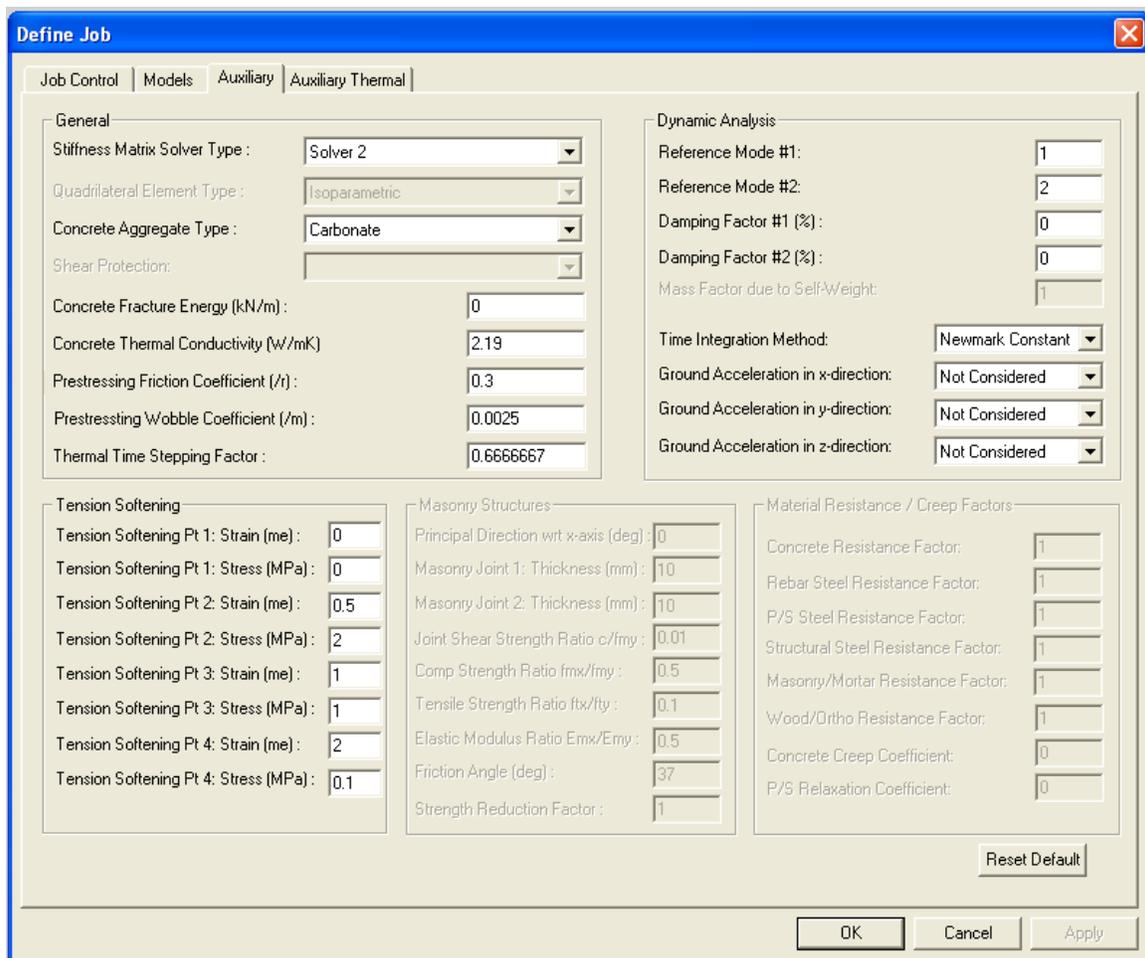


Figure B-50

The input entered for the properties of concrete related to the thermal analysis will be overridden by the input entered in the ‘Auxiliary Thermal’ tab. Therefore, the default entries can be left unchanged. This applies to the ‘Concrete Aggregate Type’, ‘Concrete Thermal Conductivity’, and ‘Thermal Time Stepping Factor’.

All the other entries in the ‘General’, ‘Dynamic Analysis’, and ‘Tension Softening’ collective frames do not relate to this analysis; hence, they can be left unchanged.

B.2.3.4. Auxiliary Thermal Parameters Definition

The next tab in the ‘Job Definition’, titled ‘Auxiliary Thermal’, is shown in Figure B-51. This tab allows the user to define several additional parameters that are required for the coupled thermal and structural analysis. The default entries are set to the models and criteria that have proven reasonable agreement with experimental results based on the experience of the developers of VecTor3.

The screenshot shows the 'Define Job' dialog box with the 'Auxiliary Thermal' tab selected. The dialog is organized into four main sections:

- Concrete Thermal Properties:** Includes fields for Concrete Aggregate Type (Carbonate), Convective Heat Transfer Coeff. (25), Emissivity (0.7), Initial Density (2400), Initial Moisture Content (4), and several dropdown menus for Density, Conductivity, Specific Heat Capacity, Compressive Strength, Peak Strain, Modulus of Elasticity, Tensile Strength, and Thermal Expansion Strain models.
- Concrete Moisture Transfer Properties:** Includes Initial Porosity (8), Porosity Model (Gawin et al. (1999)), and Permeability Model (Tenchev et al. (2001)).
- Steel Rebar Properties:** Includes dropdown menus for Yield Stress, Ultimate Stress, Young's Modulus, and Thermal Expansion Strain models.
- Thermal Solution Parameters:** Includes Thermal Time Stepping Factor (0.6666).

At the bottom right, there are buttons for 'Reset Default', 'OK', 'Cancel', and 'Apply'. A legend at the bottom indicates '1 : Pre-Cooling 2: Post-Cooling'.

Figure B-51

The 'Concrete Aggregate Type' is set to 'carbonate' as the aggregate used in the concrete mix for this column was calcareous. The default values for the 'Emissivity', 'Initial Density', and 'Initial Moisture Content' are left unchanged as 0.7, 2400 kg/m³, and 4%, respectively.

The default entries chosen are as follows:

For concrete:

Density	The Eurocode (EN 1992-1-2:2004, 2005)
Thermal conductivity	The Eurocode (EN 1992-1-2:2004, 2005)
Specific heat capacity	The Eurocode (EN 1992-1-2:2004, 2005)
Porosity	Gawin et al. (1999)
Permeability	Tenchev et al. (2001)
Compressive strength at elevated temperatures	The Eurocode (EN 1992-1-2:2004, 2005)
Residual compressive strength	Chang et al. (2006) - Model 1
Strain corresponding to peak compressive stress	The Eurocode (EN 1992-1-2:2004, 2005)
Residual strain corresponding to peak compressive stress	Chang et al. (2006)
Initial modulus of elasticity	Stress factor / corr. strain factor
Residual initial modulus of elasticity strength	Chang et al. (2006) - Model 1
Tensile strength	The Eurocode (EN 1992-1-2:2004, 2005)
Residual tensile strength	Chang et al. (2006)
Thermal expansion strain	The Eurocode (EN 1992-1-2:2004, 2005)

For reinforcement:

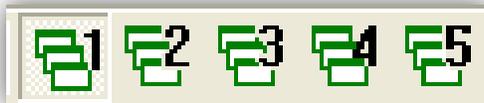
Yield stress	The Eurocode (EN 1992-1-2:2004, 2005)
Ultimate stress	The Eurocode (EN 1992-1-2:2004, 2005)
Young's modulus	The Eurocode (EN 1992-1-2:2004, 2005)
Thermal expansion strain	The Eurocode (EN 1992-1-2:2004, 2005)

B.2.4. Load Definition

The loads applied to the structural model to be analyzed are to be added at this point. After the loading cases are defined in the 'Job Definition' step, the load input toolbar, shown in Figure B-52, becomes available for user input.

**Figure B-52**

The second to sixth buttons from the left in the toolbar shown in Figure B-52, which can be seen in Figure B-53, are used to define the various loads to be included in the four loading cases defined in the 'Job Definition' step.

**Figure B-53**

Recalling the loading cases defined in the 'Analysis Job Control Parameters' in the 'Job Definition' step in Section B.2.3.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*, the first loading case (Case 1) to be defined is the thermal loading case simulating the fire conditions. For this, the first button from the left in the toolbar

shown in Figure B-53 with the number ‘1’ is pressed. With that button pressed down, all the loads to be defined will be included in ‘Case 1’.

Recalling Section A.2.5.7 of the *VecTor3: User’s Manual* which is responsible for defining this type of loading (Nodal Thermal Loads), the parameters required for defining fire conditions can be found. Pressing the button shown in Figure B-54 opens the window shown in Figure B-55 which is responsible for the nodal thermal loads definition.



Figure B-54

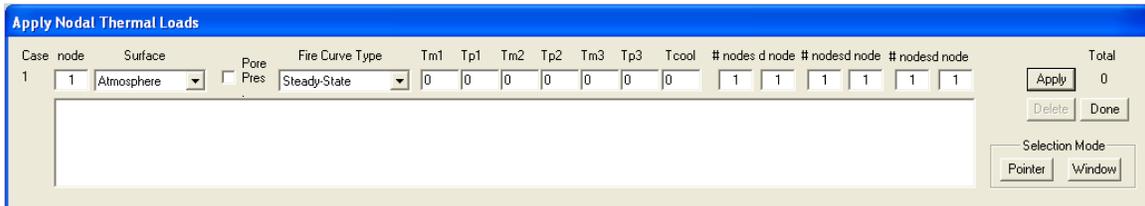


Figure B-55

It can be noticed that the loading case number (Case 1) is shown at the left-hand side of the window for referencing purposes. Returning to Figure B-5, it can be seen that the part of the column exposed to fire starts at an elevation of 330 mm from the base of the column and ends at 305 mm from the top. Also, the fire should be applied on the nodes on the outer surface of the column only. Referring to the nodes’ numbering scheme defined in Section B.2.2.1.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*, the first node at the level where fire is applied is node # 547 which lies at an elevation of 300.8 mm. The lines required to define the fire conditions for the entire fire-exposed surface are shown in Figure B-56 and Figure B-57.

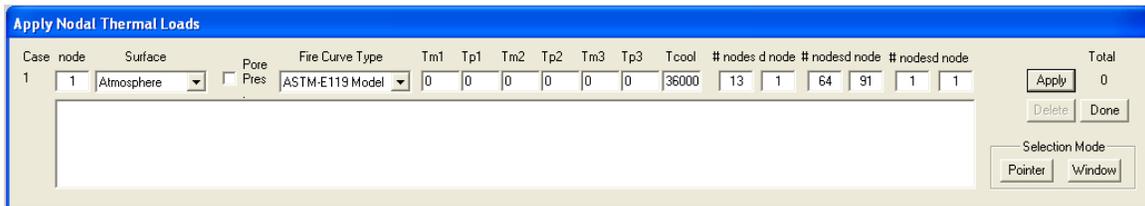


Figure B-56

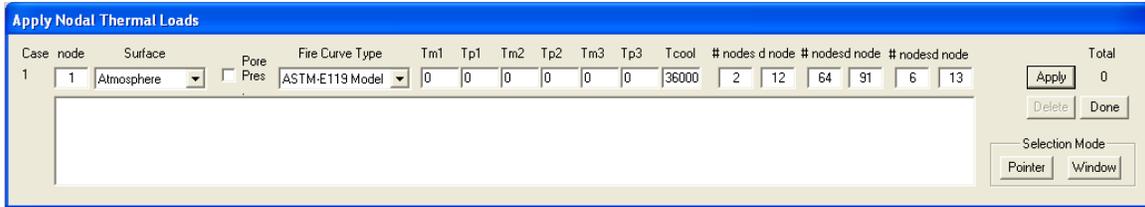


Figure B-57

Figure B-58 shows the nodes subjected to fire along the height of the column marked in red and Figure B-59 shows a cross section with the nodes subjected to fire around the surface also marked in red.

