FORMWORKS-PLUS: IMPROVED PRE-PROCESSOR FOR VECTOR ANALYSIS SOFTWARE

by

Vahid Sadeghian

A thesis submitted in conformity with the requirements

for the degree of Master of Applied Science

Graduate Department of Civil Engineering

University of Toronto

FORMWORKS-PLUS: IMPROVED PRE-PROCESSOR FOR VECTOR ANALYSIS SOFTWARE

Vahid Sadeghian

Master of Applied Science

Department of Civil Engineering

University of Toronto

2012

Abstract

VecTor© is a suite of computer programs developed for nonlinear finite element analysis of reinforced concrete. A graphics-based pre-processor (FormWorks) was developed for 2D concrete membrane structures, greatly contributing to the software's utility and success. However, modeling other types of structures is a time consuming process, requiring manual definition of the finite element mesh, loads and analysis parameters in standard text files. A user-friendly pre-processor is required for the entire suite of programs if they are to be of greater use to design engineers.

The purpose of this study is to develop an updated version of FormWorks, FormWorks 3.5, which is more user-friendly and compatible with the improvements made in VecTor2 over the past ten years. In addition, an extended version of FormWorks, FormWorks-Plus, will be created for the remaining VecTor programs with a wide range of viewing features, and facilities for specification of node coordinates, elements, loads and material properties.

Acknowledgements

My greatest appreciation goes to my supervisor, Professor Frank J. Vecchio, for all his support, expert advice, patience, and understanding throughout my graduate studies. I have felt privileged to be his student and immensely enjoyed working with him. I would also like to thank Professor Evan C. Bentz for his valuable advice while reviewing this thesis.

I would like to thank to my friends and colleagues who have helped me throughout my Masters degree: Trevor Hrynyk, Fady El Mohandes, Seong-Cheol Lee, Luca Facconi, Serhan Güner, Heather Trommels, and David Carnovale.

I would like to acknowledge the generous financial support that I received from NSERC, LEA Consulting Ltd., the Department of Civil Engineering, and Professor Vecchio.

Last but not least, I thank my father, mother and brother for their constant help, support and encouragement.

Table of Contents

List of Tables

List of Figures

Chapter 4: Modeling 3D Solid Structures

Chapter 5: Modeling 3D Shell Structures

Chapter 6: Modeling 2D Plane Frame Structures

Notation

$1, 2$	= principal axis
a_g	= maximum aggregate size
A_s	= reinforcement ratio
b	= width of the specimen
c	= cement volume
c_c	= thermal expansion coefficient for concrete
c_s	= thermal expansion coefficient for steel
C_{inc}	= cyclic load factor increment
$CMOD_j$	= crack mouth opening displacement at j
d	= distance from top of the section
D_b	= diameter of reinforcing bar
D_c	= thickness of concrete layer
D_f	= fibre diameter
E_c	= initial tangent stiffness of concrete
E_l	= wood elastic modulus for the longitudinal direction
E_{mx}, E_{my}	= masonry initial elastic modulus in x and y directions
E_s	= initial tangent stiffness of reinforcement
E_t	= wood elastic modulus for the transverse direction
f_c'	= concrete cylinder uniaxial compressive strength
f_{c-l}'	= wood compressive strength in the longitudinal direction
f_{c-t}'	= wood compressive strength in the transverse direction
f_{mx}, f_{my}	= masonry compressive strength in x and y directions
$f_{R,j}$	= residual flexural tensile strength corresponding with $\mathrm{CMOD_j}$
f_t'	= uniaxial cracking strength of concrete
f_{t-l}'	= wood tensile strength in the longitudinal direction
f_{t-t}'	= wood tensile strength in the transverse direction
f_u	= ultimate strength of reinforcement
f_{unl}	= unloading stress
f_y	= yield strength of reinforcement
F_j	= load corresponding with $\mathrm{CMOD_j}$
$Fr1k, Fr3k$	= fibre residual flexural strength

F_u	= fibre tensile strength
G_f	= concrete fracture energy
h_{sp}	= distance between the notch tip and the top of the specimen
k, l, m	= direction cosines
k_c	= thermal diffusivity
L	= span length
L_f	= fibre length
LF_f	= final load factor
LF_i	= initial load factor
LS_{inc}	= load factor increment
N_x	= number of concrete layers
OS	= centerline offset
R	= number of repetitions
$Rho - t$	= ratio of the transverse reinforcement
$Rho - z$	= ratio of the out-of-plane reinforcement
S	= average crack spacing
T	= thickness
T_s	= thickness of one steel plate
T_u	= fibre bond strength
v_{lt}	= longitudinal shear strength of the wood
V_f	= fibre volume fraction
w	= water volume
W_c	= width of the cross section
x, y	= global axis
x', y'	= orthotropy axis
Y_s	= location of the reinforcement layer from the top of the cross section
α_0	= a constant equal to 1 for rounded aggregates and 1.44 for crushed aggregates
α	= direction of the bed joints with respect to x axis
β	= angle between the axes x' and 1
θ	= angle between the axes x and 1
$\Delta\epsilon_p$	= elastic strain offset of the reinforcement relative to the unstrained concrete
ε_0	= concrete compressive strain, corresponding to f'c
ε_m	= maximum strain

ε_p = strain offset

$\varepsilon_{r1}, \varepsilon_{r2}$ = reference strains

ε_{sh} = reinforcement strain hardening strain

ε_u = reinforcement ultimate strain

μ = shear strength reduction factor for masonry structures

υ_0 = initial Poisson's ratio

υ_{lt} = initial Poisson's ratio for longitudinal stress-transverse strain

υ_{tl} = initial Poisson's ratio for transverse stress-longitudinal strain

ψ_c = creep coefficient of concrete

ψ_s = relaxation coefficient for the pre-stressing steel

Chapter 1

Introduction and Program Structure

1.1 Introduction

Satisfying performance-based design codes, designing complex infrastructure, and analyzing structural behavior under extreme load conditions all have combined to create a need for nonlinear finite element programs to analyze reinforced concrete structures. The VecTor© suite of programs has been developed at the University of Toronto to analyze different types of structures such as beams (VecTor1), 2D membrane structures (VecTor2), 3D solid structures (VecTor3), plates and shells (VecTor4), plane frames (VecTor5), and axisymmetric solids (VecTor6). Several experimental programs with different types of specimens have been undertaken at the University of Toronto and elsewhere to verify the accuracy of the programs. In addition, analyzing real-world structures including frames, slabs, shear walls, silos, bridges, offshore platforms, crash barriers and nuclear containment structures have been demonstrated the value of the VecTor programs in determining the complex nonlinear behaviour of concrete structures.

A user-friendly pre-processor is required for the entire suite of programs if they are to be of greater use to design engineers. A pre-processor would aid in creating appropriate structural models, inputting and checking data, selecting proper analysis parameters, and specifying appropriate loads. In addition, its graphing capabilities would allow the user to see the structure from different views, cut various sections and permit a wide range of plots to demonstrate the structure shape, material specifications and applied loads. A graphics-based pre-processor (FormWorks) was developed for program VecTor2 by Peter Wong (2002), greatly contributing to the software's utility and success. However, the remaining VecTor programs do not have pre-processors and they run in a DOS environment from Fortran executable files; standard text editors are used to create input data files.

This master thesis consists of two main parts. The first part describes a new version of FormWorks, **FormWorks 3.5**, that is compatible with the improvements made in VecTor2 over the past ten years. The second part describes an extended version of FormWorks, **FormWorks-Plus**, that can be used with the remaining VecTor programs and is capable of modeling different types of structures.

The first objective of this thesis was to improve the previous version of FormWorks to support different types of materials. Formwork 3.5 (a further developed version of FormWorks 2.0) is compatible with

VecTor2 program and allows the user to select from a wide range of material types. Table 1-1 shows new features of FormWorks 3.5:

<table>
<tr><th rowspan="2">FormWorks Version</th><th colspan="2">Material Types</th></tr>
<tr><th>Type</th><th>Components</th></tr>
<tr><td rowspan="3">FormWorks 2.0</td><td>Reinforced Concrete</td><td>Ductile Steel Reinforcement</td></tr>
<tr><td>Reinforcement</td><td>---</td></tr>
<tr><td>Bond</td><td>---</td></tr>
<tr><td rowspan="21">FormWorks 3.5</td><td rowspan="9">Reinforced Concrete</td><td>Ductile Steel Reinforcement</td></tr>
<tr><td>Prestressing Steel</td></tr>
<tr><td>Tension Only Reinforcement</td></tr>
<tr><td>Compression Only Reinforcement</td></tr>
<tr><td>External Bonded FRP Fabric</td></tr>
<tr><td>Steel-Fibre-Hooked</td></tr>
<tr><td>Steel-Fibre-Straight</td></tr>
<tr><td>Shape Memory Alloy Type 1</td></tr>
<tr><td>Shape Memory Alloy Type 2</td></tr>
<tr><td>Structural Steel</td><td>---</td></tr>
<tr><td>Masonry</td><td>---</td></tr>
<tr><td>Wood (fixed orthotropic)</td><td>---</td></tr>
<tr><td>Concrete-Steel Laminate</td><td>Steel Skin Plate</td></tr>
<tr><td rowspan="2">Concrete-SFRC Laminate</td><td>SFRC Laminate-Hooked Fibre</td></tr>
<tr><td>SFRC Laminate-Straight Fibre</td></tr>
<tr><td rowspan="2">Masonry-SFRC Laminate</td><td>SFRC Laminate-Hooked Fibre</td></tr>
<tr><td>SFRC Laminate-Straight Fibre</td></tr>
<tr><td>Concrete-Ortho Laminate</td><td>Orthotropic Laminate</td></tr>
<tr><td>Reinforcement</td><td>---</td></tr>
<tr><td>Bond</td><td>---</td></tr>
</table>

Table 1-1 – Comparing features of FormWorks 2.0 and FormWorks 3.5

Note: all the new features of FormWorks 3.5 are also available in FormWorks-Plus.

The second objective of this thesis was to develop FormWorks to support the remaining VecTor programs. FormWorks 2.0, developed by Peter Wong in 2002, is a 2D program in the XY plane which only supports elements, loads and material properties associated with the VecTor2 program. The new extended edition of FormWorks, FormWorks-Plus, allows the user to view 3D structures in different planes, and take sections at any location; it has the feature of “quick section” to locate nodes and create section at those locations automatically. Previous versions of FormWorks only used the Microsoft Foundation Classes (MFC) of Microsoft Visual C++ program. These classes cannot provide advanced graphical utilities for drawing 3D elements and structures. However, FormWorks-Plus is supported by a program called 3D view which uses applications to draw complex three-dimensional shapes. In addition, new types of elements, loads, and material properties were added to FormWorks-Plus to make it compatible with the remaining VecTor programs. The following tables compare the features of FormWorks-Plus with previous versions:

FormWorks Version	Viewing Features
FormWorks 2.0	XY plane
FormWorks-Plus	1. XY, XZ, ZY planes 2. Sectional view 3. Projection view 4. Supported by 3D view program

Table 1-2 – Viewing features of FormWorks 2.0 and FormWorks-Plus

FormWorks Version	VecTor Type	Element Types
FormWorks 2.0	VT2:	Rectangular Triangular Quadrilateral Truss Link Interface
FormWorks-Plus	VT2:	Rectangular Triangular Quadrilateral Truss Link Interface
	VT3:	Hexahedral Wedge Truss
	VT4:	9 Noded Shell Truss
	VT5:	Frame Element
	VT6:	4-Node Axisymmetric 3-Node Axisymmetric Truss Ring bars

Table 1-3 – Element types supported by FormWorks 2.0 and FormWorks-Plus

Before the features of the new versions of FormWorks can be explained in detail, it is necessary to become familiar with the structure of program. The next part of this chapter explains the methods of programming that were used in writing and developing FormWorks.

1.2 Structure of Program

1.2.1 Background Information

As discussed previously, FormWorks is a pre-processor software that generates input files for the VecTor suite of nonlinear finite element analysis programs for reinforced concrete structures. The role of FormWorks is to provide the user an interface for generating, visualizing and checking finite element models.

The FormWorks program was written in the C++ programming language using Microsoft Foundation Classes, and compiled with Microsoft Visual C++ Version 6.0. The Microsoft Foundation Classes (MFC) are a set of predefined classes upon which Windows programming with Visual C++ is built. These classes represent an object-oriented approach (which is a method of using "objects" – data structures consisting of data fields and methods together with their interactions – to design applications and computer programs) to Windows programming that encapsulates the Windows API. (All of the communications between any Windows application and Windows itself uses the Windows Application Programming Interface, otherwise known as the Windows API). The process of writing a Windows program involves creating and using MFC objects, or objects of classes derived from MFC. The objects of these MFC-based class types incorporate member functions for communicating with Windows, for processing Windows messages, and for sending messages to each other. These derived classes inherit all of the members of their base classes. These inherited functions do practically all of the general routine work necessary for a Windows application to work (Horton, 2008).

FormWorks or any other Windows program written using MFC has a fundamental class name CWinApp. An object of this class includes everything necessary for starting, initializing, running and closing the application. FormWorks needs a window as the interface to the user, referred to as a frame window. The MFC class CFrameWnd is designed specifically for this purpose (Prosise, 1999).

The structure of the FormWorks application consist of two main classes — a document and a view. A document is the name given to the collection of data in the application with which the user interacts. Handling application data in this way enables standard mechanisms to be provided within MFC for managing a collection of application data as a unit. These mechanisms are inherited by the document class from the base class defined in the MFC library, so one gets a broad range of functionality built-in to the application automatically, without having to write any code. A view always relates to a particular document object. A document contains a set of application data in the program, and a view is an object that provides a mechanism for displaying some or all of the data stored in a document. It defines how the data is to be displayed in a window and how the user can interact with it (Horton, 2008). The following simple example shows that each view displays the data that the document contains in a different form:

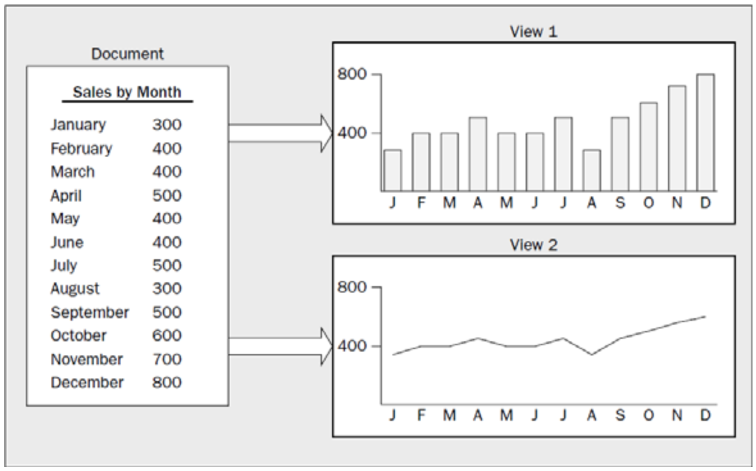


Figure 1-1 – Simple example to show Document and View definitions (Horton, 2008).

MFC incorporates a mechanism for integrating a document with its views, and each frame window with a currently active view. A document object automatically maintains a list of pointers to its associated views, and a view object has a data member holding a pointer to the document that it relates to. Each frame window stores a pointer to the currently active view object. Figure 1-2 shows graphical representation of these interrelationships (Prosise, 1999).

In MFC programming, one has a choice as to whether the program deals with just one document at a time or with several. The **Single Document Interface** is supported by the MFC library for programs that only require one document to be open at a time. A program using this interface is referred to as an **SDI** application. For programs needing several documents to be open at one time, like FormWorks, one must use the **Multiple Document Interface**, which is usually referred to as **MDI**. With the MDI, as well as being able to open multiple documents of one type, the program can also be organized to handle documents of different types simultaneously with each document displayed in its own window (Horton, 2008).

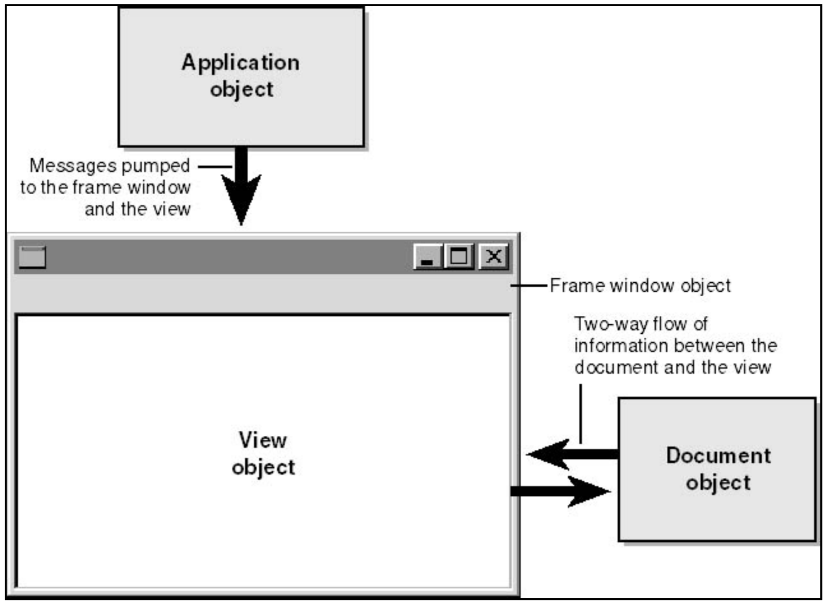


Figure 1-2 – Graphical representation of Document, View and Frame Window interrelationships (Prosise, 1999).

Previous versions of FormWorks used the MFC library to draw structures. MFC encapsulates the Windows interface to the screen and printer and relieves the user of the need to worry about much of the detail involved in programming graphical output. To display 2D structures with simple elements, this method worked sufficiently well; for 3D structures with complex elements shapes, the program should be supported with more advanced graphical tools. FormWorks-Plus uses a supporting program named **3D View** which is written with the MFC library and **OpenGL (Open Graphics Library)**. OpenGL is a standard specification defining a cross-platform for writing applications that produce 2D and 3D computer graphics. The interface consists of over 250 different function calls which can be used to draw complex three-dimensional scenes from the simplest geometric objects that the system can handle (Segal and Akeley, 2006).

1.2.2 FormWorks Architecture

The FormWorks-Plus application consists of a total of 331 files; 311 files are Visual C++ project files which should be copied directly into a project folder, and 17 files are resource files that must be placed in a folder named “res” which is located in the project folder. The 3 remaining files are input files which provide initial information to run the program.

FormWorks-Plus consists of 140 classes which were created either manually or via the creation of resources such as dialog boxes, property sheets, property pages, and menus. Following the C++ convention, each class is generally described by two files: a header file with the *.h extension and a source file with the *.cpp extension. The class names are the same as their corresponding header and source file names, with the ‘C’ prefix.

Header files contain definitions of functions and variables which can be incorporated into any class by using the pre-processor #include statement. The source files are ‘implementation’ files for the class. They contain the source code that implements the member functions.

As discussed before, each Windows Application has a number of core classes which are created by Visual C++ AppWizard. The core classes for FormWorks can be summarized in Table 1-4.

FormWorks Application Core Classes		
	Class Name	Definition
1	CPr1App: CWinApp	The base class from which a Windows application object is derived.
2	CMainFrame : CMDIFrameWnd	Provides the functionality of a Windows multiple document interface (MDI) frame window, along with members for managing the window.
3	CChildFrame: CMDIChildWnd	Provides the functionality of a Windows multiple document interface (MDI) child window, along with members for managing the window.
4	CPr1Doc : Cdocument	Provides the basic functionality for user-defined document classes.
5	CPr1View : CScrollView	Provides the basic functionality for user-defined view classes with scrolling capabilities.

Table 1-4 – FormWorks application core classes

Note: the project name for FormWorks application is **Pr1**.

The Document/View structure of FormWorks application contains instances of several major classes: CJobData, CStructureData, CLoadData, CAttributeData, CWMultiPolygon and CBandWidth. Each of these classes contains instances of smaller classes or utilizes them in data structures. Additionally, the CPr1Doc class contains the **Serialize()** member function to save data. The view class CPr1View, among other purposes, contains functions for drawing to the screen, printing and interacting with the mouse. The flow chart in Figure 1-3 summarizes the program structure.