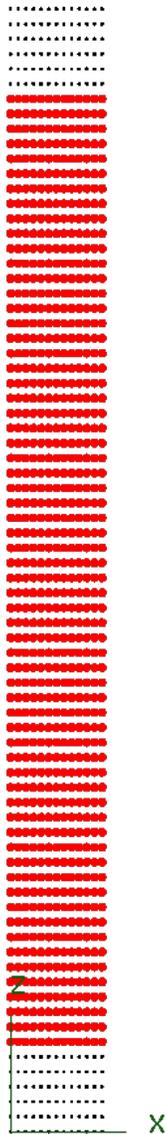


Figure B-58

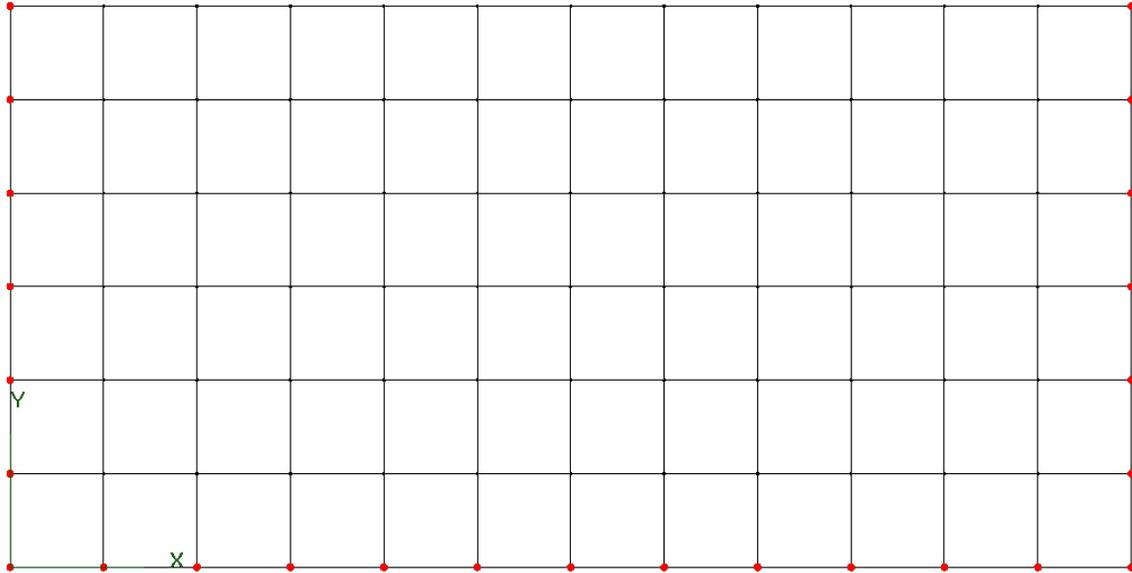


Figure B-59

The 'Surface' drop list should be left as the default entry 'Atmosphere', referring to the fact that the applied temperatures are the temperatures of the fire, not the actual temperatures of the nodes on which the fire conditions are applied (the 'Element Surface' option). The checkbox titled 'Pore Pres' should be left unticked here as coupled heat and moisture transfer analysis is not required. For the 'Fire Curve Type', the 'ASTM E119 – 12a (2012) model' should be selected as it is similar to the CAN/ULC S101-07 (2007) standard temperature-fire curve applied during the test. 'Tm1', 'Tp1', 'Tm2', 'Tp2', 'Tm3', and 'Tp3' are used to define custom temperature-time curve; therefore, they will be discarded for the fire curve type selected. 'Tcool', which denotes the time at which the fire is extinguished', will be assigned a large value as the fire was applied until failure. A value of 36000 seconds (10 hours) will be entered as the column is expected to fail a lot sooner than that. Pressing the 'Done' button closes the window when the loads' definition is completed.

The next loading case (Case 2) to be defined is the axial loading case. The axial load is to be applied in a load control regime at the top of the column resembling the test conditions. To define this loading case, the second button from the left in the toolbar shown in Figure B-53 with the number '2' is pressed. With that button pressed down, all the loads to be defined will be included in 'Case 2'.

Recalling Section A.2.5.1 of the *VecTor3: User's Manual* which is responsible for defining this type of loading (Nodal Loads), the parameters required for defining this type of load can be found. Pressing the button shown in Figure B-60 opens the window shown in Figure B-61 which is responsible for defining nodal loads.

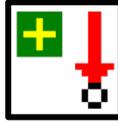


Figure B-60

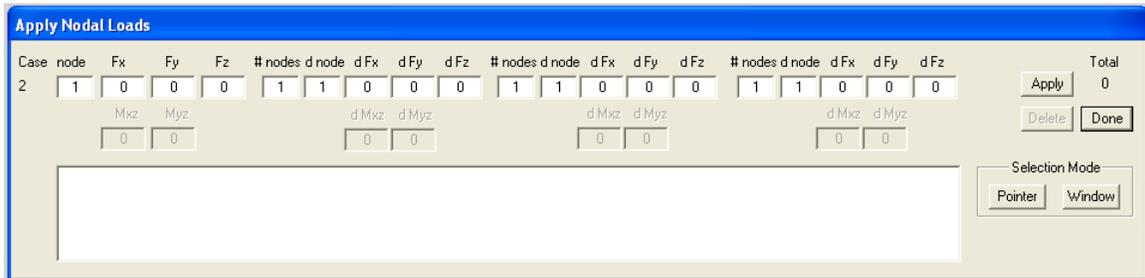


Figure B-61

To ensure uniform loading at the top of the column (besides having already linked the nodes at this level to displace together vertically in the Z-direction), the loads are applied as concentrated loads distributed among the nodes at this level based on the tributary area. The summation of the loads should add up to 795 kN, which is half the axial load applied on the column, since only half the column is modelled here. Based on a tributary area of 161.5 mm² for the corner nodes, 323 mm² for the edge nodes, and 646 mm² for the middle nodes, the loads applied on those nodes are 2.76 kN, 5.52 kN, and 11.04 kN, respectively. The ID numbers of the nodes can be found from the nodes' numbering scheme defined in Section B.2.2.1.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*. A view of the ID numbers of the nodes at the level of the application of the axial load (X-Y view at Z=3760 mm) is shown in Figure B-62.

VecTor3: Sample Coupled Thermal and Structural Analysis

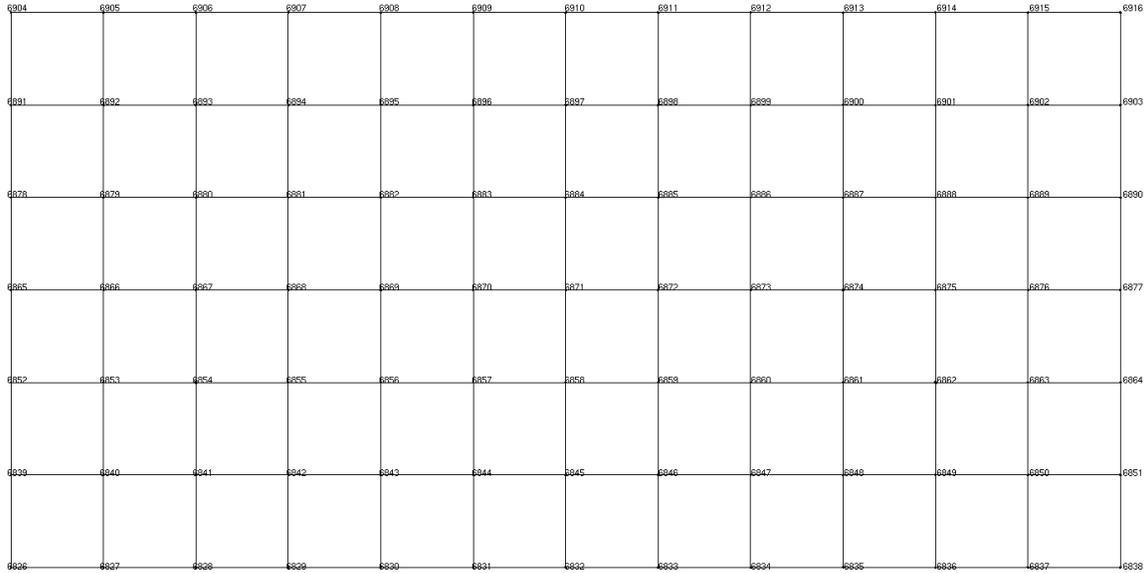


Figure B-62

The line shown in Figure B-63 defines the loads at the corner points, the lines shown in Figure B-64 and Figure B-65 define the loads at the edge points, and the line shown in Figure B-66 defines the loads at the middle points. It can be noticed that the loading case number (Case 2) is shown at the left-hand side of the window for referencing purposes. The negative sign of the applied loads is to apply the loads vertically downwards (negative Z-direction). Figure B-67 shows a section of the column along its height showing the applied axial loads at the top of the column.

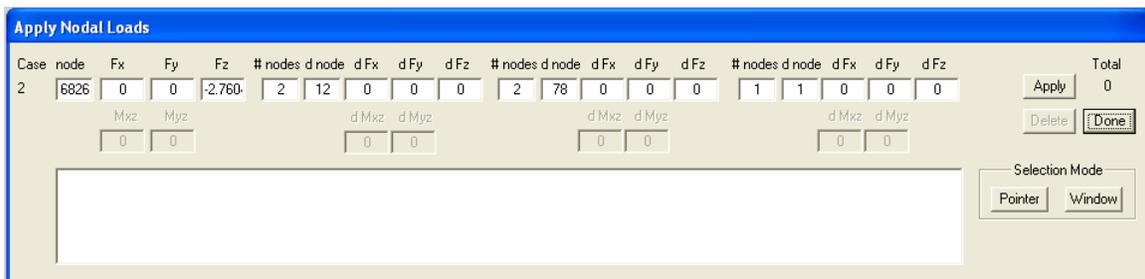


Figure B-63

VecTor3: Sample Coupled Thermal and Structural Analysis

Apply Nodal Loads

Case	node	Fx	Fy	Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	Total
2	6827	0	0	-5.520	11	1	0	0	0	2	78	0	0	0	1	1	0	0	0	0
		Mxz	Myz				d Mxz	d Myz				d Mxz	d Myz				d Mxz	d Myz		
		0	0				0	0				0	0				0	0		

Apply Total 0
Delete Done

Selection Mode
Pointer Window

Figure B-64

Apply Nodal Loads

Case	node	Fx	Fy	Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	Total
2	6839	0	0	-5.520	5	13	0	0	0	2	12	0	0	0	1	1	0	0	0	0
		Mxz	Myz				d Mxz	d Myz				d Mxz	d Myz				d Mxz	d Myz		
		0	0				0	0				0	0				0	0		

Apply Total 0
Delete Done

Selection Mode
Pointer Window

Figure B-65

Apply Nodal Loads

Case	node	Fx	Fy	Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	# nodes	d node	d Fx	d Fy	d Fz	Total
2	6840	0	0	-11.04	11	1	0	0	0	5	13	0	0	0	1	1	0	0	0	0
		Mxz	Myz				d Mxz	d Myz				d Mxz	d Myz				d Mxz	d Myz		
		0	0				0	0				0	0				0	0		

Apply Total 0
Delete Done

Selection Mode
Pointer Window

Figure B-66

To define the third loading case (Case 3) responsible for the lateral loading, the third button from the left in the toolbar shown in Figure B-53 with the number '3' is pressed. With that button pressed down, all the loads to be defined will be included in both 'Case 3'. The lateral load will be applied in the positive X-direction along the negative-Y edge at the top of the column in a displacement control regime resembling the test conditions.

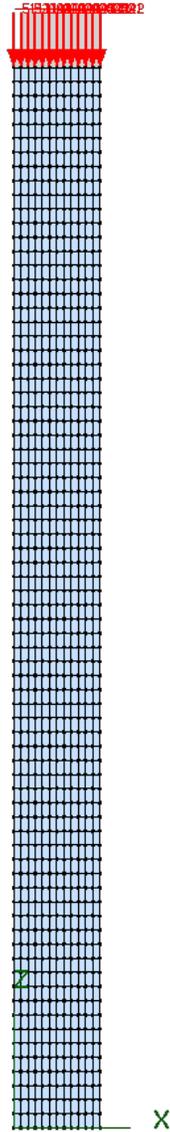


Figure B-67

Recalling Section A.2.5.2 of the *VecTor3: User's Manual* which is responsible for defining this type of loading (Support Displacements or Imposed Displacements), the parameters required for defining this type of load can be found. Pressing the button shown in Figure B-68 opens the window shown in Figure B-69 which is responsible for defining the support displacements.