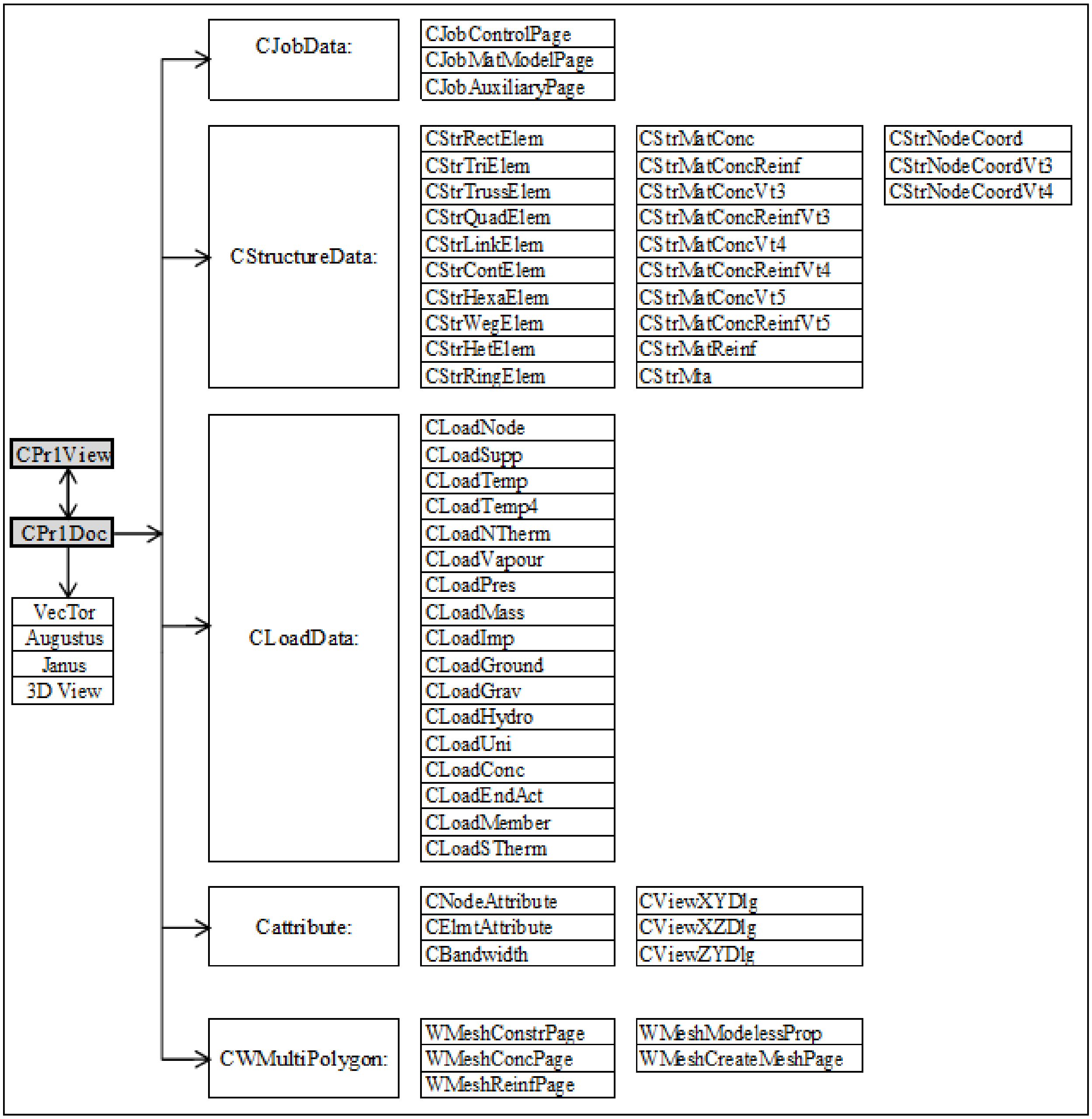


Figure 1-3 – FormWorks Source Code Structure

The **CJobData** class contains all the member variables and functions required to store and generate the Job Data for one VecTor analysis. This class consists of three main classes:

1. CJobControlPage: Manages different load cases and stores Analysis Parameters.
2. CJobMatModelPage: Contains all the member variables related to Concrete Models, Reinforcement Models, Bond Models and Analysis Models.

3. CJobAuxiliaryPage: Contains all the member variables related to Dynamic Analysis, Tension Softening, Masonry Structures and Material Resistance Factors.

Member functions are provided to interface with the Job Data dialog classes.

The **CStructureData** class contains all the member variables and functions required to store and generate the Structure Data for one VecTor analysis. This class consists of three main parts and each part consists of several classes:

1. Element part: Contains all the information to store and generate elements. FormWorks-Plus is able to generate 10 different types of elements.
2. Material part: Contains all the information to define material properties and assign them to elements. Previous versions of FormWorks had three material types: Reinforced Concrete, Reinforcement, and Bond which worked only with the VecTor2 program. FormWorks-Plus not only contains more material types for VecTor2 (see Table 1-5 for detailed information about material types), but also allows the user to define material properties related to other VecTor programs and assign them to elements.
3. Node Coordinate part: Contains all the information to define and store Node Coordinates. Since each VecTor program has its own definition of nodes, FormWorks-Plus uses four different types of node coordinate definitions to support all the VecTor programs. The First type is for defining 2D node coordinates for 2D programs which include VecTor2, VecTor5 and VecTor6. The second type is able to generate 3D node coordinates for VecTor3. The third and fourth types are for VecTor4; one of them allows the user to define top and bottom coordinates and the other is for defining centerline coordinates.

The CStructureData class contains instances of other classes as elements of CList class. For example, to access the information for hexahedral elements, CStructureData defines m_ListStrHexaElem member variable from CList class whose elements are CStrHexaElem objects.

The **CLoadData** class contains all the member variables and functions required to store and generate the Load Case Data for one load case in a VecTor analysis. The CPr1Doc class contains five instances of the CLoadData class, one for each load case. CLoadData class contains instances of other classes as elements of CList class. For example, to define and store a uniform load, CLoadData has a CList member variable called m_ListLoadUni whose elements are CLoadUni objects. FormWorks-Plus allows the user to define and apply 6 types of nodal loads and 11 types of element loads.

FormWorks Version	VecTor Type	Load and Material Types
FW 2.0	VT2:	**2D Nodal loads:** Joint Loads; Support Displacements; Lumped Mass; Impulse Forces **2DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration **Material Types:** Reinforced Concrete; Reinforcement; Bond
FW-Plus	VT2:	**2D Nodal loads:** Joint Loads; Support Displacements; Nodal Temperature; Lumped Mass; Impulse Forces **2DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration **Material Types:** Reinforced Concrete; Reinforcement; Bond; Masonry; Wood; Structural Steel; Concrete Steel Laminate; Concrete SFRC Laminate; Masonry SFRC Laminate; Concrete Ortho Laminate
	VT3:	**3D Nodal loads:** Joint Loads; Support Displacements; Nodal Temperature; Lumped Mass; Impulse Forces; Vapour Pressure **3DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration **Material Types:** Reinforced Concrete; Reinforcement; Bond
	VT4:	**3D Nodal loads:** Joint Loads; Support Displacements; Lumped Mass; Impulse Forces **3DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration; Hydrostatic Loads; Uniform Loads **Material Types:** Reinforced Concrete; Reinforcement
	VT5:	**2D Nodal loads:** Joint Loads; Support Displacements; Lumped Mass; Impulse Forces **2DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration; End Action Loads; Concentrated Loads; Uniform loads **Material Types:** Reinforced Concrete (allows the user to define different types of layers)
	VT6:	**2D Nodal loads:** Joint Loads; Support Displacements; Nodal Temperature; Impulse Forces **2DElement Loads:** Gravity Loads; Element Temperature; Concrete Prestrain; Ground Acceleration **Material Types:** Reinforced Concrete; Reinforcement

Table 1-5 – Load and material types supported by FormWorks 2.0 and FormWorks-Plus

The **CList** is a data structure that is commonly used by many of the classes. This data structure permits dynamic allocation so that the size of the list need not be predefined. The CList data structure utilizes POSITION type variables to iterate through the list. However the CList data structure does not permit random access in the manner of an array.

One of the useful features of FormWorks is that by selecting each node or element, the user can see all the properties that assigned to that node or element such as: loads, material properties, and coordinates. The **CAttributeData** class stores all this information. In addition, this class utilizes instances of other classes to draw nodes and elements. The CAttributeData class consists of two main classes:

1. CElmtAttribute: Contains information of each element and allows the user to draw elements and see their properties.
2. CNodeAttribute: Contains information of each node and allows the user to draw nodes.

The m_ListElmtAttribute member variable stores information about each defined element using the CElmtAttribute class. The m_ListNodeAttribute member variable stores information about each defined node by utilizing the CNodeAttribute class. These lists are initialized by the member function CAttributeData::InitializeAllAttributes().

The **CWMultiPolygon** class stores information for the automatic mesh generation. It contains the member variables necessary for defining the meshing boundaries and parameters and the member functions to generate the mesh. The CWMultiPolygon class makes use of following classes in CList member variables.

1. CWPolygon: Describes a single polygon region.
2. CWPolygonVoid: Describes a single polygon void.
3. CWLine: Describes a line path.
4. CWPoint: Describes a single point.

CWPolygon, CWPolygonVoid and CWLine all use lists of CWPoint objects to define the boundaries and line paths. The class is called CWMultiPolygon because the mesh generation proceeds from a CList of CWPolygon objects. FormWorks accepts mesh generation input in theCWPolygon, CWPolygonVoid, CWLine and CPoint lists. The voids, lines and points are then assigned to lists in the enclosing polygon object(s).

As discussed above, in the FormWorks source code, core classes utilize instances of other minor classes. Each of these minor classes has a dialog class which is used to create Dialog Box that permits the user to input data. When viewing the source code in the Visual C++ "ClassView", it is apparent that the names of the dialog classes are similar to those of user defined classes; 'Dlg' is added to the end of the dialog class name. These pairs of classes maybe thought of as companion classes. The member variables in the dialog class often parallel those of the companion user defined class (Wong, 2002).

There are two different types of dialog class in FormWorks: **modal dialogs** and **modeless dialogs**. They work in completely different ways. While a modal dialog remains in effect, all operations in the other windows in the application are suspended until the dialog box is closed, usually by clicking an OK or Cancel button. With a modeless dialog, the user can move the focus back and forth between the dialog box and other windows in the application just by clicking them, and the dialog box can be used at any time until the user close it. The Node Coordinates window (CStrNodeCoordDlg) is an example of a modeless dialog; the Define Job window is modal.

Different versions of FormWorks use different file extensions to save and open files. Previous versions of FormWorks stored data as a *.fws file. FormWorks 3.5 saves files with *.fwk extension and finally FormWorks-Plus uses *.fwp extension. Using different file extensions makes it easier for the user to recognize which version of FormWorks was used to create the file.

The method that FormWorks source code uses to store and retrieve data is called **serialization**. It is the process of writing or reading an object to or from a persistent storage medium. The basic idea of serialization is that an object should be able to write its current state, usually indicated by the value of its member variables, to persistent storage. Later, the object can be re-created by reading, or **deserializing**, the object's state from the storage (Horton, 2008).

MFC uses an object of the **CArchive** class as an intermediary between the object to be serialized and the storage medium. This object is always associated with a **CFile** object, from which it obtains the necessary information for serialization, including the file name and whether the requested operation is a read or write.

In addition to the Visual C++ project files, FormWorks needs two other types of files to compile. The first type are **resource files,** and they must be placed in a folder named "res". Resource files contain a collection of icons, menus, dialog boxes and toolbars. The second type are **input text files,** needed to initialize and run the FormWorks; they should be placed in the folder where FormWorks is located.

1. JobOpt file: Contains information to initialize the JobControlPage, JobModelPage and JobAuxiliaryPage

2. StrOpt file: Contains information of material reference types and maximum allowable number of elements, nodes and material types.
3. LoadOpt file: Contains maximum allowable number of nodal loads and element loads.

Chapter 2

The Basics of FormWorks

The FormWorks graphical user interface is used to model and display the structure intended for analysis. This chapter introduces some of the basic concepts of the graphical user interface. The exact configuration of the FormWorks screen elements may vary with the operating system and display hardware.

2.1 Type of Structure

After starting the program, a window automatically pops up requiring the user to specify the type of structure that is going to be modeled. By selecting the type of structure, FormWorks-Plus activates the related VecTor program. This implies that all the material types, loads, node types and element types which are related to that VecTor program will be activated. The following figure and table show the starting window and different structure types. The default value for Structure Type is option 2, Plane Membrane (2D). FormWorks 3.5 supports modeling of 2D Plane membranes only.

Note that it is possible for the user to change the Structure Type in the next steps. In the Job Control page there is an option to change Structure Type which will be discussed in Job Data section.

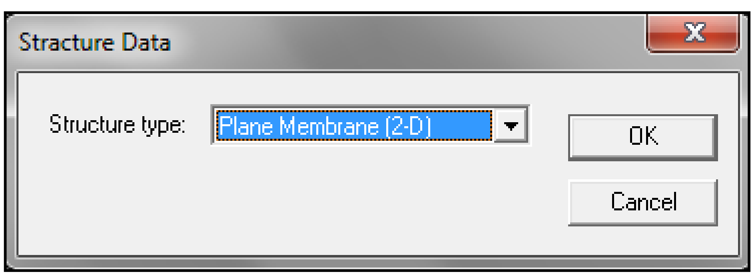


Figure 2-1 –Structure Data window

	Structure Data Window	
	Structure type	VecTor program
1	Plane Membrane (2D)	VecTor 2
2	Solid (3D)	VecTor 3
3	Shell	VecTor 4
4	Plane Frame (2D)	VecTor 5
5	Axisymmetric Solid	VecTor 6
6	Mixed type	Unavailable

Table 2-1 – Structure Types

2.2 The FormWorks Screen

After selecting the type of structure, the FormWorks graphical user interface appears on the screen and looks similar to Figure 2-2. The various parts of the interface are labelled in the figure and are described as follows.

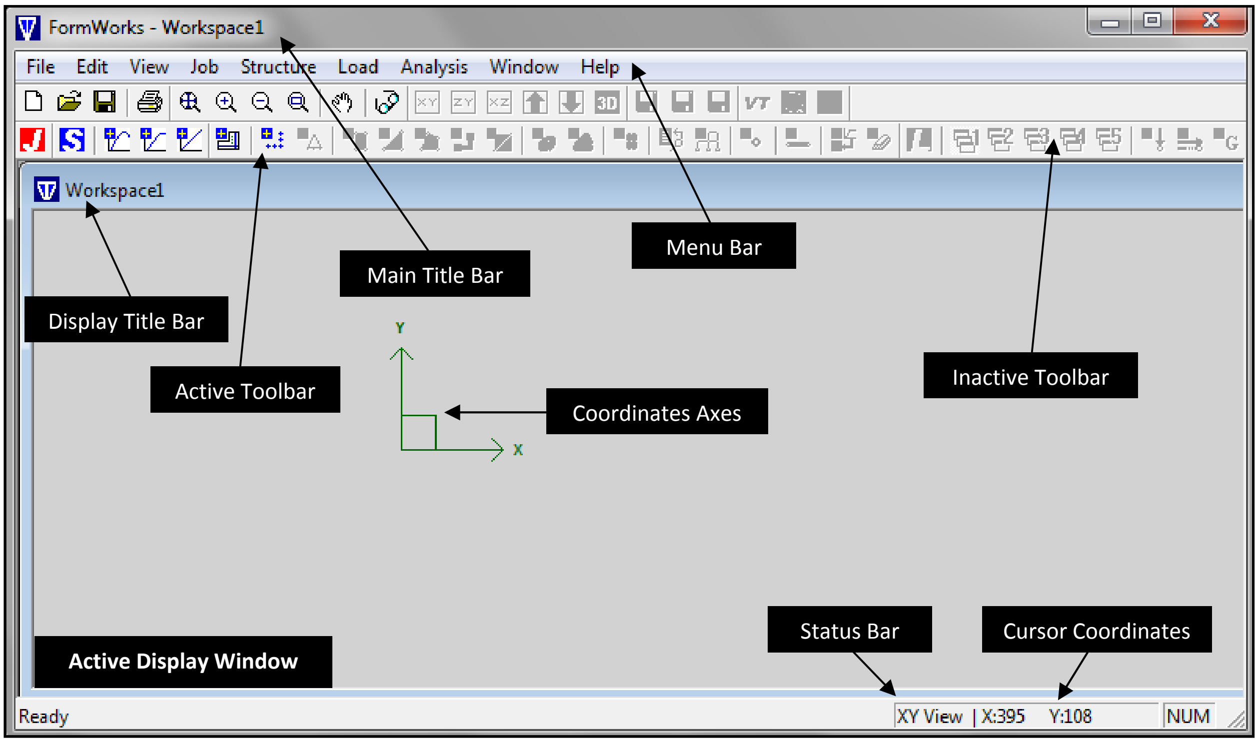


Figure 2-2 – FormWorks graphical user interface

2.2.1 Menu Bar

The menus on the Menu Bar contain almost all of the operations that can be performed using FormWorks. Those operations are called menu commands, or simply commands. Each menu corresponds to a basic type of operation. The operations are described later in this chapter.

2.2.2 Toolbar

The buttons on the toolbars provide quick access to many commonly used operations. If one holds the mouse cursor on one of these buttons, a "tool tip" will pop up showing the function of the button, as shown in Figure 2-3. A status bar appears at the bottom of the FormWorks window. A prompt on the left side of the status bar describes the function of menu items and toolbar buttons as the mouse pointer lingers over them.

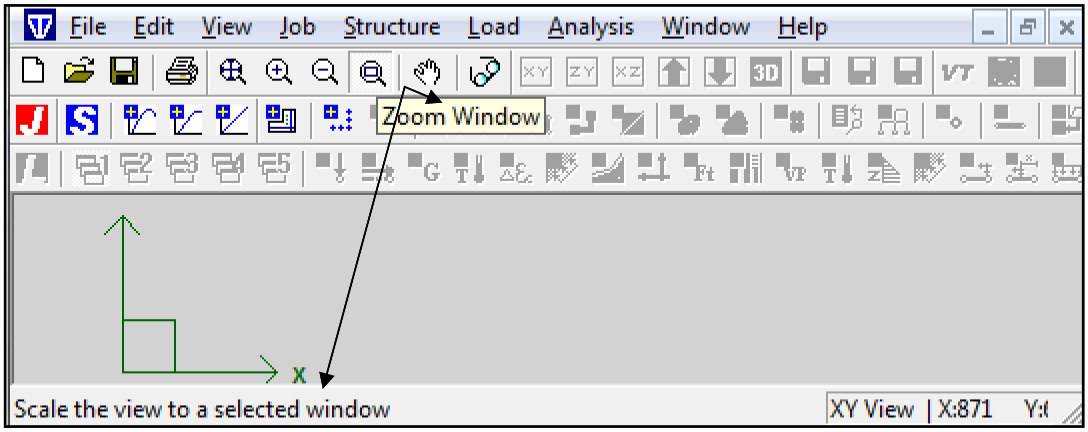


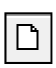
Figure 2-3 – The status bar shows the description of menu item or toolbar button

One can move the toolbars around to any of the four sides of the main window, or have them float over the display windows by dragging them to the desired location. The buttons on the toolbars can be chosen by clicking the down arrow and selecting the buttons. One can use these methods to create custom toolbars of frequently used operations. Some buttons of the toolbar appear greyed-out and become enabled as the finite element model proceeds.

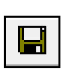
The toolbars in FormWorks program can be categorized into several types:

2.2.2.1 Main Toolbar

Consists of New, Open, Save and Print options.

- New 

  This icon allows the user to create a new document. By default, the name of the document is "Workspace" and the name reflects the order of their creation. For instance, three Workspace child windows, Workspace1, Workspace2 and Workspace3 appear in Figure 2-4. The FormWorks window is titled FormWorks-Workspace3, indicating that the third Workspace is currently active.