Figure B-68

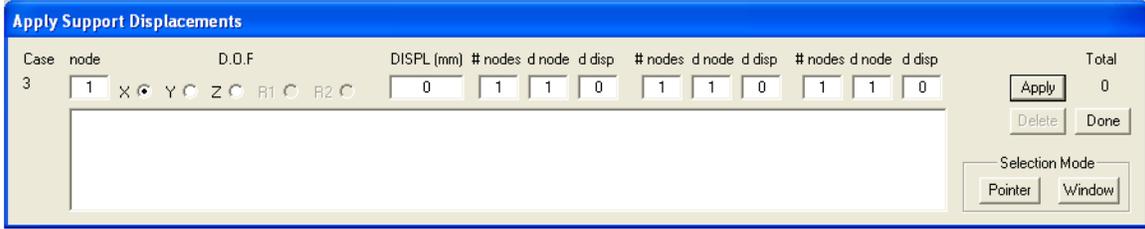


Figure B-69

The ID numbers of the nodes where the lateral displacement is to be applied can be found from Figure B-62. The line shown in Figure B-70 defines the lateral displacement to be applied in the X-direction starting from node # 6826. The applied displacement is 1 mm, which will be multiplied by the load factors defined in the ‘Analysis Job Control Parameters’ as explained in Section B.2.3.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*. Again, the loading case number (Case 3) is shown at the left-hand side of the window for referencing purposes.

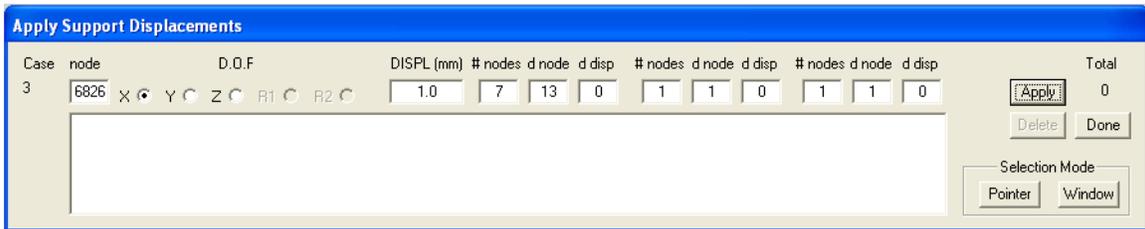


Figure B-70

Figure B-71 shows the applied lateral displacements along the height of the column and Figure B-72 shows them on a cross section.

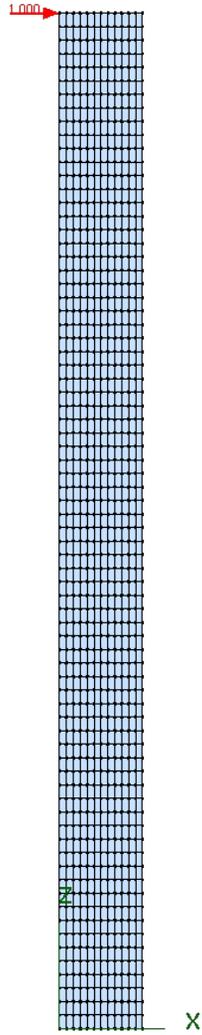


Figure B-71

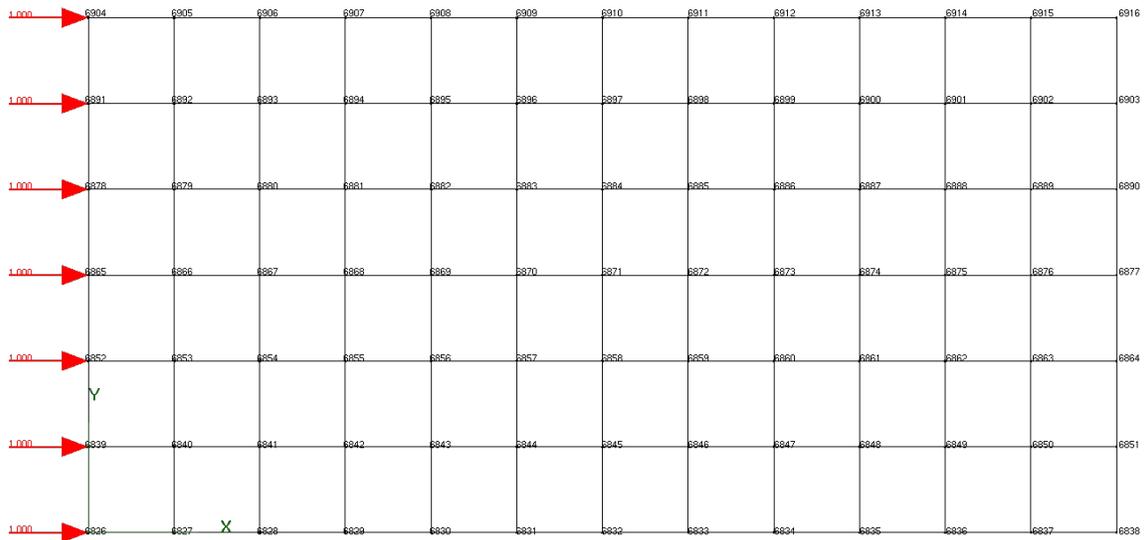


Figure B-72

As previously explained, only one loading profile needs to be defined for ‘Case 3’ and ‘Case 4’ as they both involve the same type of loading and at the same nodes but with different load factors. In Section B.2.3.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*, both ‘Case 3’ and ‘Case 4’ were assigned the same ‘Load file name’ (which was set to ‘Lateral’). There are three ways to handle this. The first one is to replicate the loading definition of ‘Case 3’ for ‘Case 4’ in the ‘Load Definition’ step. The second one is to define ‘Case 3’ only and opt not to save the file titled ‘lateral.l3r’ for ‘Case 4’ in the next step (Section B.2.5 which is responsible for saving the input files) when prompted to do so. The third way is to choose a ‘Load file name’ for ‘Case 4’ that is different from that chosen for ‘Case 3’ in Section B.2.3.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis* (‘Lateral 1’ and ‘Lateral 2’, for example).

B.2.5. Saving the Input Files and Running VecTor3

At this point, the model is ready to be analyzed. This can be done using the toolbar shown in Figure B-73. The first step is to save the analysis job (if it has not been saved yet) by pressing the button shown in Figure B-74.



Figure B-73



Figure B-74

As explained in the Introduction section (Section A.1 of the *VecTor3: User’s Manual*), there are five main input files that are required for defining the analysis job to be carried out. Pressing the first button from the left in the toolbar shown in Figure B-73, featuring the red floppy disc, creates and saves the ‘VECTOR.JOB’, ‘VT3.AUX’, and ‘VT3T.AUX’ files. Pressing the second button, featuring the blue floppy disc, creates and

saves the structure file with the name 'Struct' specified in the 'Job Control' tab in the 'Job Definition' step and the extension 'S3R'. Finally, the third button, featuring the green floppy disc, creates and saves the load files with the names specified in the 'Job Control' tab in the 'Job Definition' step and the extension 'L3R', namely, 'Thermal.L3R', 'Axial.L3R', and 'Lateral.L3R'.

At this point, VecTor3 has all the input files it requires and is ready to run. Pressing the fourth button from the left in the toolbar shown in Figure B-73, titled 'VT', starts VecTor3. Pressing this button without having already saved the input files using the buttons with the red, blue, and green floppy discs will prompt the user to save the files. When VecTor3 starts, it will appear in the form of an MS-DOS window that displays the progress of the analysis. For each load stage, the window will show the load stage number and a list of the loading cases and their respective load factors. Since this is a time-stepping analysis, the time step number and the time incident being analyzed (in seconds) will also be shown.

Next, the progress of convergence will be displayed for each iteration being analyzed. First, the convergence progress of the successive iterations of the heat transfer analysis will be displayed. After the convergence of the thermal analysis is achieved, the convergence progress of the successive iterations of the structural analysis will be displayed. This will carry on until convergence is achieved or the maximum number of iterations allowed per load stage (which was set to 60 in the 'Analysis Job Control Parameters' in the 'Job Definition' step in Section B.2.3.1 of the *VecTor3: Sample Coupled Thermal and Structural Analysis*) is reached. Then, a new stage will start. The first and second load stages progress is shown next.

VecTor3: Sample Coupled Thermal and Structural Analysis

```

* * * * *
*
*           V E C T O R 3
*
*       NONLINEAR ANALYSIS OF RC SOLIDS.
*
* * * * *

```

UNIVERSITY OF TORONTO
VERSION 2.00 : AUGUST 16 2013
COPYRIGHT : F.J. VECCHIO

```

* * * * *
*           LOAD STAGE No.:      1
*           Load case           Factor
*           FLT                 1.000
*           AXIAL                1.000
*           LATERAL              0.000
*           LATERAL              0.000
* * * * *
*           TIME STEP NO. :      1
*           Time:                0.00000000
* * * * *

```

structural Analysis Convergence

Iteration	Convergence					
1	1.015813	1.600013	9.999999	9.999999	9.999999	1.000000
2	1.300782	1.004789	1.006735	1.144991	9.999999	1.477601
3	1.009958	1.004424	1.007108	1.006533	1.005609	1.405853
4	1.005842	1.002016	1.003231	1.003408	1.005955	1.337848
5	1.002909	1.000752	1.001155	1.001784	1.003233	1.285651
6	1.001472	1.000152	1.000015	1.001141	1.001677	1.241457
7	1.000788	1.000336	1.000564	1.004615	1.000752	1.205836
8	1.000476	1.000486	1.000830	1.002092	1.000221	1.174754
9	1.000367	1.000542	1.000925	1.000000	1.000078	1.150856
10	1.000426	1.000546	1.000931	1.002915	1.000242	1.128775
11	1.000274	1.000523	1.000890	1.003763	1.000326	1.110461
12	1.000233	1.000489	1.000831	1.001114	1.000364	1.094007
13	1.000200	1.000449	1.000761	1.000000	1.000375	1.079997
14	1.000174	1.000408	1.000692	1.000000	1.000371	1.068131
15	1.000153	1.000369	1.000624	1.000000	1.000358	1.057987
16	1.000135	1.000331	1.000561	1.000000	1.000340	1.049364
17	1.000118	1.000297	1.000502	1.000000	1.000319	1.042032
18	1.000104	1.000266	1.000449	1.000000	1.000297	1.035815
19	1.000091	1.000237	1.000400	1.000000	1.000274	1.030506
20	1.000080	1.000211	1.000357	1.000000	1.000252	1.025989

STORING LOAD STAGE RESULTS IN ASCII FILE: T3_01.A3E

VecTor3: Sample Coupled Thermal and Structural Analysis

```

* * * * *
*                               LOAD STAGE No.: 2
*                               Load case      Factor
*                               FIT            1.000
*                               AXIAL         1.000
*                               LATERAL      1.000
*                               LATERAL      0.000
* * * * *
*                               TIME STEP NO. : 2
*                               Time:         60.00000000
* * * * *

```

Transient Heat Flow Convergence

Iteration	Convergence
1	0.118392E-01
2	0.123099E-03
3	0.117594E-05
4	0.110747E-07
5	0.103935E-09
6	0.973968E-12
7	0.981147E-14
8	0.239304E-15

Structural Analysis Convergence

Iteration	Convergence					
1	1.218516	1.679316	9.999999	9.999999	9.999999	1.193831
2	1.014961	1.001398	1.002168	1.196651	9.999999	1.118678
3	1.008226	1.001774	1.002948	1.001153	1.000746	1.091576
4	1.001488	1.001596	1.002647	1.000000	1.000160	1.064744
5	1.003917	1.001407	1.002330	1.000000	1.000255	1.047567
6	1.000771	1.001216	1.002010	1.000000	1.000245	1.034420
7	1.000581	1.001043	1.001721	1.000000	1.000217	1.027052
8	1.000446	1.000890	1.001468	1.000000	1.000186	1.038493
9	1.000360	1.000759	1.001250	1.000000	1.000155	1.042812
10	1.000283	1.000645	1.001061	1.000000	1.000126	1.039921
11	1.000235	1.000549	1.000904	1.000000	1.000101	1.037565
12	1.000196	1.000466	1.000767	1.000000	1.000080	1.035531
13	1.000163	1.000396	1.000650	1.000000	1.000062	1.033876
14	1.000151	1.000335	1.000551	1.000000	1.000047	1.032533
15	1.000124	1.000282	1.000463	1.000000	1.000036	1.039696
16	1.000097	1.000239	1.000393	1.000000	1.000028	1.038922
17	1.000102	1.000202	1.000331	1.000000	1.000019	1.038275
18	1.003070	1.000169	1.000278	1.000000	1.000012	1.029646
19	1.000141	1.000143	1.000235	1.000000	1.000009	1.028754
20	1.000096	1.000119	1.000196	1.000000	1.000005	1.028236

STORING LOAD STAGE RESULTS IN ASCII FILE: T3_02.A3E

For future reference, all the data displayed in the MS-DOS window while the analysis is running are also saved in a text file with the title 'VECTOR.TXT'.

After the analysis is over, the post-processor, Janus, can be used for a graphical display of the results. This can be done by starting Janus by pressing the button shown in Figure B-75, then opening the 'VECTOR.JOB' file in Janus.



Figure B-75

References

- Asatsu (Asazu), N., Unjo, S., Hoshikuma, J., and Kondo, M. (2001). Plastic hinge length of reinforced concrete columns based on the buckling characteristics of longitudinal reinforcement (In Japanese). *Journal of Structural Mechanics and Earthquake Engineering, Proceedings of Japan Society of Civil Engineers (JSCE)*, 682, 177-194.
- Aslani, F., and Bastami, M. (2011). Constitutive relationships for normal- and high-strength concrete at elevated temperatures. *American Concrete Institute (ACI) Materials Journal*, 108(4), 355-364.
- ASTM E119 – 12a, Standard Test Methods for Fire Tests of Building Construction and Materials*. (2012). American Society for Testing and Materials (ASTM). West Conshohocken, PA, 34 p.
- Attard, M. M., and Setunge, S. (1996). Stress-strain relationship of confined and unconfined concrete. *American Concrete Institute (ACI) Materials Journal*, 93(5), 432-442.
- Bauschinger, J. (1886). Über die veränderung der elastizitätsgrenze und der festigkeit des eisens und stahl durch strecken und quetschen, durch erwärmen und abkühlen und durch oftmal wiederholte beanspruchung. *Mittheilungen aus dem Mechanisch-Technischen Laboratorium der Königlichen Technischen Hochschule in München*, 13, 1-116.
- Bažant, Z. P., and Becq-Giraudon, E. (2002). Statistical prediction of fracture parameters of concrete and implications for choice of testing standard. *Cement and Concrete Research*, 32(4), 529-556.
- Bažant, Z. P., and Chern, J. C. (1987). Stress-induced thermal and shrinkage strains in concrete. *Journal of Engineering Mechanics - American Society of Civil Engineers (ASCE)*, 113(10), 1493-1511.
- Bentz, E. C. (2000). *Sectional Analysis of Reinforced Concrete Structures*. PhD Thesis, University of Toronto, Toronto, 184 p.
- Brockenbrough, R. L. (1970). Theoretical stresses and strains from heat curving. *Journal of the Structural Division - American Society of Civil Engineers (ASCE)*, 96(7), 1421-1444.
- CAN/ULC S101-07, Standard Methods of Fire Endurance Tests of Building Construction and Materials*. (2007). Underwriters Laboratories of Canada. Toronto, ON, 85 p.
- CEB-FIP Model Code for Concrete Structures - Buletin D'Information No. 124/125*. (1978). Comité Euro-International du Béton - Fédération Internationale De La Précontrainte (CEB-FIP), Lausanne, Switzerland, 348 p.

References

- CEB-FIP Model Code 1990 - Buletin D'Information No. 213/214.* (1993). Comité Euro-International du Béton - Fédération Internationale De La Précontrainte (CEB-FIP), Lausanne, Switzerland, 460 p.
- CEB1988, Concrete Structures under Impact and Impulsive Loading, Bulletin D'Information No. 187.* (1988). Comité Euro-International du Béton, Lausanne, Switzerland. 184 p.
- Centre Technique Industriel de la Construction Métallique (CTICM , Saint-Aubin, France), Méthode de Préviation par le Calcul du Comportement au Feu des Structures en Acier, Construction Métallique, No. 3.* (1982).
- Chak, I. N. (2013). *Janus: A Post-Processor for VecTor Analysis Software.* M.A.Sc. Thesis, University of Toronto, Toronto, ON, 193 p.
- Chang, C. H., and Jau, W.-C. (2001). *Study of Fired Concrete Strengthened with Confinement.* M.Sc. Thesis, National Chiao Tung University, Hsinchu, Taiwan.
- Chang, Y. F., Chen, Y. H., Sheu, M. S., and Yao, G. C. (2006). Residual stress–strain relationship for concrete after exposure to high temperatures. *Cement and Concrete Research*, 36(10), 1999-2005.
- Cheng, F.-P., Kodur, V. K. R., and Wang, T.-C. (2004). Stress-strain curves for high strength concrete at elevated temperatures. *Journal of Materials in Civil Engineering*, 16(1), 84-90.
- Collins, M. P., and Mitchell, D. (1987). *Prestressed Concrete Basics:* Canadian Prestressed Concrete Institute.
- Collins, M. P., and Porasz, A. (1989). Shear design for high strength concrete. *Bulletin d'Information No. 193 - Design Aspects of High Strength Concrete, Comité Euro-International du Béton (CEB)*, 75-83.
- Coulomb, C. A. (1776). Essai sur une application des règles de maximis et minimis à quelques problèmes de statique, relatifs à l'architecture. *Mémoires de Mathématique et de Physique, présentés à l'Académie Royale des Sciences, par divers savans, et lûs dans ses assemblée, 7*, 343-382.
- Cui, C., and Sheikh, S. (2010). Analytical model for circular normal- and high-strength concrete columns confined with frp. *Journal of Composites for Construction - American Society of Civil Engineers (ASCE)*, 14(5), 562-572.
- Dhakai, R. P., and Maekawa, K. (2002). Modeling for postyield buckling of reinforcement. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 128(9), 1139-1147.
- Dwaikat, M. B., and Kodur, V. K. R. (2009). Hydrothermal model for predicting fire-induced spalling in concrete structural systems. *Fire Safety Journal*, 44(3), 425-434.

References

- Eligehausen, R., Popov, E. P., and Bertero, V. V. (1983). *Local bond stress-slip relationships of deformed bars under generalized excitations: Experimental results and analytical model*. Earthquake Engineering Research Center, University of California, Berkeley., 169 p.
- ElMohandes, F. (2013). *Advanced Three-Dimensional Nonlinear Analysis of Reinforced Concrete Structures Subjected to Fire and Extreme Loads*. PhD Thesis, University of Toronto, Toronto, ON, 407 p.
- EN 1992-1-2:2004 - Eurocode 2: Design of Concrete Structures - Part 1-2: General Rules. Structural Fire Design. (2005). European Committee for Standardization (CEN), Brussels, Belgium, 97 p.
- ENV 1992-1-2:1996 - Eurocode 2: Design of Concrete Structures - Part 1-2: General Rules. Structural Fire Design. (1996). European Committee for Standardization (CEN), Brussels, Belgium, 63 p.
- European Convention for Constructional Steelwork (ECCS, Brussels, Belgium) – Technical Committee 3 – Fire Safety of Steel Structures, European Recommendations for the Fire Safety of Steel Structures - Calculation of the Fire Resistance of Load Bearing Elements and Structural Assemblies Exposed to the Standard Fire. (1983). Amsterdam, Netherlands. Elsevier, 106 p.
- fib Model Code 2010 - Final Draft, Bulletin No. 66. (2012). Fédération Internationale du Béton (fib), Lausanne, Switzerland. 370 p.
- Gan, Y. (2000). *Bond Stress and Slip Modeling in Nonlinear Finite Element Analysis of Reinforced Concrete Structures*. M.A.Sc. Thesis, University of Toronto, 251 p.
- Gawin, D., Majorana, C. E., and Schrefler, B. A. (1999). Numerical analysis of hygro-thermal behaviour and damage of concrete at high temperature. *Mechanics of Cohesive-frictional Materials*, 4(1), 37-74.
- Gupta, P. R. (1998). *Shear Design of Reinforced Concrete Members Under Axial Compression*. PhD Thesis, University of Toronto, Toronto, ON, 189 p.
- Harajli, M.H., and Mukaddam, M.A. (1988). Slip of steel bars in concrete joints under cyclic loading. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 114(9), 2017-2035.
- Hertz, K. D. (2005). Concrete strength for fire safety design. *Magazine of Concrete Research*, 57(8), 445–453.
- Hognestad, E. (1951). *A Study of Combined Bending and Axial Load in Reinforced Concrete Members*. University of Illinois, Engineering Experiment Station, Urbana, USA, 128 p.
- Hordijk, D. A. (1991). *Local Approach to Fatigue of Concrete*. PhD Thesis, Delft University of Technology, Delft, 210 p.

References

- Hoshikuma, J., Kawashima, K., Nagaya, K., and Taylor, A.W. (1997). Stress-strain model for confined reinforced concrete in bridge piers. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 123(5), 624-633.
- ISO 834-1:1999, *Fire-Resistance Tests - Elements of Building Construction - Part 1: General Requirements*. (1999). International Organization for Standardization (ISO). Geneva, Switzerland, 25 p.
- Izumo, J., Shin, H., Maekawa, K., and Okamura, H. (1991). *An analytical model for RC panels subjected to in-plane stresses*. Concrete Shear in Earthquake, University of Houston, Texas, USA.
- Jodai, A. (2013). *Nonlinear Finite Element Analysis and Post-Processing of Reinforced Concrete Structures under Transient Creep Strain*. M.A.Sc. Thesis, University of Toronto, Toronto, ON, 92 p.
- Kent, D. C., and Park, R. (1971). Flexural members with confined concrete. *Journal of the Structural Division - American Society of Civil Engineers (ASCE)*, 97(7), 1969-1990.
- Kodur, V. K. R., and Khaliq, W. (2011). Effect of Temperature on Thermal Properties of Different Types of High-Strength Concrete. *Journal of Materials in Civil Engineering*, 23(6), 793-801.
- Kodur, V. K. R., and Sultan, M. A. (2003). Effect of Temperature on Thermal Properties of High-Strength Concrete. *Journal of Materials in Civil Engineering*, 15(2), 101-107.
- Kodur, V. K. R., Wang, T.-C., and Cheng, F.-P. (2004). Predicting the fire resistance behaviour of high strength concrete columns. *Cement & Concrete Composites*, 26(2), 141-153.
- Kollegger, J., and Mehlhorn, G. (1990). *Experimentelle Untersuchungen zur Bestimmung der Druckfestigkeit des gerissenen Stahlbetons bei einer Querkzugbeanspruchung*. Deutscher Ausschuss Für Stahlbeton, Berlin, Germany, 132 p.
- Kupfer, H., Hilsdorf, H.K., and Rusch, H. (1969). Behavior of concrete under biaxial stress. *American Concrete Institute (ACI) Journal*, 87(2), 656-666.
- Kwon, M., and Spacone, E. (2002). Three-dimensional finite element analyses of reinforced concrete columns. *Computers & Structures*, 8(2), 199-212.
- Lee, S.-C., Cho, J.-Y., and Vecchio, F.J. (2011a). Diverse embedment model for steel fibre reinforced concrete in tension: model development. *American Concrete Institute (ACI) Materials Journal*, 108(5), 516-525.
- Lee, S.-C., Cho, J.-Y., and Vecchio, F.J. (2011b). Diverse embedment model for steel fibre reinforced concrete in tension: model verification. *American Concrete Institute (ACI) Materials Journal*, 108(5), 526-535.