- Save 

  It is advisable to regularly save the Workspace for backup and later retrieval as the finite element model progresses. A user can select the **File/Save** menu item. Alternatively, the user can click the

Save toolbar button to activate the **Save As** dialog box. By saving the file, a new Workspace file is created in the specified directory. The extension of the file depends on the version of the FormWorks. Previous versions of FormWorks used the *.fws extension; FormWorks 3.5 uses the *.fwk extension and FormWorks-Plus uses the *.fwp extension to save Workspace files.

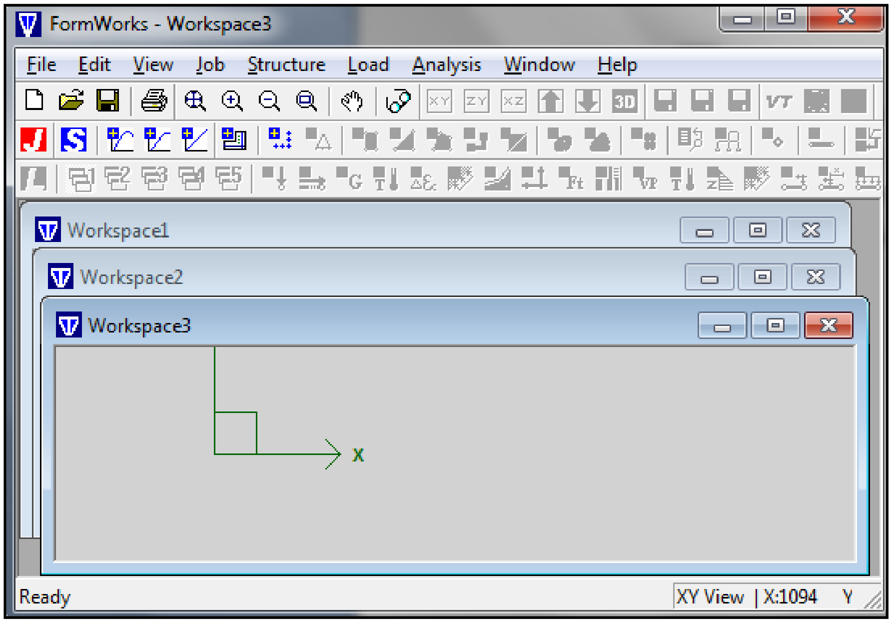


Figure 2-4 – Three Workspace child windows: Workspace1, Workspace2 and Workspace3 are created and the third Workspace is currently active

- Open 

This option is used to open saved Workspace files. One can select the **File/Open** menu item, or click the **Open** toolbar button, to activate the **Open** dialog box.

- Print 

FormWorks allows the finite element model to be printed with a standard printer. The entire finite element model is scaled to fit the selected page format and printed with the same attributes that are shown in the Workspace view (Wong, 2002).

To print the finite element model, one must complete the following procedure:

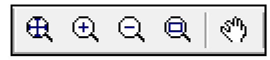
1. Select the **File/Print Setup**… menu item. Select the desired paper properties and click **Ok**

2. Select the **File/Print Preview** menu item to preview the finite element model.

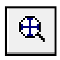
3. Select the **File/Print** menu item or click the **Print** toolbar button.

4. Click **Ok** to print the Workspace.

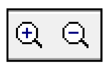
2.2.2.2 Zooming Toolbar



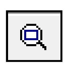
Consists of Zoom All, Zoom In, Zoom Out, Zoom Window and Pan icons.

- **Zoom All** 

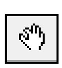
Select **View/Zoom/Zoom All** menu item, or click the **Zoom All** button, to activate this option. Zoom All allows the user to see the entire structural model in the workspace view.

- **Zoom In and Zoom Out** 

Select **View/Zoom/Zoom In** (or **Zoom Out**) menu item, or click the **Zoom In** (or **Zoom Out**) button, to activate these options. One can zoom-in to see more detail, or zoom-out to see more of the structure. Zooming in and out is performed with10% increments.

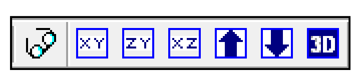
- **Zoom Window** 

Select **View/Zoom/Zoom Window** menu item, or click the **Zoom Window** button, to activate this option. The user can zoom in on a part of the structure using the mouse by dragging a window around the area of interest while holding down the left mouse button.

- **Pan** 

Select **View/Pan** menu item, or click the **Pan** button, to activate this option. Panning allows the structure to be moved dynamically around the Display Window by holding down the left mouse button while dragging the mouse in the window.

2.2.2.3 View Toolbar



Consists of Display Options, XY View, XZ View, ZY View, Quick Section and 3D View options.

- **Display Options** 

The display options hides or reveal attributes of the finite element model in the Workspace view. Select the **View/Display Options** menu item or click the **Display Options** toolbar button to display the Display Options dialog box shown in Figure 2-5.

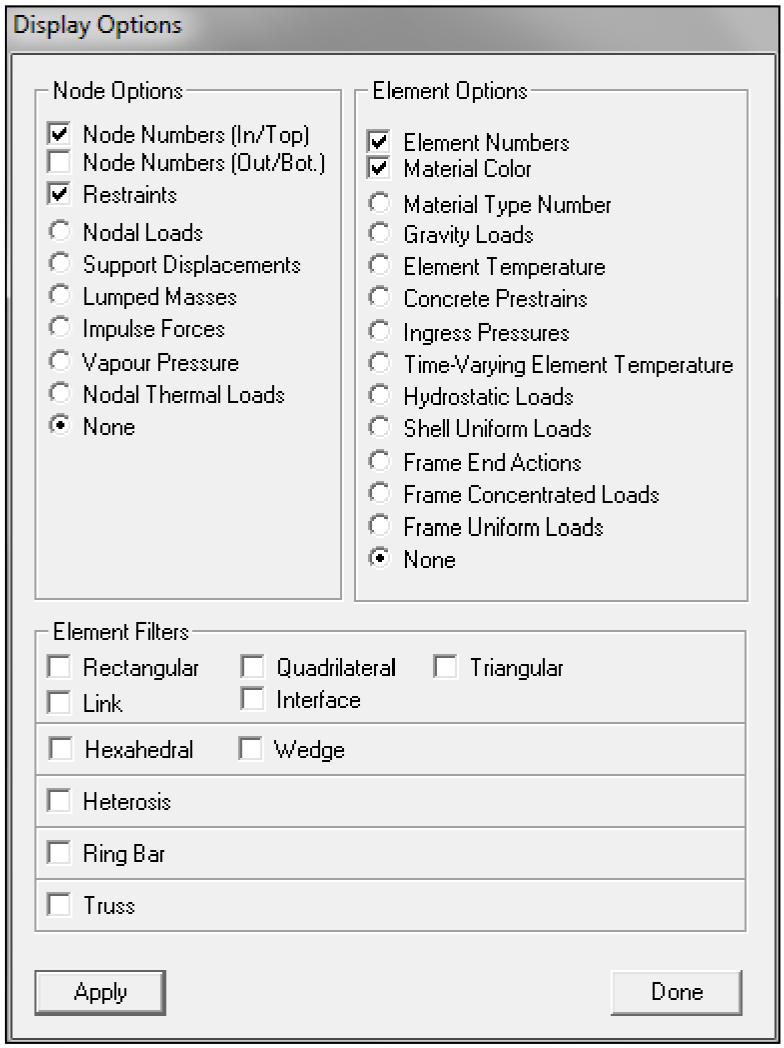


Figure 2-5 –Display Options window

Node Options:

- Node Numbers (In/Top):

  Check to reveal the node number beside each node of the finite element model. Note that for the VecTor4 program, this option reveals the In (or Top) node numbers (see Figure 5-7) of the structure.

- Node Numbers (Out/Bot.):

  This option appears greyed-out and is disabled for all VecTor programs except VecTor4. For VecTor4 program, it should be checked to reveal the Out (or Bottom) node number beside each node of the finite element model.

- Restraints:

  Check to reveal the support restraints on each node.

- Nodal Loads:

  Select to reveal applied nodal forces for the current load case.

- Support Displacements:

  Select to reveal imposed displacements for the current load case.

- Lumped Masses:

  Select to reveal Lumped Masses for dynamic analysis for the current load case.

- Impulse Forces:

  Select to reveal time-varying forces for the current load case.

- Vapour pressure:

  Select to reveal applied vapour pressures for the current load case.

- Nodal Thermal Loads:

  Select to reveal nodal thermal loads for the current load case.

- None:

  Select to hide the above load types for the current load case.

Element Options:

- Element Number:

  Check to reveal the element number in the center of each element of the finite element model.

- Material Colour:

  Check to display elements with the colour of the assigned material type or default colour.
  Uncheck to view elements drawn in black and white.

- Material Type Number:
  Select to reveal the material type labels in the center of each element.

- Gravity Loads:
  Select to reveal the density in kg/m^3 for G-forces applied to concrete elements in the load case.

- Element Temperature:
  Select to reveal the temperature in degrees Celsius for concrete and reinforcement elements in the current load case.

- Concrete Prestrains:
  Select to reveal the elastic strain offset in millistrain applied to concrete elements in the load case.

- Ingress Pressure:
  Select to reveal ingress pressures in MPa, applied to concrete elements in the load case.

- Time-Varying Element Temperature:
  Select to reveal the temperature gradients in degrees Celsius for shell or frame elements in the current load case.

- Hydrostatic Loads:
  Select to reveal the hydrostatic pressure in MPa for shell elements in the current load case.

- Shell Uniform Loads:
  Select to reveal the uniformly distributed loads in MPa for shell elements in the current load case.

- Element End Action:
  Select to reveal applied axial forces, shears and moments at the ends of frame elements in the current load case.

- Frame Concentrated Loads:
  Select to reveal applied axial force, shear and moment on frame elements in the current load case.

- Frame Uniform Loads:
  Select to reveal imposed uniformly distributed loads on frame elements in the current load case.

- None:

  Select to hide the above load types for the current load case.

Elements Filters:

The Element Filters reveals or hides element types or makes them ineligible for mouse selection.

- Heterosis, Hexahedral, Wedge, Rectangular, Quadrilateral and Triangle:

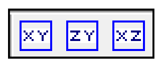
  Check to hide the element attributes, but not the element itself, and make the elements ineligible for mouse selection.

- Frame, Truss, Link, Interface and Ring Bars:

  Check to hide the elements and their attributes, and make the elements ineligible for mouse selection.

Note: If the structure type is Plane Frame (2D) then automatically the Truss option changes to Frame option.

- **XY View, XZ View and ZY View**



This facility allows the user to view the finite element model from different planes. Select the **View/View/Set XY Plane** (or other planes) menu item, or click the **Set XY Plane** (or other planes) toolbar button, to display the dialog box shown in Figure 2-6.

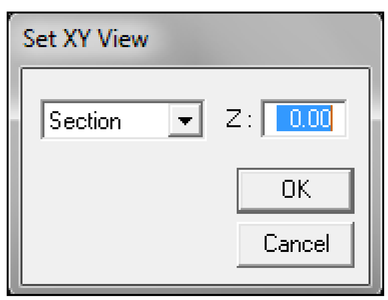
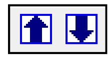


Figure 2-6 –Set XY View window

For each plane, there are two viewing options to select. The first option is a sectional view which allows making a section in a specified coordinate. For instance, to make a section in XY view one must specify the Z-coordinate of the section and then click the OK button. The second option is to view the projection of the structure on different planes. This can be helpful when the user wants to view the change in material types or loads in entire finite element model. For each Viewing dialog box there are two projection views. One displays the projection of the structure to the right (top) side and the other shows the projection to the left (bottom) side. At the bottom of the main window, the status bar shows the active plane and the location of the section.

The default viewing option is the XY plane with Z equal to zero. Note that only in 3D models XY View, XZ View and ZY View options are enabled. For 2D models these options appear greyed-out and all the drawings are displayed in the XY plane.

- **Quick Section** 

With complex finite element models, finding the exact coordinates of the nodes and making a section at those points is not easy and takes time. This feature helps the user to automatically find the next (or previous) node and make a section at that location. Select the **View/View/Next Section** (or **Previous Section**) menu item or click the **Next Section** (or **Previous Section**) toolbar button to activate it. The Quick Section feature is only available with 3D models; in 2D models it appears greyed-out.

- **3D View** 

As discussed in the first chapter, since the graphical tools of MFC library cannot handle 3D shapes very well, FormWorks-Plus uses a supportive program named "3D View" which, in turn, uses OpenGL (an advance graphical tool) to display complex 3D models. Select the **View/View/Set 3D View** menu item or click the **Set 3D View** toolbar button to activate it. This item is only available in 3D models; in 2D models it appears greyed-out.

Important Note: 3D View program uses the "Job" and "Struct" files to communicate with FormWorks. Thus, before clicking on the 3D view button, the user must save or update the "Job" and "Struct" files.

Figure 2-7 shows different parts of 3D View program.

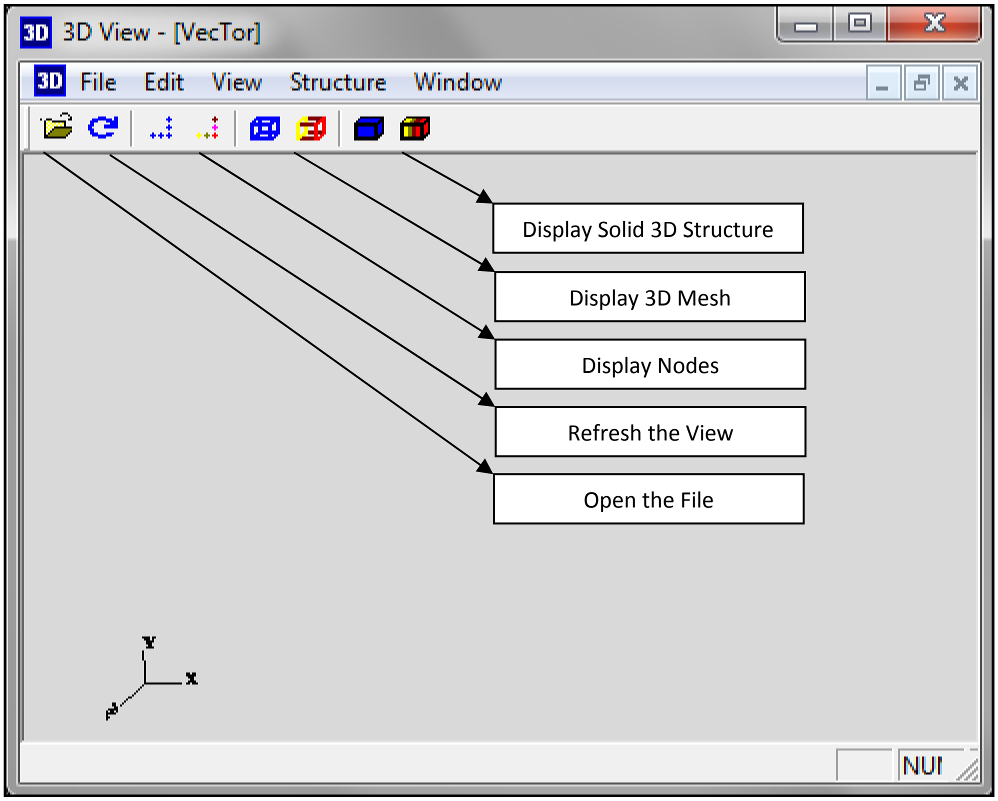


Figure 2-7 – 3D View program graphical user interface

The following figures show different viewing options in FormWorks-Plus:

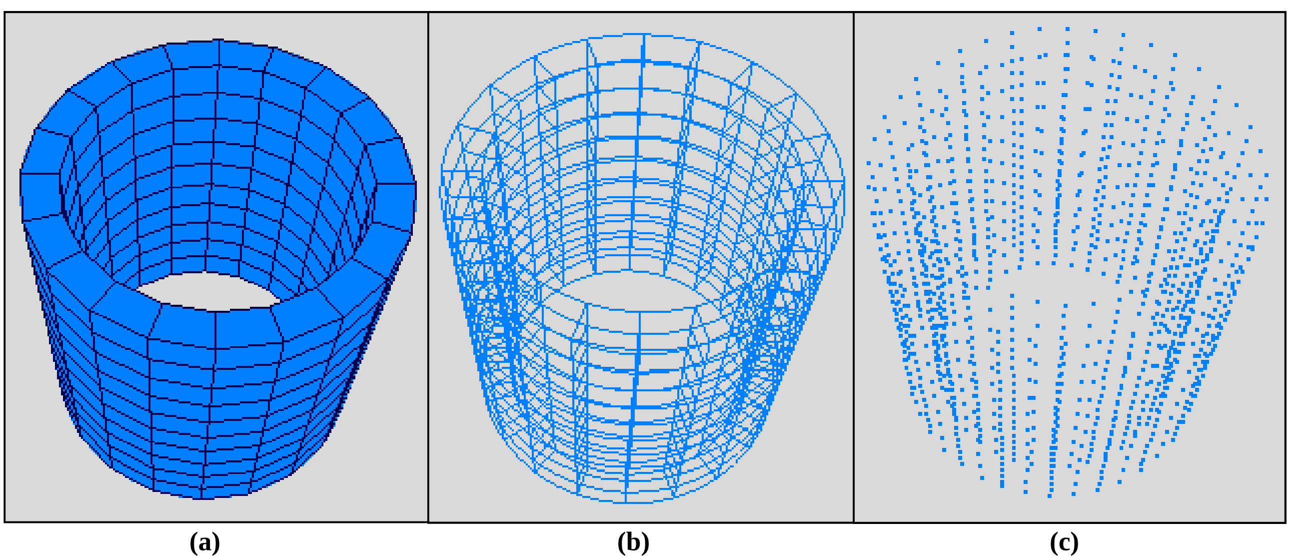


Figure 2-8 – 3D View program outputs: (a) solid view, (b) mesh view and (c) nodal view

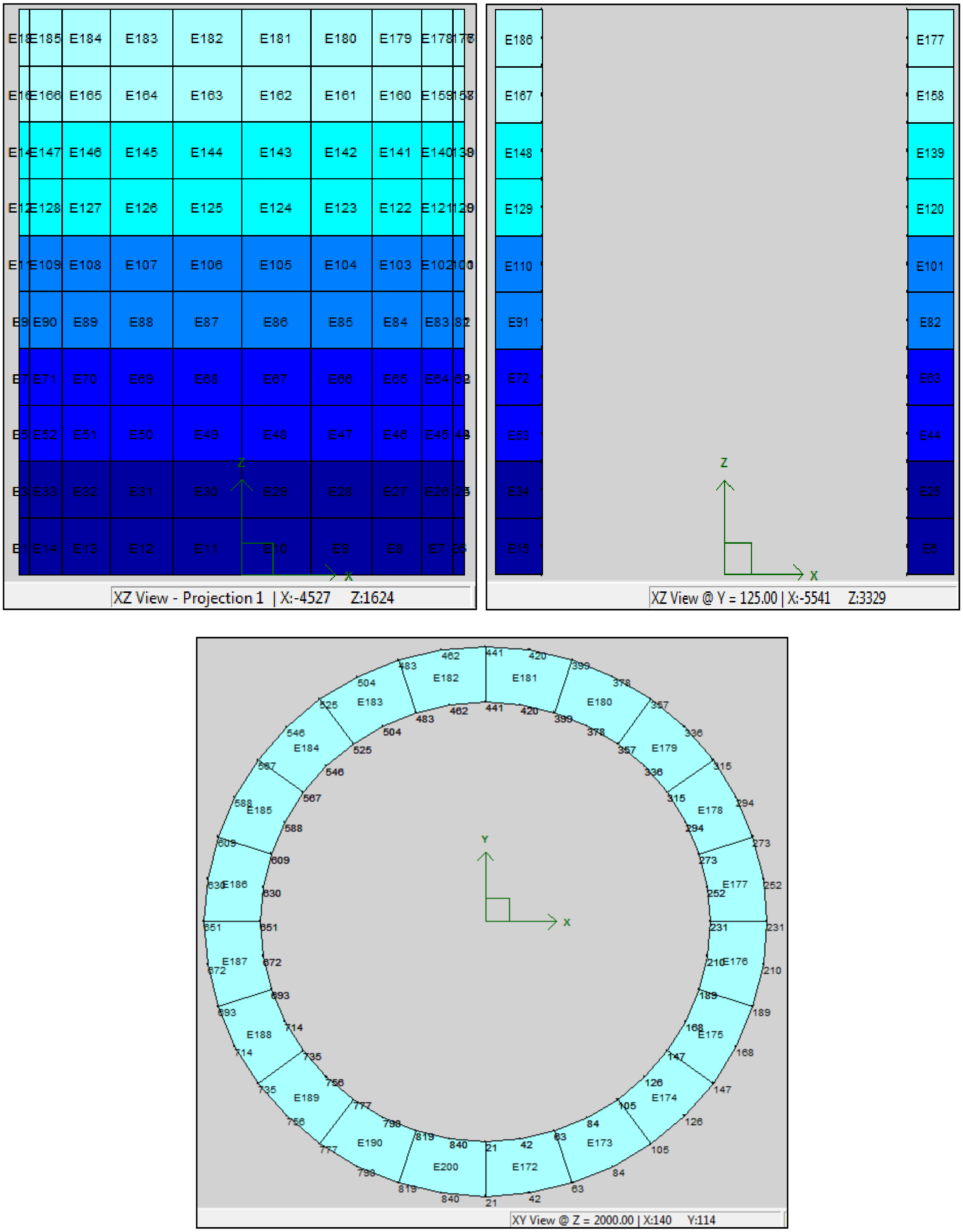
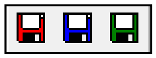


Figure 2-9 – Section and projection views in FormWorks-Plus

2.2.2.4 Save Toolbar 

To run a VecTor analysis, four input files are required. The save toolbar allows the user to generate these required input files. The first item is the **Save Job File** which stores the Job and Auxiliary data in two separate text files with the extensions of *.JOB and *.AUX. Note that to run a VecTor analysis, the job file's name must be "VecTor". The VecTor programs will not recognize any other name for the job file. The second and third items are the **Save Structural File** and **Save Load File** buttons. They save the structural and load information in two separate text files. The extension of the files depends on the type of VecTor program that is going to be used for analysis. For example, if the user is going to run a shell structure model in the VecTor4 program, FormWorks will save files with the extensions of *.S4R and *.L4R. Figure 2-10 shows how FormWorks generates input files for the VecTor programs.

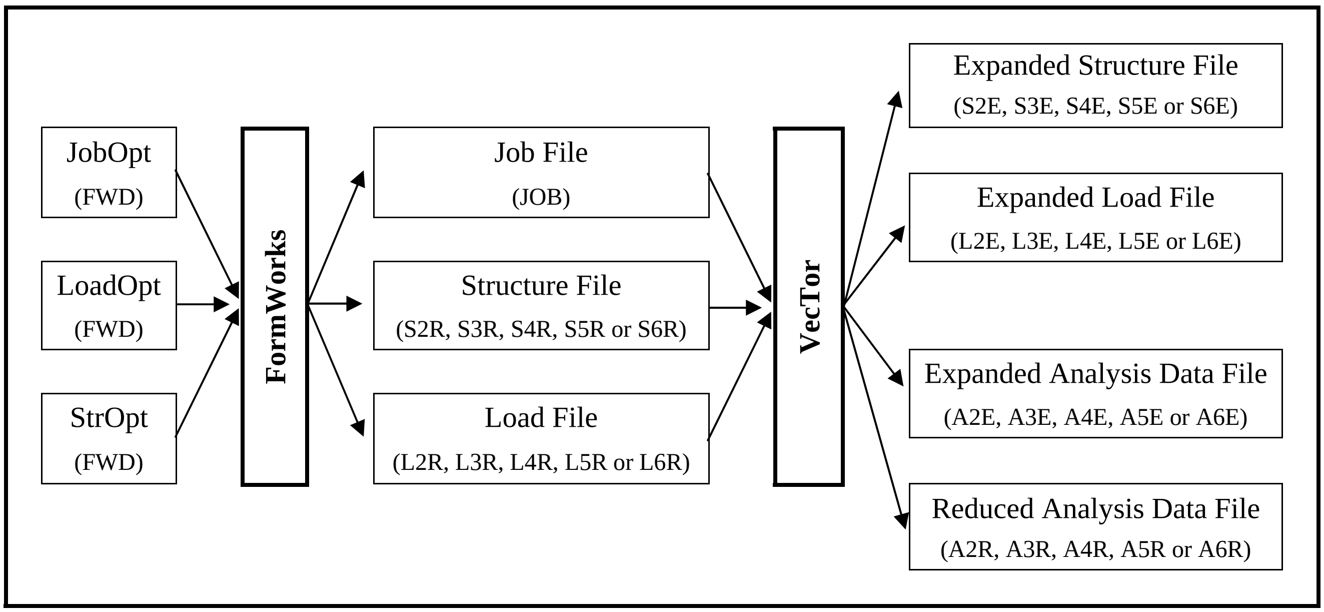
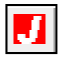


Figure 2-10 – Input and output files for FormWorks and VecTor

2.2.2.5 Analysis Toolbar 

The analysis toolbar allows FormWorks to communicate with other programs. The first item is the **Run VecTor Processor** button. Before selecting this item, the user must make sure all the input files required for a VecTor analysis have been saved or updated. The second and third items are the **Run Augustus Post-processor** and **Run Janus Post-processor** buttons. The post-processors aid in interpreting and visualizing analysis results. Augustus is an advanced 2D post-processor program which can open VecTor2 and VecTor6 output files. To visualize analysis results of the other VecTor programs the basic

Janus post-processor can be used. Janus was mainly developed for VecTor3 but most of its features are also compatible with VecTor4 and VecTor5.

2.2.2.6 Job Item 

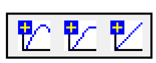
The Job item allows the user to define and store Job file's parameters. The Job window consists of three pages: Control page, Model page and Auxiliary page. The following chapters of this thesis will discuss all the member variables of Job window in detail. Select the **Job/Define Job** menu item or click the **Job** toolbar button to open Job dialog box.

2.2.2.7 Structural Toolbar

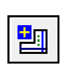
The Structural toolbar consists of several parts:

- **Structure Information** 

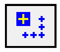
  Select the **Structure/Structure Information** menu item or click the **Structure Information** toolbar button to determine the number of defined materials types, nodes and elements currently defined in the model. The **Structure Information** dialog appears as shown in Figure 2-11. These values are updated as the model is constructed.

- **Material Properties** 

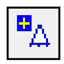
  Select the **Structure/Define Reinforced Concrete Materials** (or other types of material)menu item or click the **Define Reinforced Concrete Materials** (or other types of material)toolbar button to open related material properties page.

- **Define and Mesh Structure** 

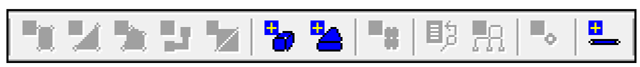
  Select the **Structure/Define and Mesh Structure** menu item, or click the **Define and Mesh Structure** toolbar button, to start the auto meshing process of model. Note that this feature is only available for 2D finite element models which will be run with VecTor2.

- **Create Nodes** 

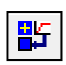
Select the **Structure/Create Nodes** menu item, or click the **Create Nodes** toolbar button, to define the nodes. As discussed before, FormWorks-Plus uses four different types of defining node coordinates to support all the VecTor programs. The first type is for defining 2D node coordinates for 2D programs which include VecTor2, VecTor5 and VecTor6. The second type is able to generate 3D node coordinates for VecTor3. The third and fourth types are designed for VecTor4; one allows the user to define top and bottom coordinates and the other is to define centerline coordinates.

- **Create Support Restraints** 

Select the **Structure/Create Support Restraints** menu item, or click the **Create Support Restraints** toolbar button, to define support restraints.

- **Create Elements** 

Select the **Structure/Create Elements** menu item, or click the **Create Elements** toolbar button, to define new elements. Depending on the type of structure, the types of elements which are irrelevant appear greyed-out.

- **Assign Material Types** 

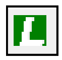
Select the **Structure/Assign Material Types** menu item, or click the **Assign Material Types** toolbar button, to assign material types to elements. Note that this item appears greyed-out if no material type or element has been defined.

- **Delete Structure** 

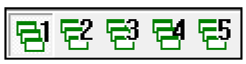
Select the **Structure/Delete Structure** menu item, or click the **Delete Structure** toolbar button, to delete an element. Note that this item appears greyed-out if no node or element has been defined.

2.2.2.8 Load Toolbar

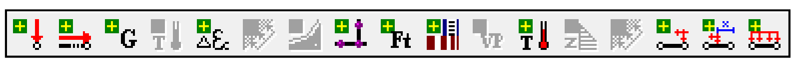
The Load toolbar consist of several parts:

- **Load Information** 

Select the **Load/Load Information** menu item, or click the **Load Information** toolbar button, to determine the number of nodal loads and element loads currently defined in the model for each load case. The **Load Information** dialog appears as shown in Figure 2-11. These values are updated as the model is constructed.

- **Load case** 

Before assigning any loads, select the **Load/Select Load Case** menu item, or click the **Load Case** tool bar button, to choose the load case to which the loads will be added. Only the load cases that are active in the Job Data may be selected.

- **Apply Loads** 

Select the **Load/Apply Nodal Loads** (or **Apply Element Loads**) menu item, or click the **Apply Loads** (or **Apply Element Loads**) toolbar button, to apply loads to nodes or elements. Depending on the type of structure being analyzed, the load types which are irrelevant appear greyed-out.

- **Delete Load** 

Select the **Load/Delete Load** menu item, or click the **Delete Load** toolbar button, to delete an applied load. This item appears grey-out when there is no applied load.

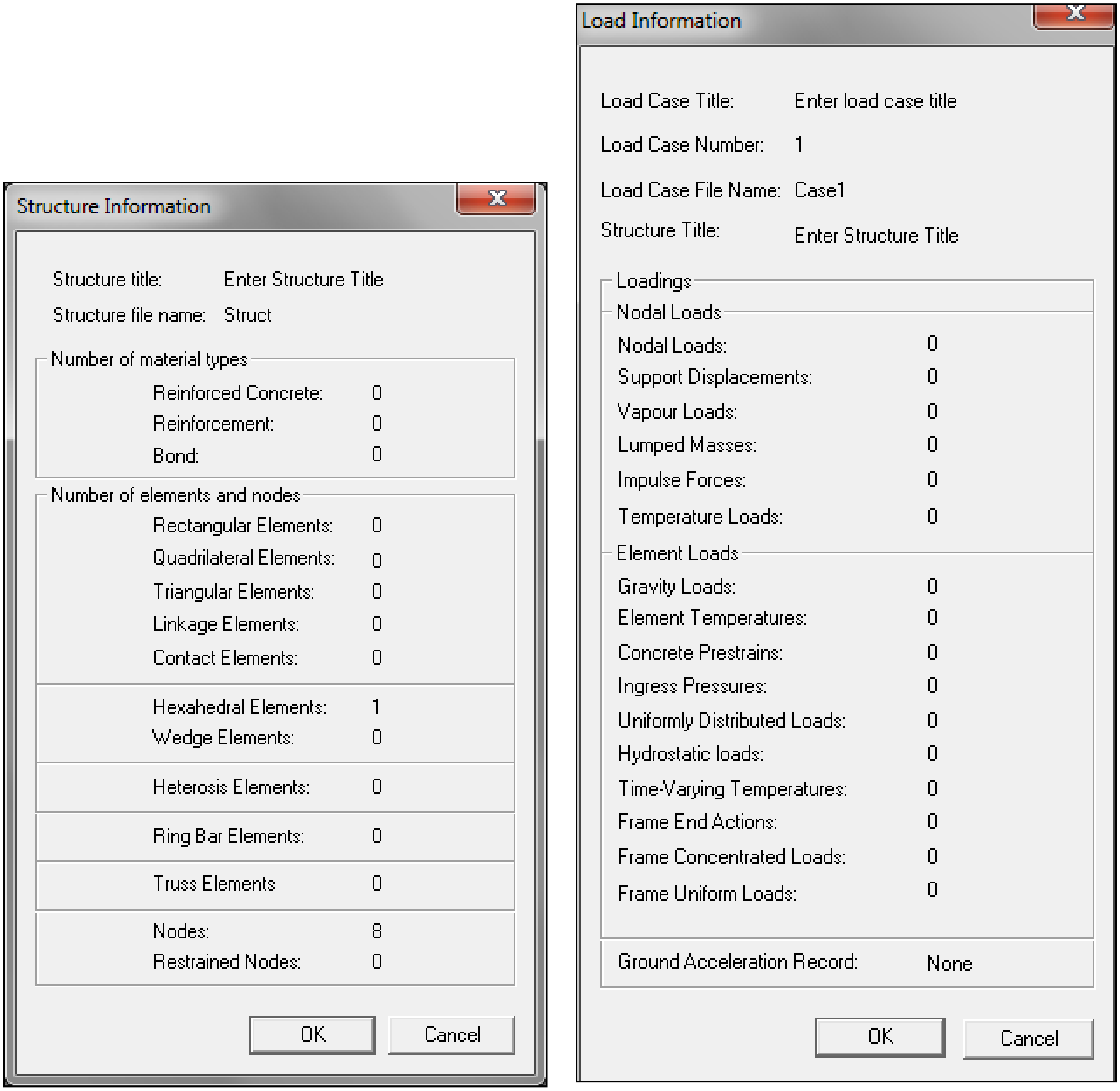


Figure 2-11 – Structure and Load information dialog boxes

2.2.3 Structure Limits and Load Limits

The VecTor programs have limitations for both the number of elements and the number of applied loads that can be modeled. Select the **Structure/Structure Limits** menu item to view limits on structural parameters such as: the number of material types, nodes, elements and a maximum bandwidth. Select the **Load/Load Limits menu** item to view limits on nodal and element loads in each load case. Figure 2-12 shows the Structure Limits and the Load Limits dialog boxes.

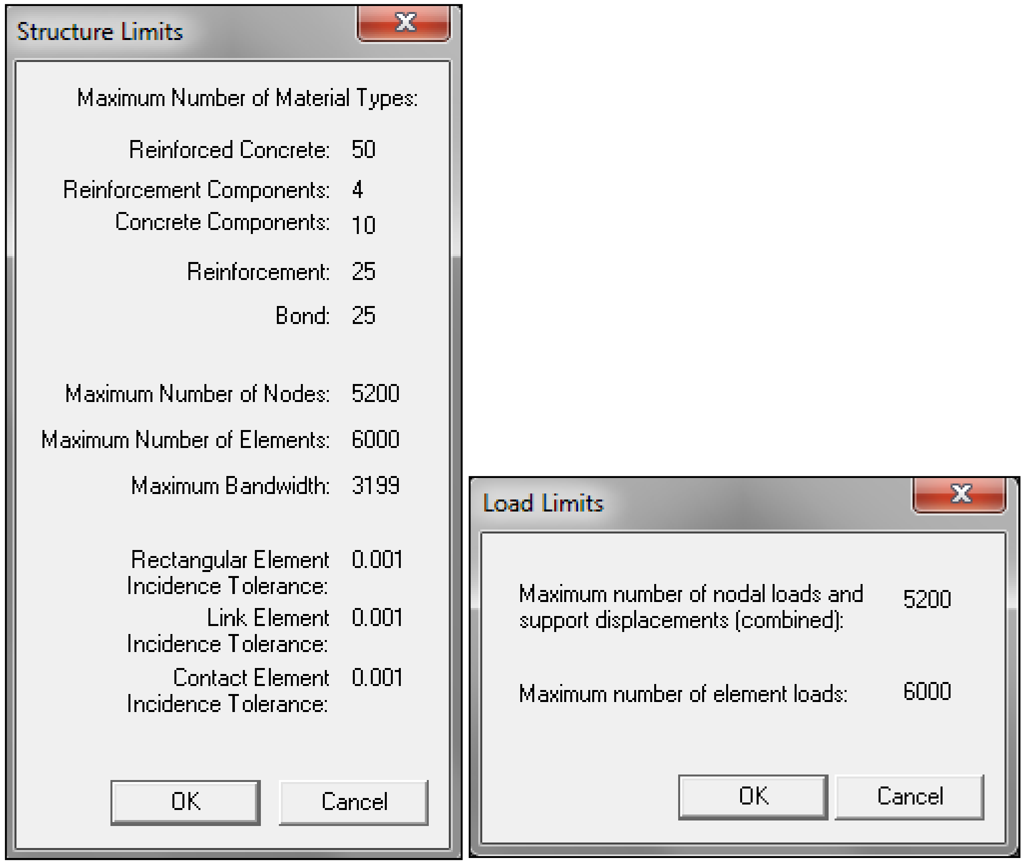


Figure 2-12 – Structure and load limits dialog boxes

2.2.4 Element Attributes

To view a summary of the attributes of an element, the user can position the mouse cross-hairs within the boundaries of the desired element and click the left mouse button. The selected element appears highlighted in green and the **Element Attributes** dialog box appears as shown in Figure 2-13. The element number, element type, incident nodes and their coordinates, material types and assigned loads are shown.

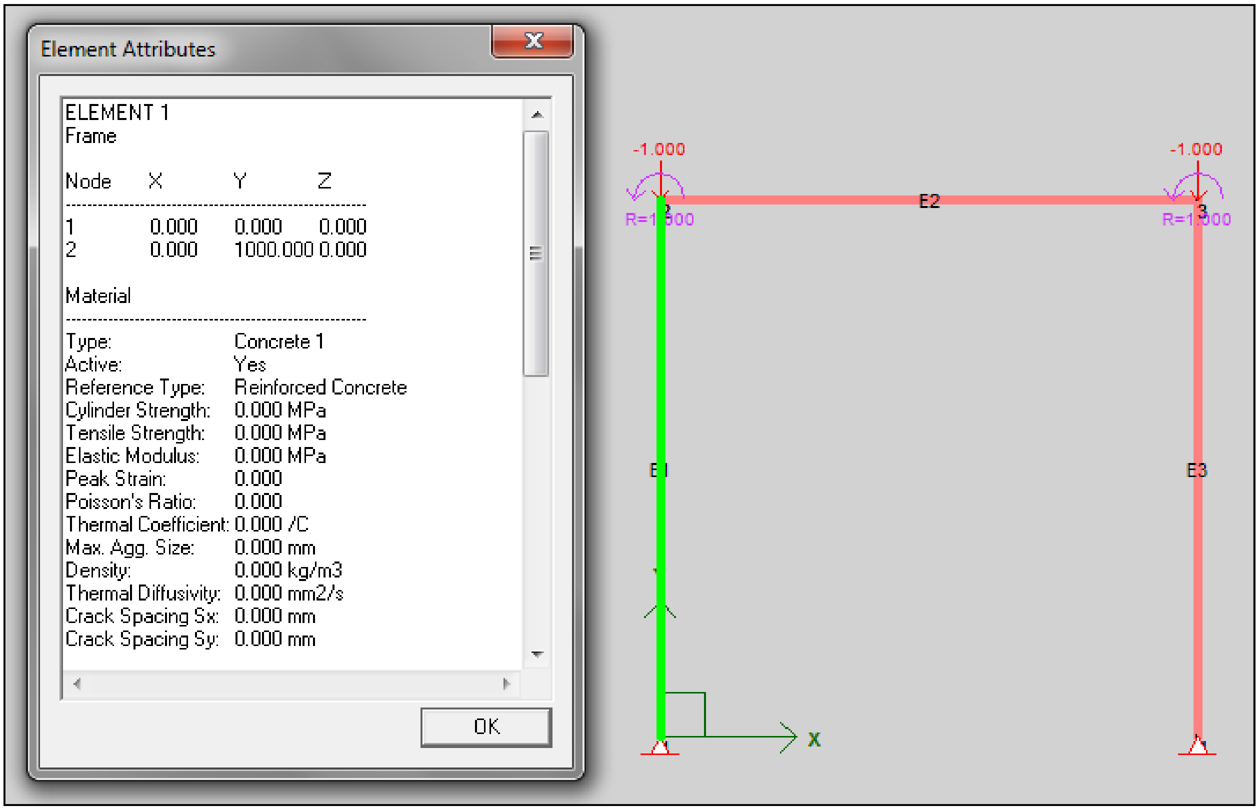


Figure 2-13 – Element Attributes window

Chapter 3

New Features for Modeling 2D Plane Membrane Structures

3.1 Introduction

VecTor2 is a nonlinear finite element analysis (NLFEA) program for modeling 2D reinforced concrete membrane structures. The theoretical bases of VecTor2 are the Modified Compression Field Theory (Vecchio and Collins, 1986) and the Disturbed Stress Field Model (Vecchio, 2000) – analytical models for predicting the response of reinforced concrete elements subjected to in-plane normal and shear stresses. The program uses a smeared, rotating-crack formulation to model concrete structures. To produce a stable and robust nonlinear solution, VecTor2 uses total-load iterative procedure with the secant stiffness formulation.