References

- Lee, S.-C., Cho, J.-Y., and Vecchio, F.J. (2013). Simplified diverse embedment model for steel fiber- reinforced concrete elements in tension. *American Concrete Institute (ACI) Materials Journal*, 110(4), 403-412.
- Li, L., and Purkiss, J.A. (2005). Stress-strain constitutive equations of concrete material at elevated temperatures. *Fire Safety Journal*, 40, 669-686.
- Li, W., and Guo, Z. (1993). Experimental investigation on strength and deformation of concrete under high temperature (in Chinese). *Journal of Building Structures*, 14(1), 8-16.
- Lie, T. T. (1992). *Structural Fire Protection (ASCE Manuals and Reports on Engineering Practice No. 78)*. New York, NY: American Society of Civil Engineers (ASCE).
- Lie, T. T., and Kodur , V. K. R. (1996). Thermal and mechanical properties of steel-fibre-reinforced concrete at elevated temperatures. *Canadian Journal of Civil Engineering*, 23(4), 511-517.
- Lie, T. T., and Lin, T. D. (1985). Fire performance of reinforced concrete columns. *American Society for Testing and Materials (ASTM) Special Technical Publication - STP882 - Fire Safety: Science and Engineering*, 176-205.
- Lie, T. T., Rowe, T. J., and Lin, T. D. (1986). Residual strength of fire-exposed reinforced concrete columns. *American Concrete Institute (ACI) Special Publications - SP92 - Evaluation and Repair of Fire Damage to Concrete*, 153-174.
- Lu, Z. (1989). *A Research on Fire Response of Reinforced Concrete Beams (in Chinese)*. PhD Thesis, Tongji University.
- Malvar, L. J., and Crawford, J. E. (1998). *Dynamic increase factors for concrete*. Twenty-Eighth Department of Defense Explosives Safety Seminar (DDESB), Orlando, FL.
- Mander, J. B., Priestley, M. J. N., and R., Park. (1988). Theoretical stress-strain model for confined concrete. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 114(8), 1804-1826.
- Miyahara, T., Kawakami, T., and and Maekawa, K. (1987). Nonlinear behavior of cracked reinforced concrete plate element under uniaxial compression. *Proceedings of Japan Society of Civil Engineers (JSCE)*, No.378/V-6, 306-319.
- Miyahara, T., Kawakami, T., and and Maekawa, K. (1988). Nonlinear behavior of cracked reinforced concrete plate element under uniaxial compression. *Concrete Library International, Japan Society of Civil Engineers (JSCE)*, 11, 131-144.
- Moftah, M. (2008). *Numerical Modelling and Performance of Reinforced Concrete Members under Fire Condition*. PhD Thesis, University of Western Ontario, Waterloo, ON, 286 p.

References

- Mohr, C. O. (1900). Welche Umstände bedingen die Elastizitätsgrenze und den Bruch eines Materials? *Zeitschrift des Vereins Deutscher Ingenieure*, 24, 1524–1530 and 1572–1577.
- Montoya, E. (2003). *Behaviour and Analysis of Confined Concrete*. PhD Thesis, University of Toronto, Toronto, 297 p.
- Morita, S., and Fujii, S. (1982). *Bond capacity of deformed bars due to splitting of surrounding concrete*. Bond in concrete: Proceedings of the International Conference on Bond in Concrete, Paisley College of Technology, Scotland.
- Mostafaei, H., Leroux, P., and Lafrance, P.-S. (2012). *Fire Endurance Of A Reinforced Concrete Column Under Both Axial And Lateral Loads*. NRC Report IRC-RR-327, National Research Council Canada (NRC), Ottawa, ON, 31 p.
- Newmark, N. M. (1959). A method of computation for structural dynamics. *Journal of Engineering Mechanics, American Society of Civil Engineers (ASCE)*, 85(EM3), 67-94.
- Okamura, H., and Maekawa, K. (1991). *Nonlinear Analysis and Constitutive Models of Reinforced Concrete*. Tokyo: Gihodo-Shuppan Co.
- Ottosen, N. S. (1979). Constitutive model for short-time loading of concrete. *Journal of the Engineering Mechanics Division, American Society of Civil Engineers (ASCE)*, 105(1), 127–141.
- Palermo, D. (2002). *Behaviour and Analysis of Reinforced Concrete Walls Subjected to Reversed Cyclic Loading*. PhD Thesis, University of Toronto, Toronto, ON, 492 p.
- Park, R., Priestley, M. J. N., and Gill, W. D. . (1982). Ductility of square-confined concrete columns. *Journal of the Structural Division, American Society of Civil Engineers (ASCE)*, 108(4), 929-950.
- Popovics, S. (1973). A numerical approach to the complete stress-strain curve of concrete. *Cement and Concrete Research*, 3(5), 583-599.
- Rayleigh, J. W. S. (1877). *The Theory of Sound, reissued in 1945, Vol. 1, 2nd ed.* Dover Publications, New York, NY, 480 p.
- Richart, F.E., Brandtzaeg, A., and Brown, R.L. (1928). *A Study of the Failure of Concrete under Combined Compressive Stresses*. University of Illinois Engineering Experimental Station, Urbana, Illinois, 104 p.
- Saab, H. A. (1990). *Non-linear Finite Element Analysis of Steel Frames in Fire Conditions*. PhD Thesis, University of Sheffield, Sheffield, England, 143 p.
- Sadeghian, V. (2012). *FormWorks-Plus: Improved Pre-processor for VecTor Analysis Software*. M.A.Sc. Thesis, University of Toronto, Toronto, ON, 147 p.

References

- Saenz, L. P. (1964). Discussion of “Equation for the stress-strain curve of concrete” by P. Desayi and S. Krishnan. *American Concrete Institute (ACI) Journal*, 61(9), 1229–1235.
- Sato, Y., and Vecchio, F.J. (2003). Tension stiffening and crack formation in RC members with FRP sheets. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 129(6), 717-724.
- Seckin, M. (1981). *Hysteretic Behaviour of Cast-in-Place Exterior Beam-Column-Slab Subassemblies*. PhD Thesis, University of Toronto, Toronto, ON, 266 p.
- Selby, R. G. (1990). *Nonlinear Finite Element Analysis of Reinforced Concrete Solids*. Master of Applied Science Thesis, University of Toronto, 144 p.
- Selby, R. G., and Vecchio, F. J. (1997). A constitutive model for analysis of reinforced concrete solids. *Canadian Journal of Civil Engineering*, 24(3), 460-470.
- Shin, K.-Y., Kim, S.-B., Kim, J.-H., Chung, M., and Jung, P.-S. (2002). Thermo-physical properties and transient heat transfer of concrete at elevated temperatures. *Nuclear Engineering and Design*, 212(1–3), 233-241.
- Shirai, S., and Noguchi, H. (1989). *Compressive deterioration of cracked concrete*. ASCE Structures Congress 1989: Design, Analysis and Testing, New York, NY.
- Smith, G. M., and Young, L. E. (1956). Ultimate flexural analysis based on stress-strain curves of cylinders. *American Concrete Institute (ACI) Journal Proceedings*, 53(12), 597-609.
- Song, Y., Zhang, Z., Qing, L., and Yu, C. (2007). Biaxial tensile-compressive experiment of concrete at high temperatures. *Frontiers of Architecture and Civil Engineering in China*, 1(1), 94-98.
- Spinella, N., Colajanni, P., and Recupero, A. (2010). Simple plastic model for shear critical sfrc beams. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 136(4), 390-400.
- Tanchev, R. T., Li, L. Y., and Purkiss, J. A. (2001). Finite element analysis of coupled heat and moisture transfer in concrete subjected to fire. *Numerical Heat Transfer, Part A: Applications*, 39(7), 685-710.
- Toumi, B., and Resheidat, M. (2010). Influence of high temperatures on surface cracking of concrete studied by image scanning technique. *Jordan Journal of Civil Engineering*, 4(2), 155-163.
- Vecchio, F. J. (1982). *The Response of Reinforced Concrete to In-Plane Shear and Normal Stresses*. Ph.D Thesis, University of Toronto, 332 p.

References

- Vecchio, F. J., and Collins, M. P. (1986). The modified compression field theory for reinforced concrete elements subjected to shear. *American Concrete Institute (ACI) Journal*, 83(2), 219-231.
- Vecchio, F. J., and Collins, M. P. (1993). Compression response of cracked reinforced concrete. *Journal of Structural Engineering - American Society of Civil Engineers (ASCE)*, 119(12), 3590-3610.
- Vecchio, F. J., and Lai, D. (2004). Crack shear-slip in reinforced concrete elements. *Journal of Advanced Concrete Technology*, 2(3), 289-300.
- Vintzeleou, E. N., and Tassios, T. P. (1987). Behavior of dowels under cyclic deformations. *American Concrete Institute (ACI) Structural Journal*, 84(1), 18-30.
- Voo, J. Y. L., and Foster, S. J. (2003). *Variable engagement model for fibre reinforced concrete in tension*. School of Civil and Environmental Engineering, University of New South Wales, UNSW Sydney, Australia, 86 p.
- Walraven, J.C., and Reinhardt, H.W. (1981). Theory and experiments on the mechanical behaviour of cracks in plain and reinforced concrete subjected to shear loading. *Heron*, 26(1A), 1-68.
- Wong, P. S., Trommels, H., and Vecchio, F. J. (2013). *VecTor2 and FormWorks User's Manual, Second Edition*. Department of Civil Engineering, University of Toronto, 318 p.
- Xie, D., and Qian, Z. (1998). Research on bond and tension of concrete after high temperature (in Chinese). *Journal of Zhejiang University (Natural Science)*, 32(5), 597-602.
- Yamamoto, T. (1999). *Nonlinear Finite Element Analysis of Transverse Shear and Torsional Problems in Reinforced Concrete Shells*. M.A.Sc. Thesis, University of Toronto, 112 p.

Appendix A: Sample VECTOR.EQR File

4

"IMPERIAL VALLEY EARTHQUAKE - EL CENTRO"

"MAY 18, 1940, 2037 PST"

"CORRECTED ACCELEROGRAM, 180 DEGREES, CALTECH IIA001"

"SOURCE: NISEE, U.C. BERKELEY, CALIFORNIA"

2674 0.02 8 G

1337 0.04 8 G

535 0.10 8 G

2674 POINTS OF ACCEL DATA EQUALLY SPACED AT 0.02 SEC. (UNITS: MM/SEC/SEC)

3	19	68	29	29	54	83	53
14	53	134	124	66	61	133	49
96	188	243	133	46	-81	-156	-85
-31	40	85	132	180	162	23	25
80	77	110	131	81	13	-151	-330
-532	-646	-525	-415	-401	-515	-569	-572
-528	-497	-498	-550	-492	-431	-362	-280
-204	-147	-213	-47	215	479	580	188
-122	-538	-968	-1100	-1059	-1070	-999	-936
-1128	-1469	-1213	-1099	-858	-636	-380	-155
107	338	599	835	1092	1325	1485	1192
1105	1003	1079	824	369	-225	-1481	-1786
-1589	-1693	-1521	-1446	-1348	-1265	-1179	-1088
-997	-764	-595	-95	542	1059	770	585
247	-128	-675	-826	-521	-300	74	447
852	1118	922	775	544	156	-183	-612
-700	-318	-38	284	594	919	1109	651
270	-151	144	330	772	1181	1592	1780
1487	1183	788	326	351	529	690	711
663	621	542	521	634	698	810	431
-55	-459	-206	-7	260	486	764	879
611	351	198	429	579	864	946	414
-59	-511	-926	-953	-934	-888	-861	-740
-624	-501	-376	-240	-125	15	-171	-421
-711	-643	-277	-65	169	-230	-590	-570
-65	147	205	209	439	509	482	-1563
-985	-1074	-673	-256	261	615	1022	1093
954	875	661	523	676	1079	770	869
592	645	-1083	-930	-1068	-1170	-1035	-1011
-1532	-1245	-1117	-874	-613	-258	24	438
694	708	690	673	623	794	630	525
463	533	623	517	206	-67	-132	49
536	1002	323	-181	117	461	716	1061
1216	1100	528	17	46	-140	-107	-338
-780	-1000	-990	-931	-683	-441	-211	17
198	578	531	84	-74	-403	-533	-498
-301	-106	-444	-792	-471	-414	-138	98
323	10	-287	-405	-235	-167	67	237
405	339	134	-48	-271	-439	-125	118
472	637	212	-64	-420	-661	-558	-500
-353	-173	-5	161	132	140	117	310
288	154	85	-76	-181	-366	-291	-13
274	245	109	92	-103	-240	-407	-526
-494	-719	-523	-260	131	693	873	616
419	-91	-549	-929	-853	-578	-396	-658
-718	-341	-43	262	567	801	961	856
661	498	284	113	-175	-476	-738	-517
-369	-113	-249	-317	-471	-389	-232	-27
226	493	632	460	353	199	63	-125
-299	-388	-320	-180	-229	-364	-462	-578
-690	-759	-583	-434	-367	-632	-662	-525
-353	-226	51	-188	-427	-149	-123	79
241	2	-243	-477	-752	-1001	-1278	-960
-278	419	989	790	1138	1223	1117	409
653	532	847	1	-234	-134	-337	-331
-194	160	474	602	315	205	70	389
836	996	1072	1098	787	713	741	744
777	841	852	908	221	52	139	73
182	224	315	285	64	-108	-360	-461
-343	-343	-278	-273	-640	-935	-743	-485
-50	351	460	89	-332	-637	-1022	-728
-474	119	326	253	144	161	262	377
305	-22	-307	-700	-830	-938	-416	-541