Incorporated into the program's analysis algorithms are material nonlinearity effects including compression softening due to transverse cracking, tension stiffening, tension softening, tension splitting and other mechanisms important in making a precise assessment of cracked reinforced concrete behaviour. In addition, the program is capable of modeling concrete expansion and confinement, cyclic loading and hysteretic response, construction and loading chronology for repair applications, bond slip, crack shear slip deformations, reinforcement dowel action, reinforcement buckling, and crack allocation process (Wong, 2002).

To create finite element models, VecTor2 utilizes a fine mesh of low-powered elements. The program's element library includes a 4-node rectangular or quadrilateral element (8 d.o.f.), a 3-node triangular element (6 d.o.f.), and a 2-node truss bar element (4 d.o.f.). The solid elements can be used to model reinforced concrete structures. The recent version of VecTor2 is also capable of modeling steel structures, masonry, wood, concrete-steel laminates, concrete-SFRC laminates, masonry-SFRC laminates, and concrete-ortho laminates. Reinforcement can be modeled either as smeared within the solid elements, or as discrete bars using the truss elements. A 2-node link and a 4-node interface element are also available to model bond-slip mechanisms.

VecTor2 is currently configured to accommodate: 6000 elements, 5200 nodes, 3200 bandwidth, 25 concrete material types, 25 reinforcement types, and 4 smeared reinforcement components per material type.

In the last 10 years, many new features have been added to VecTor2 such as additional behavioural models and several types of materials. As the main program developed, there was a need to update its pre-processor. A new version of FormWorks, FormWorks 3.5, was developed to support the recently added features of VecTor2 in the last 10 years.

The new features of FormWorks 3.5 can be categorized according to the following objectives:

- Enable the user to define different types of materials including steel structures, masonry, wood, concrete-steel laminates, concrete-SFRC laminates, masonry-SFRC laminates, and concrete-ortho laminates, and assign them to elements.
- Update the Job Page to include the most recent behavioural models and analysis parameters.
- Display the geometry of the finite element model properly; improve the Zooming features of the program.
- Add a new load type named Nodal Thermal Load to the program.
- Improve the performance of the rectangular and truss elements.

The different types of materials and new features in the Job Page will be discussed in this chapter. The other changes in FormWorks 3.5, including improvements in the Zooming tools, load types, and element types, are minor changes and should be self-evident.

3.2 The Job Data

The first step in creating the VecTor input files is to define the Job Data. At the time of analysis, FormWorks generates the *.JOB file based on the defined Job Data. Select the **Job/Define Job** menu item or click the **Job** toolbar button to open the **Job** dialog box. This window consists of three pages: the Control Page, the Model Page, and the Auxiliary Page.

3.2.1 The Job Control Page

The Job Control Page includes information about the Structure Type, Loading Data and Analysis Parameters. Figure 3-1 shows the Job Control Page window.

3.2.1.1 Job Data Group

These entry fields manage the creation of the *.JOB file.

- **Job File Name**

  Enter an alpha-numeric file name up to 8 characters long without spaces. This defines the file name to which FormWorks appends the *.JOB extension when saving the Job Data file.

- **Job Title**

  Enter a descriptive identifier up to 30 characters long to differentiate this analysis from others.

- **Date**

  Enter the date in a string up to 30 characters long.

3.2.1.2 Structure Data Group

These entry fields manage the creation of the *.S2R file.

- **Structure File Name**

  Enter an alpha-numeric file name up to 8 characters long without spaces. This defines the file name to which FormWorks appends the *.S2R extension when saving the Structure Data file.

- **Structure Title**

  Enter a descriptive identifier up to 30 characters long for the structure being analyzed.

- **Structure Type**

  Select **Plane Membrane (2D)** for a VecTor2 analysis.

3.2.1.3 Load Data Group

FormWorks allows the definition of five different load cases. Each load case consists of a load file name, load case title, initial factor, final factor, increment factor, load type, repetitions, and cyclic increment factor. While all load cases act simultaneously on the structure, different load cases can have different load factors. For example, the user can define the gravity load as a constant in Load Case 1 and a monotonically increasing lateral load in Load Case 2, both of them acting simultaneously on the structure.

Each load case is assigned one of the three loading types, **Monotonic**, **Cyclic** and **Reversed Cyclic**. Examples of each are illustrated in Figure 3-2 to 3-4.

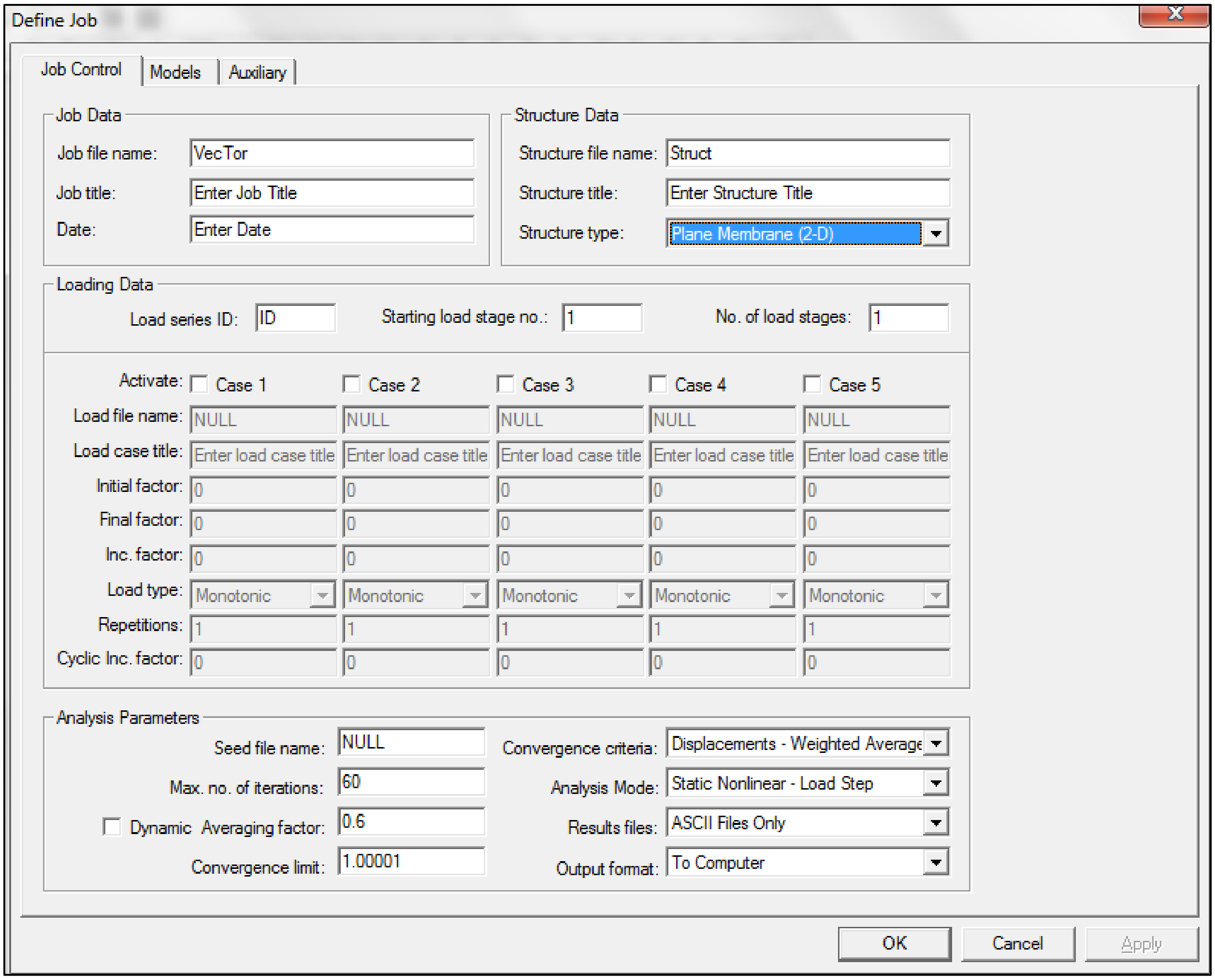


Figure 3-1 – Job Control Page in Job window

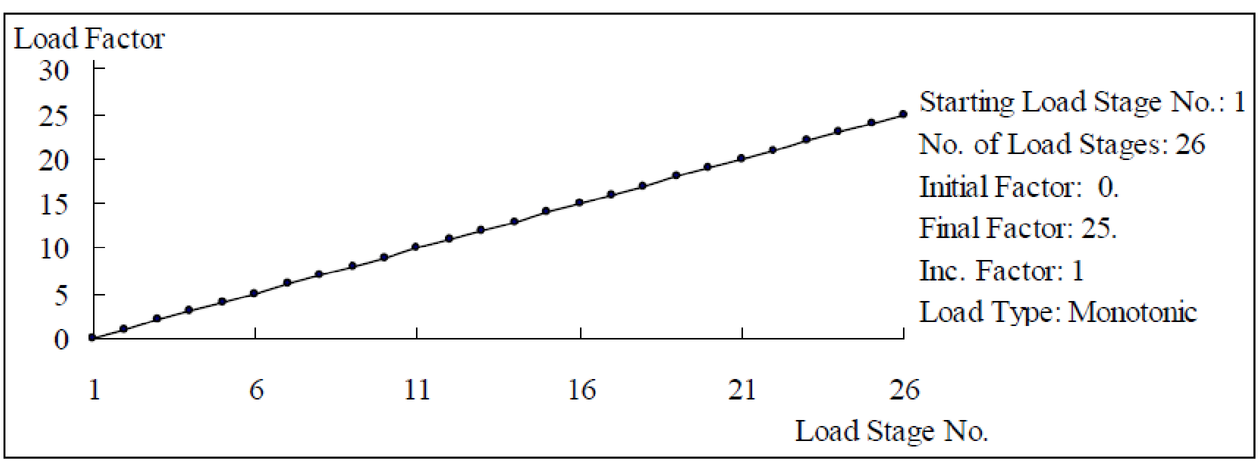


Figure 3-2 – Monotonic Type Loading. (Wong, 2002)

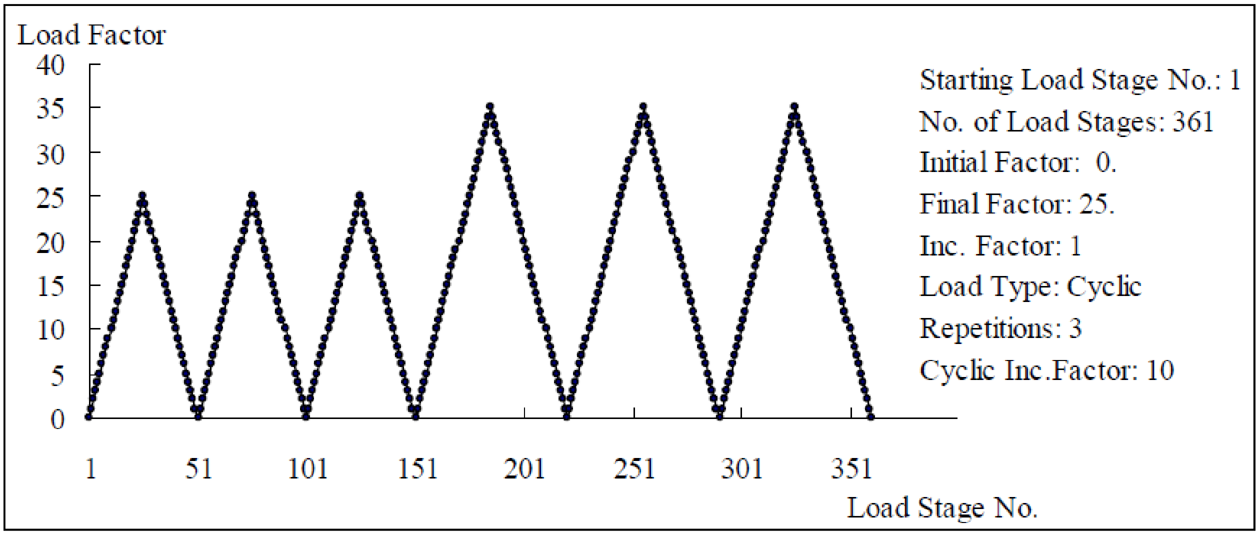


Figure 3-3 – Cyclic Type Loading. (Wong, 2002)

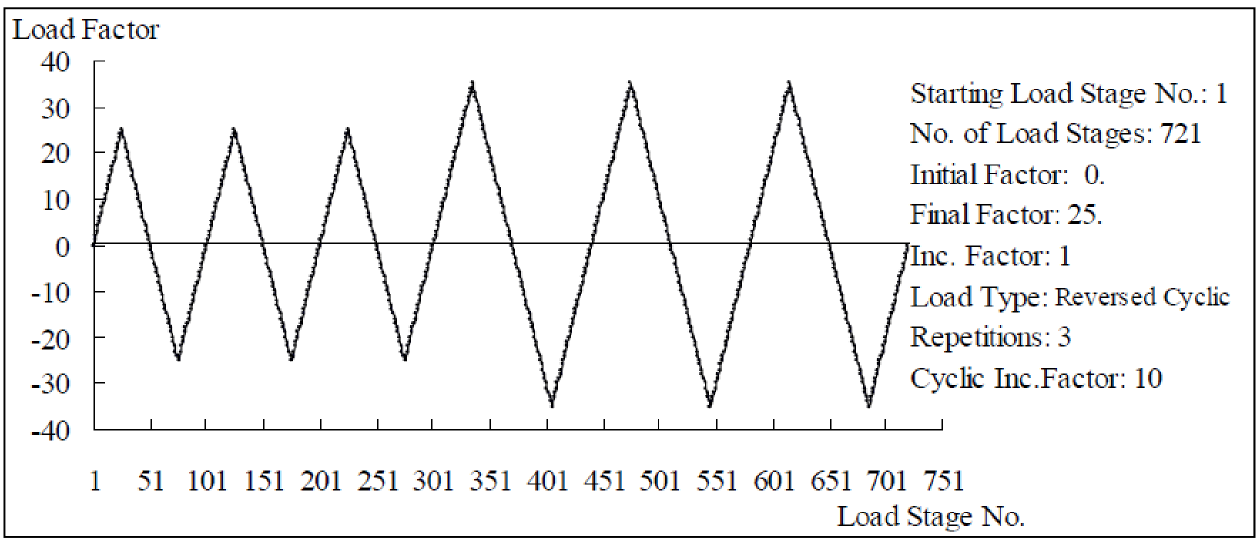


Figure 3-4 – Reversed Cyclic Type Loading. (Wong, 2002)

The size of the load steps, which is controlled by the size of the load factor increment, can appreciably impact the efficiency of the solution convergence. Many small load increments may be preferable to fewer large load increments, especially when the structure is at an advanced state of distress. Smaller load increments allow the solution to properly converge in a fewer number of iterations before the analysis proceeds to the next load step. Excessively large load increments may result in incomplete convergence. Given the overall softening response of concrete, improper converge may overestimate the strength for an imposed displacement, and underestimate the displacement for an imposed load (Wong, 2002).

The following entry fields are common to all load cases. They define the name of the *.A2E and *.A2R output files generated by VecTor2 and the number of loads stages to be analyzed. As the analysis proceeds, VecTor2 generates output files having the name **LoadCaseID_N.A2E** and/or **LoadCaseID_N.A2R**, where N is the current load stage number.

- **Load Case ID**
  Enter an alpha-numeric file name up to 5 characters long without spaces. This defines the file name to which VecTor2 appends the *.A2E and/or *.A2R extension when storing analysis results.

- **Starting Load Stage Number**
  Enter an integer greater than or equal to 1. This defines the number of the first *.A2E or *.A2R file that is stored by VecTor2. When resuming an analysis, enter an integer greater than the last completed load stage to avoid overwriting previously generated output files.

- **No. of Load Stages**
  Enter an integer greater than or equal to 1. This defines the number of load stages analyzed by VecTor2. The total number of required stages is defined by equation 3.1.

To create a load case, complete the following steps:

1. Check the load **Case** box to activate the load case. Only active load cases can be modified.
2. Complete the following entry fields for the activated load case.

- **Load File Name**
  Enter an alpha-numeric file name up to 8 characters long without spaces. This defines the file name to which FormWorks appends the *.L2R extension when generating the Load Case Data files.

- **Load Case Title**
  Enter a descriptive identifier up to 30 characters long to differentiate the load case.

- **Initial Factor**
  Enter the load factor of the first load stage.

- **Final Factor**

  For monotonic loading, enter the load factor of the last load stage. For cyclic and reversed loading, enter the maximum load factor of the first set of repetitions.

- **Inc. Factor**

  Enter the change in load factor from one load stage to the next.

- **Load Type**

  Select the desired loading type from the drop-list.

- **Repetitions**

  Enter the number of cycles per set (for cyclic and reversed cyclic loading only).

- **Cyclic Inc. Factor**

  Enter the change in final load factor from one set of repetitions to the next. For uniformity in load stage increments, the value should be a multiple of the load stage load factor increment. (For cyclic and reversed cyclic loading only)

- **Initial Load Stage**

  Enter the load stage number from which the load case should be activated.

Having specified the load factors and load factor increments, the number of load stages required to analyze all load stages can be computed as follows:

$$No.\ Stages = \begin{cases} \dfrac{LF_f - LF_i}{LS_{inc}} + 1 & \textit{for monotonic loading} \\[2ex] 2(R \cdot S)\left(\dfrac{LF_f - LF_i}{LS_{inc}}\right) + \left(\dfrac{R \cdot C_{inc}}{LS_{inc}}\right)[S(S-1)] + 1 & \textit{for cyclic loading} \\[2ex] 4(R \cdot S)\left(\dfrac{LF_f - LF_i}{LS_{inc}}\right) + \left(\dfrac{2R \cdot C_{inc}}{LS_{inc}}\right)[S(S-1)] + 1 & \textit{for reversed cyclic loading} \end{cases}$$

Equation 3.1 – Number of Load Stages (Wong, 2002).

Where LF_i is the initial load factor, LF_f is the final load factor, LS_{inc} is the load factor increment for each load stage, R is the number of repetitions, S is the number of sets of full repetitions and C_{inc} is the cyclic load factor increment.

3.2.1.4 Analysis Parameters Group

This group controls the progress of the iterative solution procedure and the analysis output:

- **Seed File Name**
  Enter **NULL** if no seed file is used. Otherwise, enter the file name of the *.A2R file.

- **Max. No. of Iterations**
  Enter the maximum number of iterations VecTor2 performs for each load stage. Regardless of the convergence quality, VecTor2 proceeds to the next load stage when this limit is reached.

- **Averaging Factor**
  Enter the weighting factor between 0 and 1 used to update the value of the material stiffness coefficients between iterations. Structures exhibiting less stability such as lightly reinforced structures require values closer to zero. Alternatively, select the **dynamic averaging factor** to allow VecTor2 to automatically choose a value based on response of the structure.

- **Convergence Limit**
  Enter a value greater than 1.0 for the maximum ratio of the convergence criteria that must be satisfied before the VecTor2 proceeds from one load stage to the next. As the value approaches 1.0, the more stringent the convergence criterion becomes.

- **Convergence Criteria**
  Select one of the following parameters by which the solution convergence is judged against the convergence limit before VecTor2 proceeds to the next load step.
  1. Secant Moduli - Weighted Average
  2. Displacements - Weighted Average
  3. Displacements - Maximum Value
  4. Reactions - Weighted Average
  5. Reactions - Maximum Value

  The default parameter is option 2, Displacements - Weighted Average.

- **Analysis Mode**
  Select one of the following options for analysis mode.
  0. Linear Elastic
  1. Static Nonlinear - Load Step
  2. Static Nonlinear - Time Step

3. Dynamic Nonlinear - General
4. Dynamic Nonlinear - EQ Record

The default mode is **Static Nonlinear-Load Step**. If utilizing time varying loads such as impulse forces or ground acceleration loads, select **Static Nonlinear-Time Step** from the drop-list. The last two options are for Dynamic analysis.

- **Results Files**

  Select one of the following file types for the analysis output. ASCII files are extended analysis files that can be read by text editors. Binary files are reduced analysis files that may be used as seed files.

  1. ASCII and binary files
  2. ASCII files only
  3. Binary files only
  4. Last load stage only

  The default parameter is option 2, ASCII files only.

- **Output Format**

  Select **To Computer**.

3.2.2 The Models Page

The second page in the Job Data window is the Models Page which includes information about material and behavioural models. Figure 3-5 shows the Models Page.

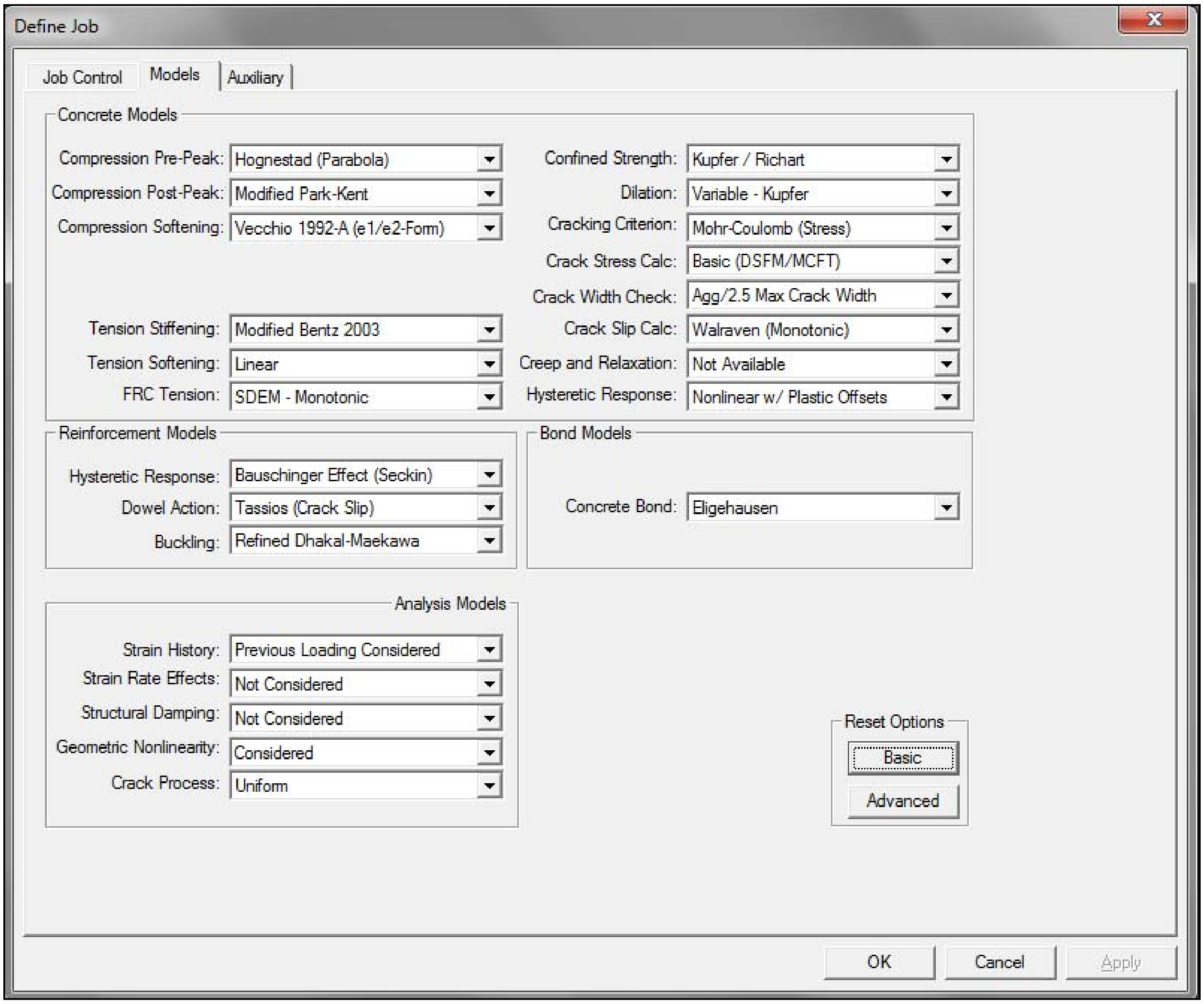


Figure 3-5 – Models Page in Job window

FormWorks 3.5 provides two default options for Models Page parameters, Basic Default and Advanced Default. For most analyses, it is advisable to select the VecTor2 basic default models.

3.2.2.1 Concrete Models

- **Compression Pre-Peak Response**

  Select one of the following options for the ascending branch of the average concrete compression stress-strain response.

  0. Linear
  1. Hognestad (Parabola)
  2. Popovics (NSC)
  3. Popovics (HSC)
  4. Hoshikuma et al
  5. Smith-Young
  6. Lee et al 2011 (FRC)

  The basic default parameter is option 1, Hognestad (Parabola).

  The advanced default parameter is option 4, Hoshikuma et al.

- **Compression Post-Peak Response**

  Select one of the following options for the descending branch of the average concrete compression stress-strain response.

  0. Base Curve
  1. Modified Park-Kent
  2. Montoya 2003
  3. Popovics / Mander
  4. Hoshikuma et al
  5. Saenz / Spacone
  6. Lee et al 2011 (FRC)

  The basic default parameter is option 1, Modified Park-Kent.

  The advanced default parameter is option 4, Hoshikuma et al.

- **Compression Softening**

  Select one of the following models for reducing the concrete compressive strength and stiffness, relative to the presence of coexisting transverse tensile strains.

  0. No Compression Softening
  1. Vecchio 1992-A (e1/e2-Form)
  2. Vecchio 1992-B (e1/e0-Form)
  3. Vecchio-Collins 1982

4. Vecchio-Collins 1986

Both the basic and the advanced default models are option 1, Vecchio 1992-A (e1/e2-Form).

- **Tension Stiffening**

  Select one of the following models for the post-cracking average tensile stress-strain response of reinforced concrete.

  0. No tension stiffening
  1. Modified Bentz 2003
  2. Vecchio 1982
  3. Collins-Mitchell 1987
  4. Bentz 1999
  5. Izumo, Maekawa Et Al
  6. Tension Chord Model (Kaufmann)
  7. Lee 2010 (w/ Post-Yield)

  The basic default model is option 1, Modified Bentz 2003.

  The advanced default model is option 7, Lee 2010 (w/ Post-Yield).

- **Tension Softening**

  Select one of the following models for the post-cracking average tensile stress-strain response of plain concrete.

  0. Not Considered
  1. Linear
  2. Bilinear
  3. Nonlinear (Yamamoto)
  4. Nonlinear (Hordijk)
  5. Exponential
  6. Custom Input (Strain Based)
  7. Custom Input (Crack Based)
  8. FRC (fib Model Code 2010)

  The basic default model is option 1, Linear Model.

  The advanced default model is option 7, Bilinear Model.

- **FRC Tension**

  Select one of the following models for the post-cracking response of FRC in tension.

  0. Not Considered

1. SDEM – Monotonic
2. SDEM - Cyclic
3. DEM (Lee et al 2011)
4. VEM (Voo& Foster 2003)
5. Fib Model Code 2010

The basic default model is option 1, SDEM – Monotonic.

The advanced default model is option 2, SDEM - Cyclic.

- **Confined Strength**

  Select one of the following models for strength and ductility enhancement of concrete subjected to biaxial or triaxial compressive stress states.

  0. Strength Enhancement Neglected
  1. Kupfer / Richart
  2. Montoya / Ottosen
  3. Selby

  The basic default model is option 1, Kupfer / Richart Model.

  The advanced default model is option 2, Montoya / Ottosen Model.

- **Dilation**

  Select one of the following models for computing the post-cracking Poisson's effect for expansion in the direction transverse to compressive stresses.

  0. Constant Poisson's Ratio
  1. Variable - Kupfer
  2. Variable - Montoya 2003
  3. Variable - Montoya w/ Limit

  The basic default model is option 1, Variable – Kupfer Model.

  The advanced default model is option 3, Variable - Montoya w/ Limit Model.

- **Cracking Criterion**

  Select one of the following models for determining the concrete cracking strength based on the stress or strain state of an assumed failure condition.

  0. Uniaxial Cracking Stress
  1. Mohr-Coulomb (Stress)
  2. Mohr-Coulomb (Strain)
  3. CEB-FIP

4. Gupta 1998
5. Mohr-Coulomb (w/ Strain History)

Both the basic and the advanced default models are option 1, Mohr-Coulomb (Stress) Model.

- **Crack Stress Calculation**

  Select one of the following models for calculating the stress at the cracks.

  0. Crack Stress Calcs Omitted
  1. Basic (DSFM/MCFT)
  2. Advanced (Lee 2009)

  The basic default model is option 1, Basic (DSFM/MCFT) Model.

  The advanced default model is option 2, Advanced (Lee 2009) Model.

- **Crack Width Check**

  Select one of the following crack width beyond which the average concrete compressive stress is reduced to reflect the inability of the concrete to transmit compressive stresses across large crack widths.

  0. Stability Check Omitted
  1. Agg/2.5 Max Crack Width
  2. Agg/5.0 Max Crack Width
  3. 10 mm Max Crack Width
  4. 5 mm Max Crack Width
  5. 2 mm Max Crack Width
  6. 1 mm Max Crack Width

  Both the basic and the advanced default models are option 1, check based on Agg/2.5 Max Crack Width.

- **Crack Slip Calculation**

  Select one of the following models for determining the crack slip strains as a component of the total strains. If **Not Considered** is selected, VecTor2 conducts the analysis based on Modified Compression Field Theory. If any other option is selected, VecTor2 conducts the analysis based on the Disturbed Stress Field Model.

  0. Not Considered
  1. Walraven (Monotonic)
  2. Vecchio-Lai (Cyclic)
  3. Maekawa (Monotonic)

4. Hybrid I (Walraven)
5. Hybrid II (Vecchio-Lai)
6. Hybrid III (Maekawa)

The basic default model is option 1, Walraven (Monotonic).

The advanced default model is option 2, Vecchio-Lai (Cyclic)**.**

- **Concrete Creep and Relaxation:**

  This option is not currently available.

- **Hysteretic Response**

  Select one of the following models for the average stress-strain response of concrete when subjected to unloading and the resulting plastic strain offsets.

  0. Linear w/ No Plastic Offsets
  1. Linear w/ Plastic Offsets
  2. Nonlinear w/ Plastic Offsets
  3. Palermo 2002 (w/ Decay)

  The basic default model is option 2, Nonlinear w/ Plastic Offsets.

  The advanced default model is option 3, Palermo 2002 (w/ Decay)**.**

3.2.2.2 Reinforcement Models

- **Hysteretic Response**

  Select one of the following models for the average stress-strain response of reinforcement when subjected to unloading and the resulting plastic strain offsets.

  0. Linear
  1. Bauschinger Effect (Seckin)
  2. Elastic-Hardening (Curvilinear)
  3. Elastic-Hardening (Trilinear)
  4. Elastic-Plastic (Bilinear)
  5. Seckin (w/ Local Accumulation)

  Both the basic and the advanced default models are option 1, Bauschinger Effect (Seckin).

- **Dowel Action**

  Select one of the following models for determining the contribution to shear resistance of the reinforcing bars crossing cracks.

  0. Not Considered

1. Tassios (Crack Slip)
2. Tassios (Strength)

Both the basic and the advanced default models are option 1, Tassios (Crack Slip).

- **Buckling**

  Select the model for determining the failure of truss bar elements in compression due to buckling and associated splitting of the concrete cover. Note that to use these models the truss bar elements must be connected to the concrete with bond elements.

  0. Not Considered
  1. Refined Dhakal-Maekawa
  2. Dhakal-Maekawa 2002
  3. Asatsu

  Both the basic and the advanced default models are option 1, Refined Dhakal-Maekawa.

3.2.2.3 Bond Models

- **Concrete Bond or Adhesion**

  Select one of the following models for the bond stress-slip relationship of between concrete and embedded reinforcing bars.

  0. Perfect Bond
  1. Eligehausen
  2. Gan-Vecchio
  3. Harjli
  4. Fujii
  5. Eligehausen (No Cyclic Damage)
  6. Gan (No Cyclic Damage)

  The basic default model is option 1, Eligehausen Model.

  The advanced default model is option 2, Gan-Vecchio**.**

3.2.2.4 Analysis Models

- **Strain History**:

  Previous loading should be considered in analyses involving cyclic and reversed cyclic loading, so as to be available for the hysteretic response models. Both the basic and the advanced default options are Previous Loading Considered.

- **Strain Rate Effects:**

  Select one of the following models for determining the strain rate effects.

  0. Not Considered
  1. CEB (Full)
  2. CEB (Approx)

  Both the basic and the advanced default models are option 0, Not Considered.

- **Structural Damping:**

  Select one of the following models for determining the damping of the structure.

  0. Not Considered
  1. Rayleigh Damping
  2. Alternative Damping

  The basic default is option 0, Not Considered.

  The advanced default is option 1, Rayleigh Damping.

- **Geometric Nonlinearity:**

  Determine to consider Geometric Nonlinearity or not.

  0. Not Considered
  1. Considered

  Both the basic and the advanced default options are to consider Geometric Nonlinearity.

- **Crack Process:**

  Select one of the following models for determining the crack allocation process.

  0. Uniform
  1. Variable (Sato 2002)

  The basic default model is option 0, Uniform.

  The advanced default model is option 1, Variable (Sato 2002).

3.2.3 The Auxiliary Page

The third page of the Job Data window is the Auxiliary Page which includes information about material properties, masonry structures and dynamic analysis options. Figure 3-6 shows the Auxiliary Page.

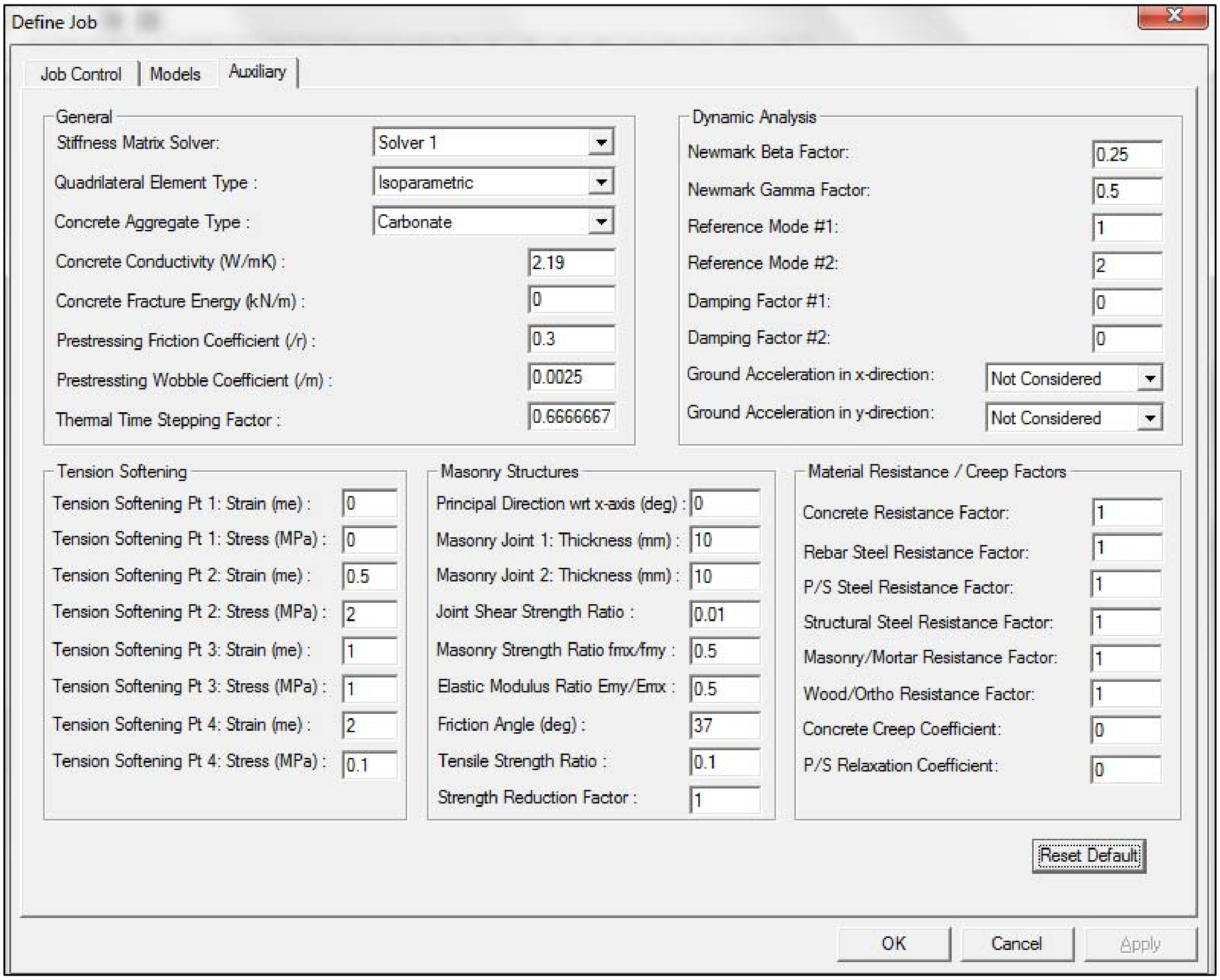


Figure 3-6 – Auxiliary Page in Job window

FormWorks 3.5 provides default options for Auxiliary Page parameters. In most of the analysis, it is advisable to select the VecTor2 default models.

3.2.3.1 General

- **Stiffness Matrix Solver**

  Select the solver to use. Currently, only Solver 1 (Cholesky decomposition) is available.

- **Quadrilateral Element Type**

  Select one of the following types of quadrilateral element to use in the finite element mesh.

  1. Isoparametric
  2. Degenerate

  Degenerate elements allow for a mixture of both quadrilateral and triangular elements. Isoparametric quadrilateral elements are convenient for structures with complex geometries, however are more computationally intensive. The default option is Isoparametric type.

- **Concrete Aggregate Type**

  Select one of the following options for concrete aggregate type. The two options are

  1. Carbonate
  2. Siliceous

  The only difference between the two aggregate types with respect to nonlinear finite element analysis is their response to thermal loading. The default value is Carbonate type.

- **Concrete Thermal Conductivity**

  Enter the concrete thermal conductivity. The default value is 2.19 W/mK.

- **Concrete Fracture Energy**

  Enter the concrete fracture energy. The default value is calculated based on the following formula when input parameter is 0:

$$G_f = \frac{2.5\ \alpha_0 \left(\frac{f'_c}{0.051}\right)^{0.46} \left(1+\frac{a_g}{11.27}\right)^{0.22} \left(\frac{w}{c}\right)^{-0.3}}{1000}$$

Bazant (2002)

$\alpha_0 = 1.44$

$\frac{w}{c} = 0.35$

- **Prestressing Friction Coefficient**

  Enter the prestressing friction coefficient. The default value is 0.30 /rad.

- **Prestressing Wobble Coefficient**

  Enter the prestressing wobble coefficient. The default value is 0.0025/m.

- **Thermal Time Stepping Factor**

  Enter the thermal time stepping factor. This factor determines the time step used in the Crank-Nicolson Method. In general, the accuracy of the result will decrease with increasing time step size. The default value is 2/3 or 0.666667.