Appendix A: Sample VECTOR.EQR File

-980	-415	95	606	962	988	451	-21
-374	-81	88	363	746	468	-385	-35
-244	-35	-238	-427	-281	-243	-138	-21
46	-89	-223	-418	-643	-471	-109	320
607	125	174	-280	40	297	505	424
93	-222	-352	-62	267	791	709	519
318	72	-142	-584	-1008	-1384	-1326	-1175
-707	-630	-752	-574	-303	-66	385	842
1230	1442	1702	1899	2101	1984	1826	1652
1447	1230	1170	1202	714	385	-85	-499
-932	-1343	-1656	-851	-453	-28	488	394
70	-406	-1025	-1197	-1372	-1385	-1445	-1115
-783	-362	-246	-544	-698	-920	-966	-834
-724	-568	-411	-259	-37	226	413	556
733	617	361	309	252	92	-46	-188
-210	-245	-251	-273	-210	-67	77	202
125	50	-31	-126	416	700	568	559
422	327	203	130	125	89	-115	-68
97	-63	-168	-426	-298	-96	94	393
405	202	50	-257	-377	-307	-264	-168
-78	45	181	36	-138	-290	-488	-408
-239	-61	5	-159	-264	-398	-291	-237
-99	31	167	308	430	594	537	175
-44	-244	-428	314	505	237	265	-157
-342	-223	-256	-386	-361	20	393	738
1028	1100	566	255	-149	-379	-181	-48
251	211	-21	-207	-496	-683	-430	-147
20	299	725	928	55	-568	-819	-688
-460	-178	119	376	596	518	205	58
-422	-391	-339	-501	-377	-310	-274	-358
-329	-168	35	229	666	754	439	233
-7	-293	-494	-315	-226	-41	69	-35
-176	-170	-47	76	335	429	309	235
71	96	378	459	501	366	303	180
225	224	273	238	160	97	36	-35
-21	44	186	364	462	573	707	610
249	75	57	50	64	-17	-302	-548
-490	-471	-338	-223	-76	37	187	289
416	494	636	485	240	62	-221	-248
102	-69	-163	-292	-442	-535	-372	-179
55	353	415	331	304	190	122	15
-50	-207	-298	-359	-439	-483	-448	-422
-377	-327	-272	-225	-165	-125	-115	-122
-113	-118	-3	82	257	367	524	-111
-306	-265	-321	-159	-50	88	16	-44
-153	-230	-178	-151	-91	-38	26	57
-4	-58	-122	-70	221	460	805	831
462	228	-91	-345	-591	-779	-853	-740
-646	-630	-577	-390	-248	-125	-184	-134
-28	187	168	37	-69	-81	8	37
119	175	245	303	367	415	390	361
334	260	141	22	-102	-188	-73	58
283	301	148	33	-173	-243	-226	-206
-146	-5	134	276	431	484	546	576
640	503	230	8	-280	-533	-813	-941
-761	-629	-362	-71	56	40	69	55
71	-80	-91	-103	-258	-364	-481	-544
-317	-94	5	-62	-107	-213	-57	57
200	311	442	450	503	536	460	338
232	81	-54	-202	-318	-448	-567	-652
-524	-414	-251	-87	107	299	482	671
715	489	371	165	-49	-252	-531	-652
-432	-308	-94	51	242	391	551	412
262	58	-136	-287	-317	-257	-146	-7
175	398	565	722	812	784	843	680
511	340	147	-33	-228	-410	-580	-386
-453	-571	-668	-790	-948	-912	-634	-262
239	847	913	735	615	363	151	-118
-319	-217	-166	-51	-86	-99	-164	-28
176	386	463	362	284	160	15	-145
-300	-466	-578	-503	-453	-367	-273	-218
-345	-366	-473	-491	-299	-112	81	308

Appendix A: Sample VECTOR.EQR File

212	25	-138	-380	-459	-339	-245	-79
74	224	338	413	284	270	189	133
67	2	-94	-179	-275	-358	-278	-163
-54	95	166	201	302	370	454	354
217	73	-66	52	167	293	477	335
1	-134	-379	-658	-673	-329	-165	148
428	711	519	356	51	-436	-514	-469
-402	-223	-107	36	157	277	189	116
2	-109	-241	-340	-477	-385	-196	-85
85	203	354	416	347	267	139	9
-131	-293	-318	-198	-139	-34	64	79
-11	53	116	195	186	139	171	208
146	85	13	-66	-145	-229	-313	-401
-479	-588	-676	-548	-413	-207	-5	220
459	680	933	925	797	647	371	153
-143	-290	-16	243	484	728	437	152
-188	-508	-830	-767	-618	-468	-263	23
384	714	991	684	-179	-757	-584	-517
-293	-68	150	393	615	869	823	617
462	207	-76	-344	-586	-846	-1081	-1176
-989	-855	-656	-465	-257	-47	166	379
593	792	962	1147	1310	1487	1358	1133
905	566	266	-70	-439	-729	-460	-157
203	571	905	628	308	-70	-414	-259
-171	55	294	515	765	872	728	630
432	191	-36	-277	-317	-339	-366	-410
-395	-418	89	489	955	1150	892	700
392	80	-289	-600	-515	-492	-400	-308
-200	-123	-42	-163	-453	-698	-852	-889
-829	-717	-529	-238	-13	119	254	351
407	467	437	221	22	-198	-447	-356
-144	66	351	609	588	441	328	136
-14	-192	-209	-147	-192	-168	-101	-24
62	156	143	131	96	65	15	59
179	283	348	184	69	-131	-316	-540
-483	-297	-128	67	-8	-91	-176	-290
-401	-490	-439	-350	-247	-160	-101	-188
-216	-252	-219	-132	-142	-222	-250	-326
-378	-330	-289	-254	-220	-188	-158	-132
-121	-104	-70	-33	-16	13	-20	-93
-147	-238	-244	-184	-166	-119	-71	-21
3	-102	-174	-277	-169	-42	158	164
19	-42	-154	-251	-352	-280	-181	-20
102	211	280	132	89	6	-49	-98
-79	-21	10	-88	-157	-214	-159	-33
104	230	286	232	133	6	-121	-246
-251	-192	-140	-65	2	76	142	220
244	172	111	36	-50	-104	-166	-126
-25	58	57	0	-17	-11	-1	42
169	212	290	211	47	-75	-124	-37
68	185	284	390	436	410	343	281
200	129	63	-2	-58	-69	-100	-113
-151	-186	-197	-203	-227	-164	-47	46
147	248	262	235	243	278	313	332
234	143	69	-12	-124	-225	-333	-333
-307	-236	-160	-81	-54	-25	11	46
83	113	127	98	57	15	-43	-93
-150	-201	-254	-306	-350	-371	-340	-190
-71	73	214	324	265	244	187	134
75	27	39	63	30	-1	-43	-60
-33	-13	-1	-61	-86	-107	-126	-147
-119	-71	-25	25	32	42	73	113
154	196	243	214	144	84	22	77
111	174	192	127	83	7	-38	-28
-11	-74	-176	-252	-200	-170	-98	-32
42	111	177	125	77	13	-53	-78
-91	-109	-110	-109	-71	-8	44	118
129	72	29	-35	-97	-143	-137	-143
-135	-117	-38	21	109	115	58	19
-47	-58	-67	-72	-71	-53	-28	-53
-104	-147	-198	-154	-124	-72	-44	-35
-25	-5	26	42	50	76	115	160

Appendix A: Sample VECTOR.EQR File

137	78	29	-35	-52	-82	-118	-133
-123	-126	-134	-113	-58	-13	46	99
160	204	233	242	210	168	119	71
49	30	42	61	53	17	-11	-52
-89	-113	-92	-45	11	68	121	71
39	-12	-23	-22	-1	25	40	-9
-45	-98	-148	-194	-165	-144	-111	-85
-57	-23	16	54	86	99	118	130
149	141	101	63	15	-23	-48	-75
-98	-107	-111	-81	-17	47	89	88
75	53	53	72	37	7	-19	-34
-55	-75	-79	-51	-45	-45	-52	-16
56	75	89	103	109	118	96	66
37	9	25	33	53	64	68	53
-6	-61	-104	-103	-81	-34	9	59
98	104	89	53	33	30	25	22
-14	-46	-83	-123	-142	-95	-62	-10
19	29	44	50	62	57	39	13
-23	-56	-94	-121	-104	-85	-26	28
90	142	165	164	143	125	99	69
25	-12	-48	-23	-15	-11	-1	5
4	-7	-12	1	12	26	36	25
10	-9	-29	-50	-67	-71	-73	-67
-64	-57	-58	-75	-88	-106	-118	-114
-103	-90	-76	-65	-72	-82	-94	-107
-121	-131	-124	-119	-111	-104	-81	-43
-10	72	119	136	133	121	109	92
75	40	14	-25	-13	35	69	126
145	113	92	78	63	55	74	87
106	107	103	106	115	121	126	115
103	79	58	34	9	-7	27	64
110	155	148	134	119	96	77	56
59	63	68	76	82	91	90	58
29	10	3	-13	-21	-35	-33	-12
4	9	17	11	-4	-16	-33	-48
-65	-64	-49	-35	11	28	52	54
44	34	31	36	37	25	10	-3
-12	-19	-46	-82	-108	-101	-78	-53
-22	7	32	38	29	23	12	20
35	27	13	2	-17	-25	-20	-16
-10	-2	-10	-24	-39	-50	-47	-22
17	53	96	132	146	130	109	83
63	71	74	79	45	16	-18	-56
-76	-77	-83	-81	-76	-72	-85	-92
-106	-116	-128	-139	-144	-131	-122	-107
-102	-119	-134	-153	-172	-170	-157	-126
-94	-60	-33	-2	20	26	22	16
8	-6	-31	-54	-71	-59	-48	-28
-10	15	37	60	58	50	56	65
75	81	74	69	61	53	44	41
53	62	73	64	57	46	43	58
72	83	81	82	79	79	56	24
-6	-40	-57	-75	-92	-100	-88	-81
-64	-37	-11	2	12	22	38	57
76	90	81	72	57	44	16	-2
-13	-27	-37	-50	-55	-47	-39	2
42	80	85	96	98	102	102	96
90	85	89	94	103	114	124	136
146	151	135	121	107	95	81	69
53	40	23	21	28	34	28	14
6	-5	-15	-16	-11	-8	0	6
13	1	-10	-25	-34	-34	-39	-53
-66	-82	-95	-108	-112	-109	-108	-106
-120	-133	-148	-165	-169	-158	-133	-97
-64	-26	0	25	34	29	26	18
18	31	38	48	58	67	61	51
27	5	-20	-33	-48	-60	-72	-85
-91	-83	-79	-71	-81	-95	-112	-130
-146	-134	-122	-107	-90	-73	-59	-68
-78	-84	-67	-54	-35	-20	-38	-55
-80	-104	-120	-130	-119	-102	-62	-25
17	50	62	52	42	24	3	-3