3.2.3.2 Dynamic Analysis Options

- **Newmark Beta Factor**

  Enter the Newmark Beta Factor to be used in the analysis. The default value in VecTor2 is $\beta = 0.25$ which defines constant acceleration in Newmark's method of direct integration. For the linear acceleration method, $\beta = 1/6$ can be used. Caution is advised when using any other value for β.

- **Newmark Gamma Factor**

  Enter the Newmark Gamma Factor. The default value is 0.5.

- **Reference Mode #1**

  Enter the first mode shape number to be used in Rayleigh damping. The default value is 1.

- **Reference Mode #2**

  Enter the second mode shape number to be used in Rayleigh damping. The default value is 2.

- **Damping Factor #1**

  Enter the damping factor assigned to Reference Mode #1. The default value is 0 (no damping).

- **Damping Factor #2**

  Enter the damping factor assigned to Reference Mode#2. The default value is 0 (no damping).

- **Ground Acceleration in x-direction**

  Choose 'Considered' or 'Not Considered' for the ground acceleration in the x-direction. The Considered option is to be chosen when the Analysis Mode on the Job Control page is set to Dynamic Nonlinear—General.

- **Ground Acceleration in y-direction**

  Choose 'Consider' or 'Not Consider' for the ground acceleration in the y-direction. The Considered option is to be chosen when the Analysis Mode on the Job Control page is set to Dynamic Nonlinear—General.

3.2.3.3 Tension Softening Options

By specifying tension softening points, a custom tension softening curve can be developed.

- **Tension Softening Pt. 1: Strain**
  Enter the strain, expressed in units of millistrain, for the first tension softening point.

- **Tension Softening Pt. 1: Stress**
  Enter the stress, expressed in MPa, for the first tension softening point.

- **Tension Softening Pt. 2: Strain**
  Enter the strain for the second point on the tension softening curve.

- **Tension Softening Pt. 2: Stress**
  Enter the stress for the second point on the tension softening curve.

- **Tension Softening Pt. 3: Strain**
  Enter the strain for the third point on the tension softening curve.

- **Tension Softening Pt. 3: Stress**
  Enter the stress for the third point on the tension softening curve.

- **Tension Softening Pt. 4: Strain**
  Enter the strain for the fourth point on the tension softening curve.

- **Tension Softening Pt. 4: Stress**
  Enter the stress for the fourth point on the tension softening curve.

Note: If in the Models Page, Custom Input (Crack Based) is selected as the Tension Softening Model, then the strain points are in terms of crack width (mm).

Note: If in the Models Page, FRC (fib Model Code 2010) is selected as the Tension Softening Model, the Tension Softening part in Auxiliary Page changes as shown in Figure 3-7.

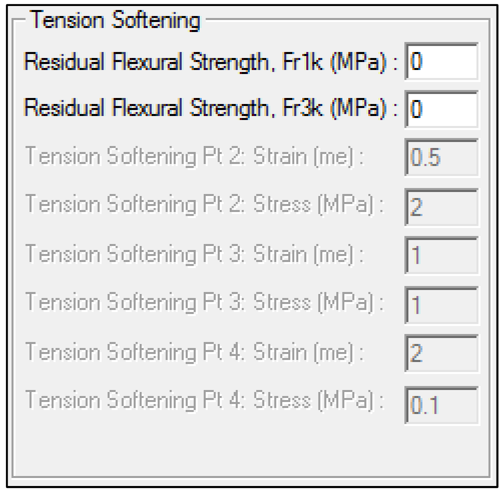


Figure 3-7 – Tension Softening parameters in Auxiliary Page when FRC (fib Model Code 2010) is selected as Tension Softening Model

- **Residual Flexural Strength, Fr1k:**
  Enter the residual flexural strength of Fiber Reinforced Concrete, Fr1k.

- **Residual Flexural Strength, Fr3k:**
  Enter the residual flexural strength of Fiber Reinforced Concrete, Fr3k.

Note: The residual flexural strength can be calculated by the formula expressed in fib Model Code 2010:

$$f_{R,j} = \frac{3F_j L}{2bh_{sp}^2}$$

where:

$f_{R,j}$ is the residual flexural tensile strength corresponding with Crack Mouth Opening Displacement at j $(CMOD_j)$

F_j is the load corresponding with $CMOD_j$

L is the span length (mm)

b is the width of the specimen (mm)

h_{sp} is the distance between the notch tip and the top of the specimen (mm)

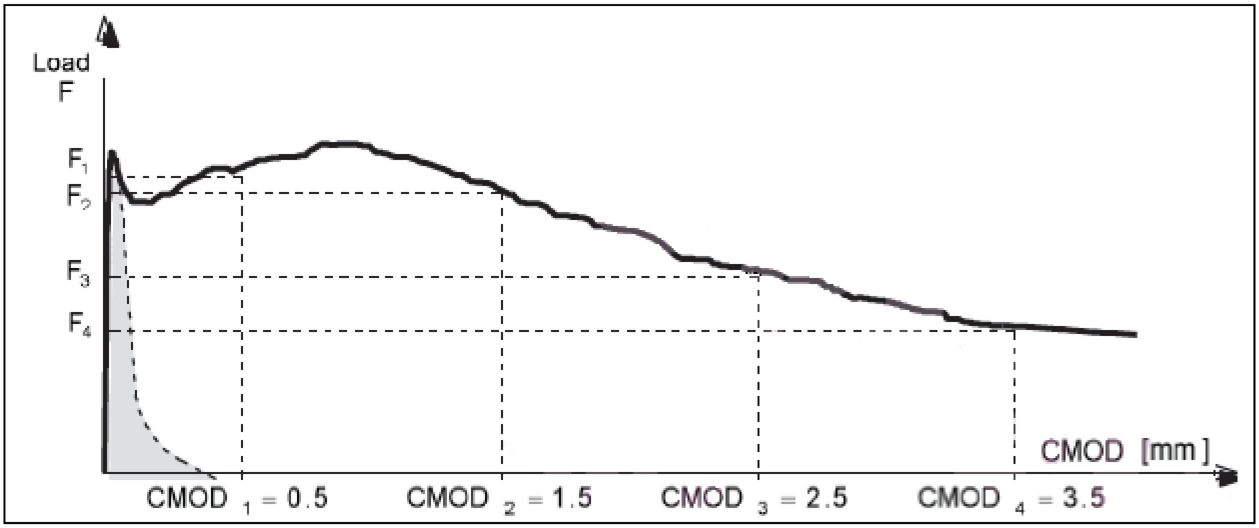


Figure 3-8 – Inverse analysis of beam in bending performed to obtain stress - crack opening relation (fib Model Code 2010).

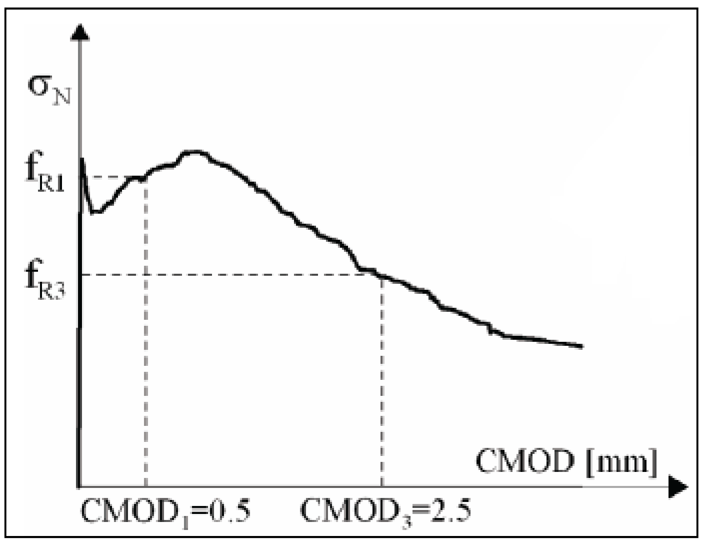


Figure 3-9 – Typical results from a bending test on a softening material (fib Model Code 2010).

3.2.3.4 Masonry Structures

- **Principal Direction wrt x-axis:**

  In degrees, enter the direction of Joint 1, as shown in Figure 3-10. Once the direction of Joint 1 is set, Joint 2 is considered to be perpendicular to it. The default value of the principal direction in VecTor2 is 0 degrees, meaning that it is assumed to be aligned with the x-axis (ie. horizontal).

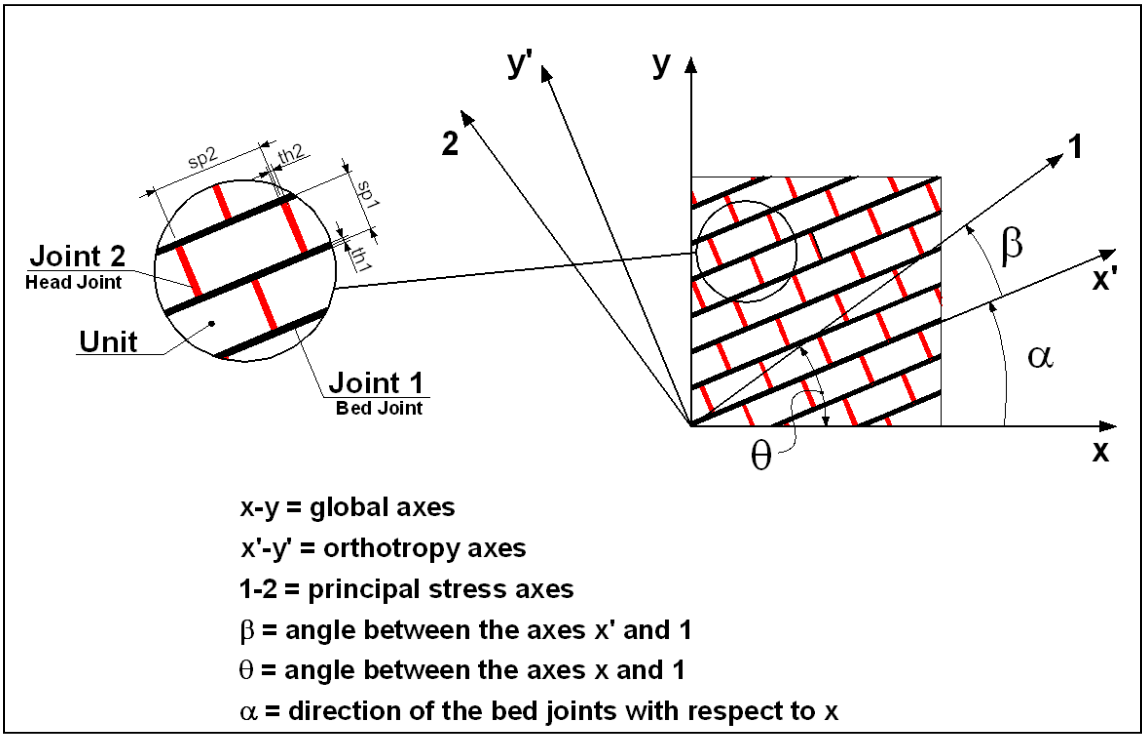


Figure 3-10 – Determining the principal direction wrt x-axis for Masonry Structures (Schlöglmann, 2004).

- **Masonry Joint 1: Thickness:**

  Enter the thickness of Joint 1, $th1$, in millimeters. The default value is 10 mm.

- **Masonry Joint 2: Thickness:**

  Enter the thickness of Joint 2, $th2$, in millimeters. The default value is 10 mm.

- **Joint Shear Strength Ratio:**

  Enter the Joint Shear Strength Ratio, c/f_{my}, for the masonry structure. This is the ratio between the shear strength of the joints, c, and the compressive strength in y-direction, f_{my}.The default value is 0.01.

- **Masonry Strength Ratio:**

  Enter the masonry strength ratio, f_{mx}/f_{my}. This defines the ratio between the compressive strength of masonry in x-direction, f_{mx}, and the compressive strength of masonry in y direction, f_{my}. The default value is 0.5.

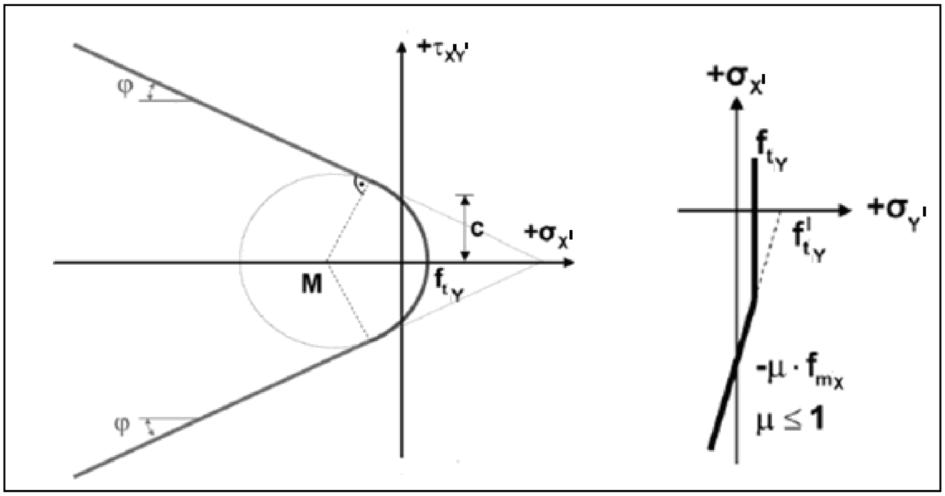


Figure 3-11 – Determining Joint Shear Strength Ratio for Masonry Structures (Schlöglmann, 2004).

- **Elastic Modulus Ratio:**

  Enter the elastic modulus ratio, E_{my}/E_{mx}. This defines the ratio between the initial elastic modulus in y-direction, E_{my}, and the initial elastic modulus in x-direction, E_{mx}. The default value is 0.5.

- **Friction Angle:**

  Enter the friction angle, φ, of the joints, in degrees. The friction angle is illustrated in the Joint Shear Strength Ratio subsection. The default value is 37 degrees.

- **Brick Strength Ratio:**

  Enter the brick strength ratio. This defines the ratio between the tensile strength of masonry in the x-direction, evaluated when $\alpha = 0$, and the compressive strength, f_{my}. The default value is 0.1

- **Strength Reduction Factor:**

  Enter the shear strength reduction factor, μ. This is the reduction factor for the f_{mx} strength, applicable in cases of uniaxial compression. The reduction factor, μ, accounts for the decrease in stress parallel to the bed joints under uniaxial compression. The default value is 1.

3.2.3.5 Material Resistance/Creep Factors

- **Concrete Resistance Factor:**

  Enter the concrete resistance factor. The default value is 1.

- **Rebar Steel Resistance Factor:**
  Enter the rebar steel resistance factor. The default value is 1.

- **P/S Steel Resistance Factor:**
  Enter the prestressing steel resistance factor. The default value is 1.

- **Structural Steel Resistance Factor:**
  Enter the structural steel resistance factor. The default value is 1.

- **Masonry/Mortar Resistance Factor:**
  Enter the masonry/mortar resistance factor. The default value is 1.

- **Wood/Ortho Resistance Factor:**
  Enter the wood resistance factor. The default value is 1.

- **Concrete Creep Coefficient:**
  Enter the creep coefficient of concrete, ψ_c. The default value is 0 (no creep).
  $$E_c' = \frac{E_c}{(1+\psi_c)}$$

- **P/S Relaxation Coefficient:**
  Enter the relaxation coefficient for the prestressing steel, ψ_s. The default value is 0 (no relaxation).
  $$E_s' = \frac{E_s}{(1+\psi_s)}$$

3.3 Reinforced Concrete Material Types

As discussed before, reinforced concrete materials in the FormWorks include concrete materials, as well as other materials, that are modeled with or without smeared reinforcement components or laminate components. In VecTor2, reinforced concrete is applied to solid elements including rectangular, quadrilateral, or triangular elements. To add, modify or delete concrete material types, click the **Structure/Define Reinforced Concrete Materials** menu item or click the **Define Reinforced Concrete Materials** toolbar button.

Although referred to as Reinforced Concrete, a wide variety of materials can be analyzed in VecTor2 and modeled in FormWorks 3.5 (or FormWorks-Plus).

Reinforced Concrete materials can include:

- Reinforced concrete
- Structural steel
- Masonry
- Wood (fixed orthotropic)
- Concrete-steel laminate
- Concrete-SFRC laminate
- Masonry-SFRC laminate
- Concrete-ortho laminate

The materials listed above can be modeled with or without smeared reinforcement. The reinforcement components that are available to be used with the above material types are:

- Ductile steel reinforcement
- Prestressing steel
- Tension only reinforcement
- Compression only reinforcement
- External bonded FRP Fabric
- Steel-fibre—hooked
- Steel-fibre—straight
- Steel skin plate (for concrete-steel laminates only)
- SFRC laminate—hooked fibre (for concrete-SFRC laminates only)
- SFRC laminate—straight fibre (for concrete-SFRC laminates only)
- Orthotropic laminate (for concrete-ortho laminates only)
- Shape Memory Alloy Type 1
- Shape Memory Alloy Type 2

Brief descriptions of how to model the above concrete materials and reinforcement components in FormWorks are included in the following sections.

To **Add Concrete Material Types**, complete the following steps:

1. Enter the properties in the **Concrete Properties** group. Note that properties marked by * are assigned default values when '0' are entered in their fields.
2. Click **Add** in the Concrete Types group. The newly added concrete type appears in the Concrete Types list box and the reinforcement component properties fields are enabled.

To **Modify Concrete Material Types**, complete the following steps:

1. In the Concrete Types list box, select the concrete type to be modified.
2. Re-enter the properties in the Concrete Properties group as desired.
3. Click **Update** in the Concrete Types group to store the modified concrete properties.

To **Delete Concrete Material Types**, complete the following steps:

1. In the **Concrete Types** list box, select the concrete type to be deleted.
2. Click **Delete** in the Concrete Types group. The concrete material type is deleted from the list box and the remaining concrete types are renumbered.

Perform the same process to add, update or delete Reinforcement Components Properties.

3.3.1 Reinforced Concrete

The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-12 when Reinforced Concrete is selected as the main material type.

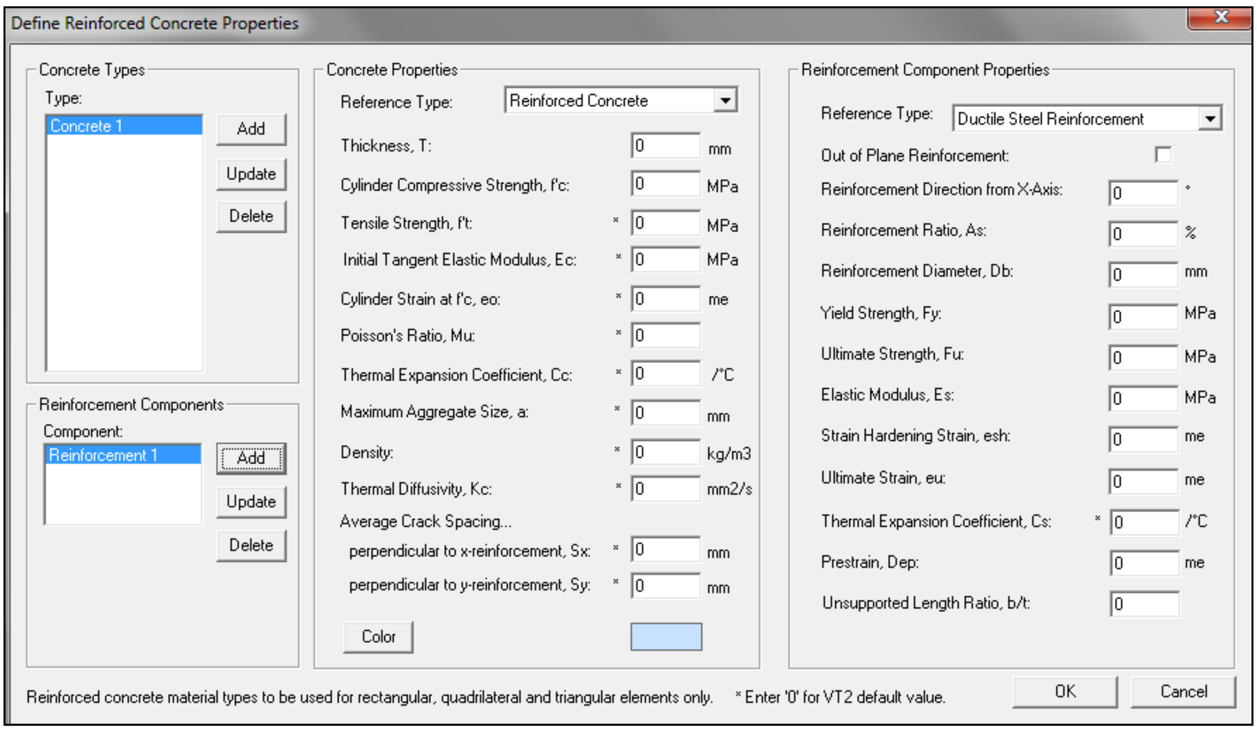


Figure 3-12 – Define Reinforced Concrete Properties window when Reinforced Concrete type is selected as the main material type

Enter the following properties in the **Concrete Properties** group:

- **Thickness, T :**

  Enter the out of plane thickness of the concrete, in millimeters.