Appendix A: Sample VECTOR.EQR File

2	-2	-18	-33	-45	-44	-45	-42
-40	-39	-53	-66	-82	-97	-89	-82
-71	-68	-87	-101	-121	-134	-146	-157
-167	-174	-181	-176	-170	-164	-152	-125
-100	-71	-43	-26	-13	2	15	31
32	24	21	1	-30	-49	-68	-72
-60	-47	-41	-36	-31	-25	-6	10
29	45	64	71	66	64	59	76
91	99	107	112	95	76	56	32
18	12	9	12	5	-4	-13	-23
-34	-46	-62	-64	-51	-36	-15	4
13	11	8	3	4	5	7	9
9	3	-1	-7	5	23	41	56
59	64	67	67	57	49	39	29
32	39	45	54	62	71	77	80
86	99	103	93	85	80	94	104
120	132	120	110	97	94	93	92
90	89	89	95	99	106	100	93
84	77	72	68	64	64	67	61
39	20	-4	-18	-25	-35	-37	-28
-20	-8	9	30	66	99	135	148
162	173	181	158	136	109	88	84
79	76	74	62	44	30	29	29
25	19	13	16	20	25	15	-2
-18	-37	-47	-58	-68	-77	-86	-90
-89	-86	-82	-78	-81	-85	-90	-91
-70	-52	-34	-19	-4	-3	-7	-3
7	15	8	-1	-10	-22	-36	-49
-62	-58	-55	-49	-44	-26	3	25
35	46	56	72	84	102	104	95
87	83	94	106	92	64	39	8
-22	-29	-21	-15	-6	-24	-38	-58
-65	-47	-38	-32	-26	-22	-31	-40
-49	-61	-64	-53	-48	-38	-35	-46
-44	-34	-27	-15	-5	6	13	10
9	5	-3	-11	-15	-18	-21	-24
-27	-36	-46	-56	-72	-84	-101	-100
-89	-47	-69	-130	-179	-233	-220	-225
-196	-131	-64	2	82	80	38	59
6	-50	-106	-160	-161	-141	-120	-92
-64	-84	-79	1	51	140	132	51
-11	-95	-49	-8	50	101	112	136
109	60	22	50	69	70	82	86
98	136	131	87	48	22	33	33
48	47	41	36	30	44	56	73
80	78	77	81	96	108	120	109
98	82	54	20	-14	-38	-14	6
16	-11	-30	-52	-45	-56	-55	-46
-42	-33	-32	-31	-25	25	9	-21
-33	-40	-41	-30	-24	-14	-6	4
12	21	31	27	11	9	11	14
18	23	30	45	57	74	75	63
57	44	33	21	21	26	24	15
-14	-54	-91	-135	-120	-78	-46	8
13	-13	-27	-58	-49	-52	-36	-43
-34	-24						
1337 POINTS OF VELOC DATA EQUALLY SPACED AT 0.04 SEC. (UNITS: CM/SEC)							
11.832	11.940	12.095	12.314	12.517	12.773	13.222	13.542
13.870	14.585	15.142	14.870	14.512	14.646	15.176	15.703
15.857	16.201	16.654	16.610	15.269	12.920	11.163	9.163
6.921	4.902	2.813	1.096	-0.029	-0.738	-0.828	0.924
1.758	-0.407	-4.635	-8.831	-12.831	-18.108	-22.377	-24.887
-25.470	-24.088	-20.727	-15.500	-10.526	-6.335	-3.239	-4.802
-11.445	-17.942	-23.702	-28.759	-33.112	-36.232	-36.473	-33.042
-30.854	-31.539	-34.387	-35.434	-33.613	-29.603	-26.587	-25.913
-28.021	-29.393	-28.268	-24.727	-22.047	-21.934	-20.356	-15.630
-8.992	-4.351	-2.560	-0.461	2.314	4.761	6.979	9.819
11.437	10.259	10.298	12.294	15.425	16.936	18.572	21.826
23.542	21.537	17.771	14.201	11.236	9.234	8.253	7.986
6.262	3.989	3.752	2.872	1.076	1.511	2.572	4.511
0.883	-2.923	-3.847	-1.335	2.828	6.192	8.575	12.179
15.280	16.078	12.068	7.625	3.036	-2.103	-5.582	-6.687

Appendix A: Sample VECTOR.EQR File

-5.093	-2.293	0.367	3.207	5.351	7.503	9.367	9.307
9.809	12.672	12.750	14.506	18.560	22.506	23.114	22.772
21.207	17.437	13.902	12.126	12.148	14.033	14.658	13.245
11.416	10.458	7.959	6.521	6.903	6.959	5.628	5.125
6.069	7.286	7.053	5.779	6.364	8.323	7.986	5.686
3.775	3.072	3.522	4.050	5.075	5.755	5.507	4.303
4.260	5.133	5.323	4.332	2.378	-0.078	-0.989	1.400
3.925	3.614	0.354	-2.051	-4.482	-5.926	-4.878	-1.748
1.587	3.528	3.863	1.998	-0.143	-0.987	-2.342	-3.822
-3.881	-2.037	-0.132	0.681	0.194	-1.200	-2.109	-3.528
-5.837	-8.629	-10.448	-12.741	-14.806	-15.561	-16.313	-17.161
-16.883	-16.882	-18.832	-22.862	-26.337	-24.787	-21.081	-16.382
-13.794	-11.231	-10.615	-11.454	-12.649	-12.048	-10.054	-9.259
-7.575	-3.676	0.379	3.334	6.340	9.650	12.537	13.002
13.468	14.412	15.361	14.848	13.222	11.915	10.452	7.199
5.434	6.546	6.852	4.224	1.272	1.360	2.336	3.064
4.386	4.339	1.802	-1.320	-3.797	-5.512	-3.243	0.145
0.180	-0.268	1.292	2.589	1.890	1.339	-0.034	-0.938
-1.071	-1.427	-3.130	-4.824	-3.686	-2.655	-3.002	-1.864
-0.417	-1.119	-1.329	1.227	3.292	3.613	1.294	-3.807
-8.190	-10.909	-13.113	-13.162	-9.862	-4.046	3.554	11.449
18.026	23.104	27.390	28.787	26.772	21.499	17.689	17.667
19.012	17.245	12.454	6.865	2.406	0.652	-1.379	-4.883
-8.241	-10.512	-11.477	-10.648	-8.388	-6.060	-4.829	-4.438
-5.070	-6.020	-7.027	-7.295	-6.690	-6.496	-6.364	-3.979
-1.871	-0.592	-0.003	0.185	0.031	-0.167	-1.484	-1.878
-0.594	0.267	-0.575	-1.831	-2.509	-2.315	-2.199	-3.406
-4.949	-5.304	-5.882	-7.234	-8.098	-7.969	-6.756	-4.602
-3.759	-4.720	-4.015	-2.771	-3.162	-4.206	-5.596	-5.523
-2.627	1.166	2.093	1.005	0.979	1.631	0.701	-1.590
-2.293	-0.950	1.685	-0.216	-2.872	-3.570	-2.103	-0.266
-0.367	-1.910	-3.627	-4.899	-6.217	-6.848	-5.688	-3.075
-2.177	-3.263	-4.613	-4.853	-5.029	-5.592	-5.152	-3.650
-2.801	-2.161	-0.363	1.174	2.061	3.007	3.917	4.307
4.253	4.507	5.883	8.197	10.373	10.828	11.049	10.777
8.890	7.121	6.261	6.447	7.628	9.668	11.515	11.659
11.044	10.844	9.654	7.771	7.097	8.272	9.654	10.459
10.562	9.799	8.346	6.493	4.824	3.522	2.635	2.105
1.634	1.282	1.699	3.213	3.209	2.052	1.363	1.504
1.279	0.489	-0.080	-0.219	-0.084	-0.325	-0.367	1.578
4.508	5.335	3.962	0.960	-2.019	-4.502	-6.107	-6.791
-7.271	-6.756	-6.582	-6.805	-6.604	-5.890	-4.672	-3.084
-1.636	-0.641	-0.557	-1.108	-0.781	0.251	0.291	-0.594
-1.378	-1.400	-0.284	1.661	3.998	5.875	5.842	3.684
0.227	-2.154	-2.602	-2.397	-2.148	-2.327	-2.884	-4.351
-6.236	-6.736	-6.962	-7.553	-7.296	-6.031	-4.187	-2.151
-0.784	-0.444	-1.219	-2.999	-5.394	-6.995	-7.313	-6.125
-3.587	-1.522	-0.871	-1.955	-4.220	-5.362	-5.112	-3.536
-1.900	-1.658	-2.685	-3.662	-3.646	-2.109	0.713	3.936
6.652	7.990	7.843	6.216	4.411	2.148	-1.048	-4.455
-5.373	-2.528	0.470	1.962	1.559	0.641	0.287	-0.160
-0.204	1.207	2.678	3.297	2.722	0.910	-1.125	-2.586
-3.640	-5.190	-6.945	-7.386	-6.476	-6.352	-7.708	-9.090
-9.418	-8.557	-7.110	-6.097	-5.574	-5.597	-6.322	-7.591
-8.249	-7.948	-7.078	-5.583	-4.203	-3.907	-3.703	-2.472
-1.322	-1.968	-4.335	-5.831	-5.273	-2.903	-1.620	-2.954
-4.807	-5.761	-5.640	-4.740	-4.317	-4.774	-6.173	-7.615
-7.895	-7.051	-5.517	-4.497	-4.471	-5.506	-6.359	-6.501
-6.290	-6.080	-5.388	-4.754	-4.021	-3.693	-3.957	-4.873
-6.468	-8.798	-10.983	-11.814	-10.920	-8.169	-4.591	-2.128
-1.594	-2.333	-1.378	0.998	1.551	-0.482	-3.466	-5.284
-5.116	-2.313	-0.133	-2.411	-4.322	-4.603	-3.052	0.125
2.643	3.443	2.094	-1.265	-5.687	-9.042	-10.885	-11.070
-9.553	-6.414	-1.847	3.794	8.323	10.628	10.315	7.957
7.387	9.637	12.106	11.859	10.756	10.989	13.077	16.314
18.734	19.513	18.606	17.246	15.665	14.523	16.544	20.692
23.374	23.637	21.633	19.733	18.518	18.031	17.211	14.511
11.050	8.257	7.239	7.717	9.080	10.858	11.760	10.939
9.635	9.974	12.130	13.929	14.515	13.908	13.212	12.582
12.495	13.013	13.515	13.754	14.067	15.160	15.945	15.436
13.556	12.350	12.347	11.981	10.824	9.004	7.619	6.951
6.258	5.318	4.694	3.859	2.578	1.252	0.235	-0.520

Appendix A: Sample VECTOR.EQR File

-1.062	-1.460	-1.611	-1.622	-1.974	-2.841	-3.619	-4.095
-4.204	-4.579	-5.476	-5.570	-5.065	-5.285	-6.295	-7.388
-7.508	-6.702	-6.068	-6.017	-6.341	-6.452	-6.775	-7.519
-7.641	-6.793	-5.910	-5.886	-6.751	-7.525	-7.793	-7.496
-6.671	-5.971	-5.840	-6.264	-6.707	-6.559	-6.520	-6.561
-6.310	-5.428	-4.669	-4.895	-5.026	-4.305	-2.806	-1.207
-0.102	0.419	0.420	0.125	-0.352	-1.074	-1.905	-2.507
-2.316	-1.411	-0.436	0.676	1.887	2.475	2.396	1.488
0.183	-0.756	-1.131	-1.222	-1.036	-0.600	-0.220	-0.176
-0.555	-1.362	-2.576	-4.009	-4.799	-4.510	-3.383	-2.442
-1.910	-1.741	-1.545	-1.559	-1.754	-1.814	-2.022	-2.447
-2.988	-3.274	-3.218	-3.029	-2.575	-1.787	-0.971	-0.638
-0.351	0.301	0.830	0.889	0.783	0.447	-0.432	-1.070
-1.190	-0.749	-0.246	-0.197	-0.498	-0.916	-1.315	-1.358
-0.949	-0.648	-0.786	-1.306	-1.865	-2.272	-2.159	-1.762
-1.714	-1.944	-2.225	-2.430	-2.667	-3.262	-3.892	-4.204
-4.342	-4.351	-4.192	-3.875	-3.303	-2.980	-3.073	-3.407
-3.913	-4.423	-4.843	-4.880	-4.475	-3.673	-2.746	-2.083
-1.774	-1.624	-1.407	-1.331	-1.534	-1.941	-2.114	-1.846
-1.544	-1.553	-1.620	-1.531	-1.554	-1.942	-1.644	-3.206
-3.544	-3.632	-3.420	-3.018	-2.491	-1.959	-1.717	-1.795
-2.092	-2.515	-2.805	-2.640	-2.300	-2.066	-1.831	-1.799
-1.942	-2.226	-2.452	-2.637	-2.667	-2.372	-1.968	-1.527
-1.262	-1.183	-1.038	-0.789	-0.622	-0.855	-1.247	-1.387
-1.161	-0.768	-0.539	-0.420	-0.367	-0.556	-1.027	-1.421
-1.483	-1.363	-1.158	-0.944	-0.902	-1.131	-1.570	-1.870
-1.750	-1.210	-0.573	-0.082	0.182	0.135	0.027	-0.011
0.003	-0.020	-0.018	0.083	0.179	0.141	-0.055	-0.335
-0.606	-0.842	-1.137	-1.556	-2.006	-2.365	-2.643	-2.974
-3.403	-3.909	-4.380	-4.779	-4.955	-4.702	-4.177	-3.693
-3.326	-3.156	-3.205	-3.078	-2.611	-2.147	-1.836	-1.590
-1.237	-0.814	-0.385	0.099	0.557	0.876	1.010	1.033
1.297	1.865	2.400	2.789	3.038	3.291	3.593	3.948
4.184	4.235	4.192	4.068	4.015	4.053	4.088	4.018
3.824	3.581	3.473	3.593	3.798	3.941	4.081	4.178
4.170	4.073	3.755	3.368	3.163	3.186	3.323	3.411
3.498	3.601	3.601	3.513	3.450	3.427	3.330	3.145
3.070	3.289	3.795	4.311	4.650	4.929	5.206	5.266
5.060	4.747	4.425	4.121	3.746	3.279	2.728	2.200
1.763	1.290	0.677	0.007	-0.495	-0.742	-0.759	-0.665
-0.604	-0.639	-0.850	-1.087	-1.200	-1.143	-0.928	-0.714
-0.453	-0.142	0.130	0.341	0.521	0.771	1.030	1.222
1.454	1.772	2.096	2.389	2.486	2.342	2.043	1.663
1.350	1.201	1.207	1.300	1.528	1.865	2.147	2.309
2.309	2.205	2.013	1.827	1.833	2.121	2.496	2.900
3.285	3.634	4.013	4.468	5.011	5.593	6.077	6.454
6.726	6.881	6.973	7.098	7.161	7.141	7.082	7.054
7.079	7.082	6.989	6.849	6.638	6.314	5.891	5.452
5.012	4.478	3.832	3.214	2.822	2.706	2.791	2.910
2.990	3.108	3.301	3.553	3.742	3.758	3.624	3.384
3.052	2.716	2.413	2.031	1.513	0.977	0.551	0.256
-0.016	-0.331	-0.541	-0.655	-0.884	-1.292	-1.791	-2.176
-2.271	-2.091	-1.884	-1.790	-1.791	-1.811	-1.940	-2.118
-2.288	-2.458	-2.726	-3.091	-3.415	-3.708	-4.118	-4.654
-5.282	-5.980	-6.684	-7.336	-7.837	-8.122	-8.230	-8.224
-8.115	-8.014	-8.022	-8.219	-8.491	-8.685	-8.828	-8.915
-8.872	-8.690	-8.417	-8.162	-7.859	-7.464	-7.038	-6.736
-6.599	-6.549	-6.510	-6.525	-6.618	-6.806	-7.047	-7.185
-7.181	-7.138	-7.119	-7.099	-7.066	-7.053	-7.064	-6.972
-6.760	-6.506	-6.248	-6.054	-5.924	-5.769	-5.552	-5.270
-4.946	-4.559	-4.185	-3.847	-3.425	-2.922	-2.485	-2.106
-1.738	-1.381	-1.001	-0.590	-0.220	0.090	0.363	0.623
0.852	0.926	0.862	0.730	0.618	0.591	0.726	1.124
1.717	2.406	3.038	3.480	3.815	4.119	4.361	4.494
4.606	4.681	4.745	4.830	4.823	4.685	4.455	4.148
3.793	3.450	3.131	2.791	2.449	2.241	2.165	2.149
2.144	2.188	2.185	2.094	1.898	1.664	1.467	1.374
1.461	1.645	1.929	2.322	2.703	3.051	3.448	3.706
3.739	3.638	3.580	3.487	3.269	3.073	2.945	2.843
2.684	2.448	2.230	2.071	1.900	1.761	1.698	1.719
1.762	1.777	1.737	1.665	1.569	1.423	1.193	0.853
0.462	0.210	-0.297	-1.163	-2.030	-2.552	-2.529	-2.249

Appendix A: Sample VECTOR.EQR File

-2.087	-2.287	-2.875	-3.438	-3.807	-4.119	-4.145	-3.681
-3.457	-3.706	-3.722	-3.358	-2.865	-2.613	-2.422	-2.132
-1.780	-1.280	-0.927	-0.802	-0.656	-0.473	-0.330	-0.157
0.125	0.437	0.773	1.205	1.641	1.958	2.039	1.935
1.948	1.912	1.732	1.520	1.330	1.189	1.071	1.105
1.038	0.883	0.758	0.702	0.717	0.803	0.899	0.939
0.996	1.089	1.266	1.546	1.804	1.981	2.077	2.174
2.215	2.002	1.521	1.199	1.181	1.140	0.947	0.759
0.611							
535 POINTS OF DISPL DATA EQUALLY SPACED AT 0.10 SEC. (UNITS: CM)							
1.350	2.579	3.857	5.214	6.682	8.187	9.720	11.340
13.006	14.284	15.056	15.318	15.287	15.391	15.043	13.618
11.234	8.948	7.764	7.320	5.703	2.729	-0.847	-4.005
-7.481	-10.399	-13.155	-15.873	-18.068	-19.461	-19.783	-19.519
-18.645	-17.619	-16.205	-14.277	-12.141	-10.821	-9.992	-9.466
-9.220	-9.078	-8.960	-9.277	-8.811	-7.581	-6.347	-6.235
-6.889	-7.115	-6.698	-5.857	-4.770	-3.253	-1.047	0.874
2.090	3.405	4.534	5.239	5.831	6.310	6.918	7.536
8.080	8.331	8.724	9.146	9.543	9.824	9.714	9.960
9.729	9.113	9.079	9.294	9.121	8.673	8.506	8.389
7.967	6.985	5.486	3.749	1.943	-0.496	-2.763	-4.162
-5.380	-6.602	-7.448	-7.434	-6.633	-5.397	-3.984	-2.763
-2.021	-1.453	-1.278	-1.096	-0.770	-1.017	-1.432	-1.484
-1.350	-1.321	-1.481	-1.880	-2.247	-2.446	-2.574	-2.309
-2.601	-3.772	-4.832	-4.310	-2.169	0.576	2.474	4.256
5.075	5.063	4.325	3.218	2.528	2.049	1.378	0.707
0.253	0.206	0.220	0.096	0.086	-0.085	-0.309	-0.772
-1.433	-2.153	-2.551	-2.890	-3.258	-3.655	-3.472	-3.309
-3.347	-3.320	-3.466	-3.580	-3.688	-4.157	-4.671	-4.888
-5.306	-5.775	-6.023	-5.965	-5.624	-5.135	-4.499	-3.410
-2.285	-1.509	-0.692	0.506	1.652	2.508	3.502	4.597
5.366	5.775	6.023	6.316	6.627	6.835	6.919	6.968
7.171	7.703	7.652	7.104	6.480	5.877	5.451	5.366
5.342	5.410	5.363	5.452	6.020	6.338	6.156	5.979
5.619	4.986	4.312	3.987	3.957	3.626	2.965	2.677
2.517	2.037	1.813	1.547	1.342	1.811	2.591	2.977
2.664	2.487	2.647	2.664	2.756	3.048	2.991	2.570
1.859	1.154	0.227	-0.486	-1.075	-1.862	-2.606	-3.064
-3.367	-3.681	-4.221	-4.496	-5.069	-5.585	-6.214	-7.011
-7.548	-8.164	-8.849	-9.427	-9.869	-10.464	-11.545	-12.599
-12.964	-13.205	-13.169	-13.605	-13.968	-14.310	-14.692	-14.506
-14.640	-15.635	-16.671	-16.772	-15.881	-15.135	-14.098	-13.065
-11.596	-9.765	-8.229	-6.515	-4.271	-2.387	-0.745	0.235
0.942	1.964	2.916	4.097	5.434	6.637	7.908	9.293
10.764	11.957	12.998	13.699	14.204	14.500	14.502	14.341
14.126	13.782	13.318	12.744	12.168	11.407	10.740	10.076
9.320	8.634	7.954	7.170	6.525	5.873	5.202	4.591
4.093	3.705	3.651	3.679	3.586	3.357	3.293	3.493
3.665	3.601	3.510	3.501	3.395	3.009	2.663	2.498
2.365	2.206	1.954	1.674	1.512	1.495	1.596	1.703
1.662	1.645	1.646	1.569	1.538	1.445	1.284	1.157
0.974	0.689	0.316	-0.034	-0.289	-0.584	-0.992	-1.352
-1.522	-1.616	-1.706	-1.841	-1.935	-2.029	-2.211	-2.500
-2.747	-2.879	-3.021	-3.228	-3.378	-3.503	-3.681	-3.876
-3.982	-4.038	-4.052	-4.120	-4.153	-4.141	-4.178	-4.269
-4.320	-4.399	-4.521	-4.523	-4.465	-4.422	-4.380	-4.327
-4.314	-4.372	-4.525	-4.767	-5.113	-5.563	-5.966	-6.274
-6.556	-6.745	-6.856	-6.866	-6.774	-6.648	-6.412	-6.094
-5.703	-5.279	-4.871	-4.467	-4.102	-3.742	-3.343	-2.939
-2.599	-2.289	-1.954	-1.614	-1.289	-0.994	-0.647	-0.200
0.297	0.736	1.091	1.321	1.429	1.387	1.280	1.180
1.043	0.898	0.808	0.784	0.814	0.900	1.058	1.255
1.386	1.463	1.566	1.735	1.909	2.047	2.239	2.527
2.915	3.435	4.049	4.701	5.365	6.023	6.674	7.279
7.787	8.169	8.418	8.653	8.915	9.228	9.546	9.799
9.966	10.012	9.974	9.878	9.717	9.464	9.231	9.015
8.772	8.485	8.119	7.662	7.050	6.280	5.434	4.600
3.757	2.864	1.957	1.103	0.330	-0.358	-1.025	-1.705
-2.429	-3.151	-3.867	-4.573	-5.235	-5.839	-6.400	-6.882
-7.272	-7.549	-7.727	-7.808	-7.798	-7.719	-7.621	-7.544
-7.423	-7.150	-6.760	-6.301	-5.815	-5.310	-4.837	-4.440
-4.127	-3.875	-3.631	-3.386	-3.183	-3.008	-2.788	-2.475

Appendix A: Sample VECTOR.EQR File

-2.081	-1.682	-1.314	-0.985	-0.697	-0.456	-0.245	-0.030
0.176	0.351	0.449	0.433	0.239	0.056	-0.177	-0.524
-0.866	-1.192	-1.454	-1.648	-1.751	-1.788	-1.786	-1.727
-1.579	-1.349	-1.118	-0.916	-0.758	-0.617	-0.504	-0.397
-0.272	-0.127	0.076	0.314	0.543	0.693	0.818	

End of File.