- **Cylinder Compressive Strength, f'c:**

  Enter the peak compressive stress of a standard concrete test cylinder, in MPa.

- **Tensile Strength, f't:**

  Enter the uniaxial cracking strength of concrete, in MPa. The default value is $\mathrm{f_t} = 0.33\sqrt{\mathrm{f_c'}}$ MPa

- **Initial Tangent Modulus, Ec:**

  Enter the tangent stiffness of the concrete stress-strain response at zero-strain, in MPa. The default value is $E_c = 3320\sqrt{f_c'} + 6900$ MPa.

- **Cylinder Strain at f'c, eo:**

  Enter the compressive strain, ε_0, corresponding to f'c, as a positive value, in millistrain. The default value is $\varepsilon_0 = 1.8 + 0.0075 f_c'$ millistrain.

- **Poisson's Ratio, Mu:**

  Enter the initial Poisson's ratio, υ_0, of the concrete as a positive value. The default value is 0.15.

- **Thermal Expansion Coefficient, Cc:**

  Enter the concrete strain increase per temperature increase of 1°C. The default value is 10×10^{-6} /°C.

- **Maximum Aggregate Size, a:**

  Enter the maximum aggregate size, in millimeters. The default value is 10 mm.

- **Density:**

  Enter the mass density of the concrete, in $\mathrm{kg/m^3}$. The default value is 2400 $\mathrm{kg/m^3}$.

- **Thermal Diffusivity, Kc:**

  Enter the rate at which temperature changes occur in the concrete. The default value is 1.20 $\mathrm{mm^2/s}$.

- **Colour:**

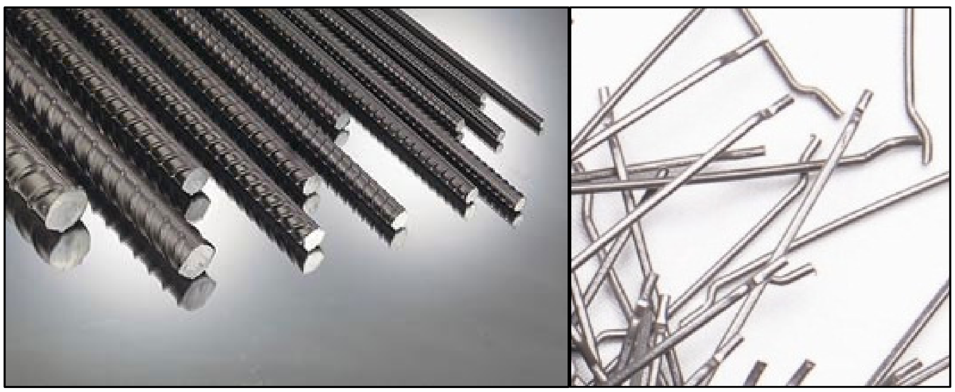
  Select the display colour of the concrete material type in the FormWorks Workspace.

- **Average Crack Spacing:**

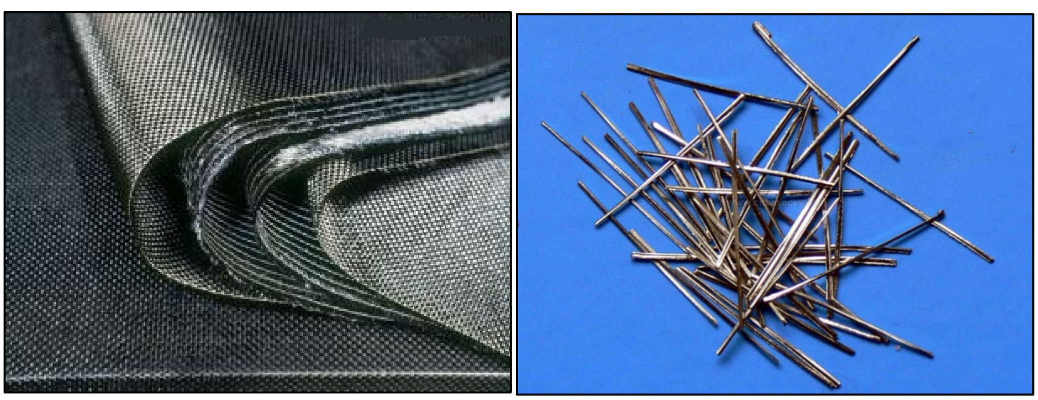
  Enter the crack control parameter indicating the spacing of cracks perpendicular to the x-axis (Sx), and perpendicular to the y-axis (Sy), in millimeters. The default values are computed by the CEB-FIP model, unless the Tension Chord (Kauffmann) model is selected for tension stiffening.

Select one of the following types of reinforcement from the drop list in the **Reinforcement Component Properties** group:

1. Ductile Steel Reinforcement
2. Prestressing Steel
3. Tension Only Reinforcement
4. Compression Only Reinforcement
5. External Bonded FRP Fabric
6. Steel-Fibre-Hooked
7. Steel-Fibre-Straight
8. Shape Memory Alloy Type 1
9. Shape Memory Alloy Type 2



(a) Steel Reinforcement **(b) Steel-Fibre-Hooked**



(c) FRP Fabric **(d) Steel-Fibre-Straight**

Figure 3-13 – Types of reinforcement which can be used as Reinforced Concrete components

3.3.1.1 Ductile Steel Reinforcement

Enter the following properties in the **Reinforcement Component Properties** group:

- **Out of Plane Reinforcement:**
  Check this box to orient the reinforcement perpendicularly to the XY plane.

- **Reinforcement Direction from X-Axis:**
  For in-plane reinforcement, enter the inclination of the reinforcement axis, measured counterclockwise from the positive x-axis in degrees. Enter a value between 0° and 360°.

- **Reinforcement Ratio, rho:**
  Enter the ratio of cross-sectional area of the reinforcement to the area of concrete over which it is smeared, expressed as a percentage.

- **Reinforcement Diameter, Db:**
  Enter the diameter of the reinforcing bar, in millimeters.

- **Yield Strength, Fy:**
  Enter the yield stress of the reinforcement, in MPa.

- **Ultimate Strength, Fu:**
  Enter the maximum stress the reinforcement can attain before rupturing, in MPa. The ultimate strength must be greater than the yield strength.

- **Elastic Modulus, Es:**
  Enter the stiffness of the initial linear-elastic branch of the stress-strain response, in MPa.

- **Strain Hardening Strain, esh:**
  Enter the strain at which the reinforcement stress-strain response begins to ascend from the yield plateau to the ultimate strength, in millistrain. The value must be greater than or equal to the yield strain, Fy/Es.

- **Ultimate Strain, ε_u:**
  Enter the strain at which the reinforcement ruptures, in millistrain.

- **Thermal Expansion Coefficient, Cs:**
  Enter the reinforcement strain increase per temperature increase of 1°C. The default value is 10×10^{-6} /°C.

Prestrain, $\Delta\epsilon_p$:

Enter the elastic strain offset of the reinforcement relative to the unstrained concrete, in millistrain.

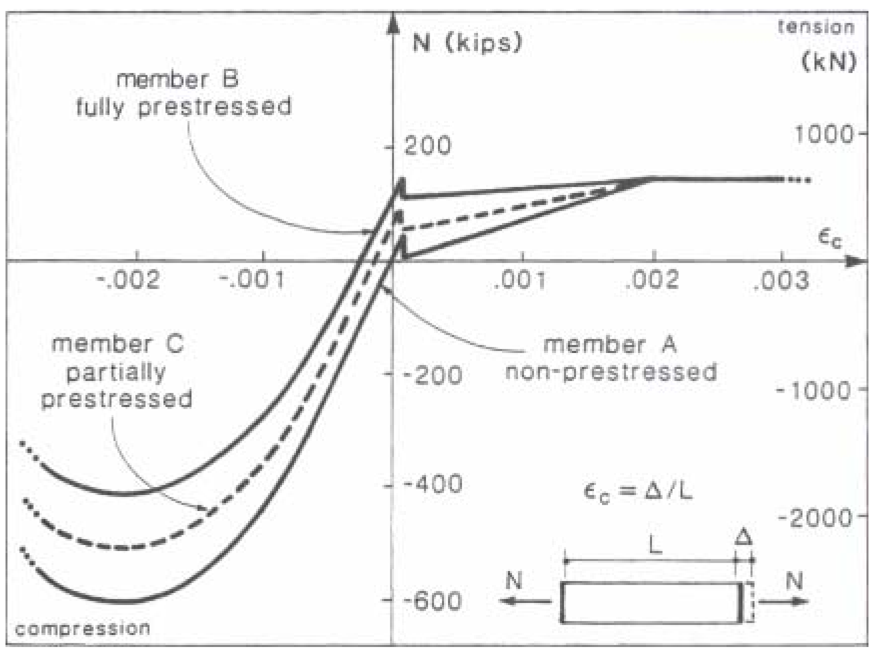
- **Unsupported Length Ratio, b/t:**

  Enter the unsupported length ratio of the reinforcement, where **b** is the effective spacing of the supports (e.g. ties or anchors) and **t** is the diameter of the bar or thickness of the plate.

3.3.1.2 Prestressing Steel

Prestressing is a method by which the steel reinforcement is been tensioned against the concrete. To satisfy the equilibrium of internal stresses, tensile stresses in the steel must be equal to compressive stresses in the concrete. This will cause a pre-compression force in the concrete which increases its cracking load and improves the concrete response in tension. Figure 3-14 compares the axial response of non-prestressed, partially prestressed, and fully prestressed members in tension (Collins and Mitchell, 1997).

In FormWorks, the input parameters for defining Prestressing Steel are the same as Ductile Steel Reinforcement. For more information refer to Section 3.1.1.



3-14 Influence of prestressing on load-deformation response (Collins and Mitchell, 1997).

3.3.1.3 Tension (or Compression) Only Reinforcement

The input parameters are the same as Ductile Steel Reinforcement. For more information see Section 3.1.1.

3.3.1.4 External Bonded FRP Plate/Fabric

For **External Bonded FRP Plate/Fabric,** the input parameters are the same as Ductile Steel Reinforcement except, instead of defining Reinforcement Ratio (As) and Reinforcement Diameter (Db), the Thickness of the plate (or fabric) must be defined.

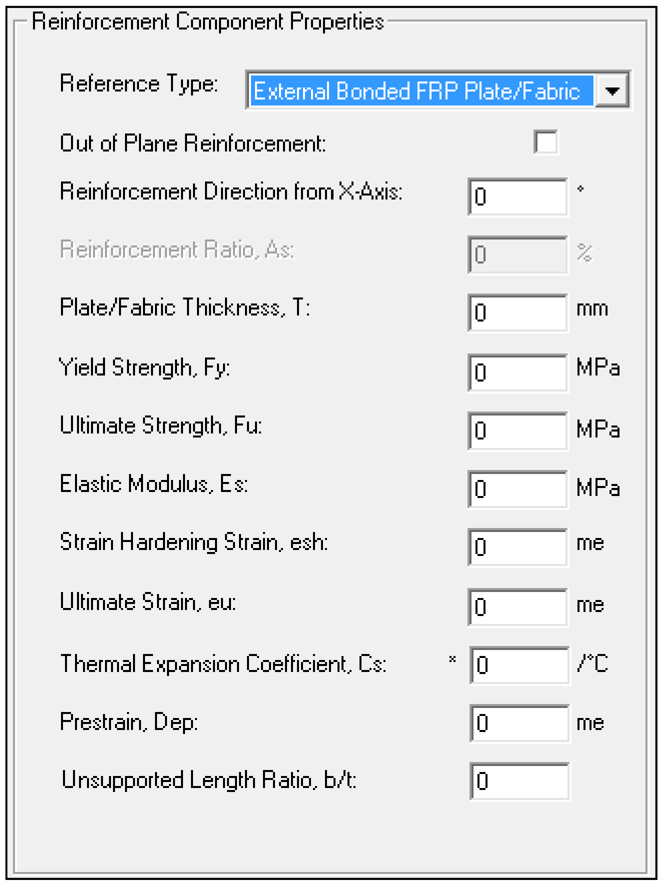


Figure 3-15 – The External Bonded FRP Fabric properties

3.3.1.5 Steel-Fibre-Straight (or Hooked)

The two types of steel fibres that can be modeled in VecTor2 are straight fibres and hooked fibres. The main difference between the two types of fibres is in the quality of bond between the fibres and the concrete matrix. The deformed ends of the hooked fibres enable a stronger bond between the steel fibres and the concrete compared to straight fibres.

The effect of fibres on the behaviour of concrete is dependent on fibre volume content, fibre length, fibre aspect ratio, fibre tensile strength, concrete strength, and fibre orientation. The majority of these are required inputs in FormWorks. To define the Steel-Fibre-Hooked or the Steel-Fibre-Straight components enter the parameters as shown in Figure 3-16.

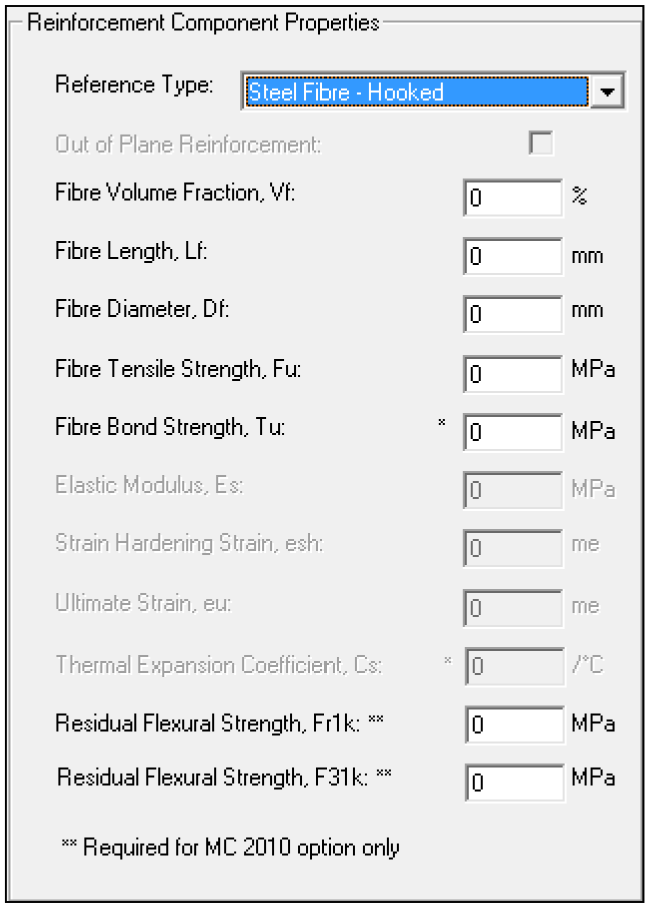


Figure 3-16 – The Steel-Fibre-Hooked (or Straight) properties

- **Fibre Volume Fraction, V_f:**
  Enter the fibre volume fraction, in %.

- **Fibre Length, L_f:**
  Enter the fibre length, in millimetres.

- **Fibre Diameter, D_f:**
  Enter the fibre diameter, in millimetres.

- **Fibre Tensile Strength, F_u:**

  Enter the fibre tensile strength, in MPa.

- **Fibre Bond Strength, T_u:**

  Enter the fibre bond strength, in MPa. Enter zero for the default value, defined as the following:

	Material Type	Component Type	Formula
Fibre Bond Strength (τ_u)	Concrete	Hooked	$2.5 \times 0.33\sqrt{f_c'}$
		Straight	$1.2 \times 0.33\sqrt{f_c'}$
	Mortar	Hooked	$2.0 \times 0.33\sqrt{f_c'}$
		Straight	$1.0 \times 0.33\sqrt{f_c'}$

Table 3-1 – Default values for Fibre Bond Strength, in MPa

Note: If maximum aggregate size ($a_{g,max}$) $\leq 2.0\ mm$ the material is considered as mortar.

- **Residual Flexural Strength:**

  Enter the fibre Residual Flexural Strength (Fr1k and Fr3k), in MPa. This parameter is required only when FRC (fib Model Code 2010) is selected as the Tension Softening Model.

3.3.1.6 Shape Memory Alloy Type 1 and Type 2

Shape Memory Alloy (SMA) materials can be used to replace conventional reinforcing steel under seismic loading conditions, and are useful due to the material's ability to dissipate large amounts of energy without excessive permanent deformation. The hysteresis for conventional reinforcing steel includes large strain offsets; after an earthquake, the structure may be left with a large residual displacement. Shape memory alloys minimize or eliminate these large strain offsets such that after a seismic event, the structure will retain its original shape or the deformations will be much smaller than if conventional reinforcing steel was used.

The idealized behaviour of SMA, with no strain offsets is modeled with Shape Memory Alloy 1. Shape Memory Alloy 1 has a flag-shaped hysteresis, as shown in Figure 3-17.

Developed at the University of Ottawa, the hysteresis for Shape Memory Alloy 2 differs from SMA 1 in that it incorporates strain hardening as well as small strain offsets (Palermo, et al., 2011). In the FormWorks, the input parameters are same as Ductile Steel Reinforcement. For more information see Section 3.1.1.

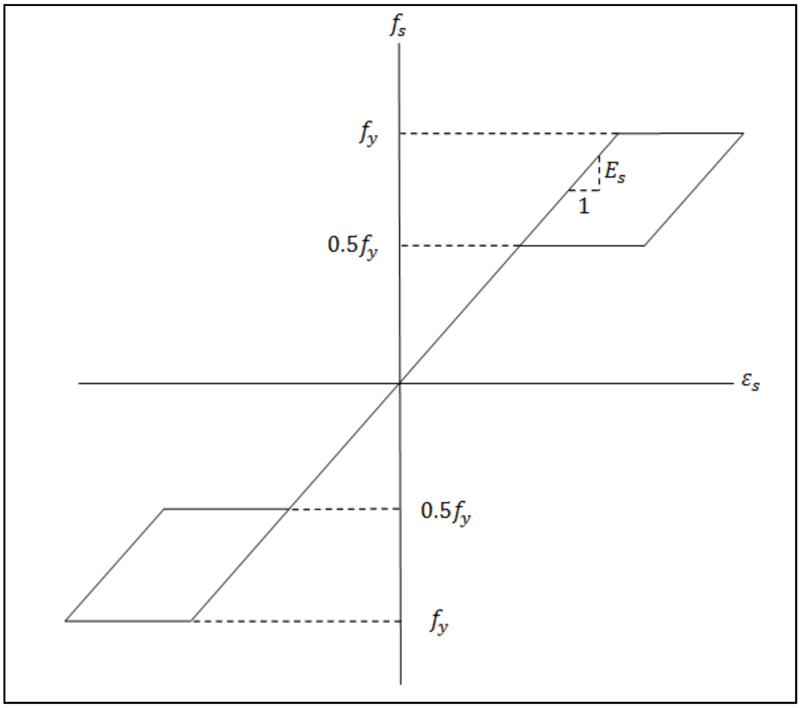


Figure 3-17 – The stress-strain response for Shape Memory Alloy 1

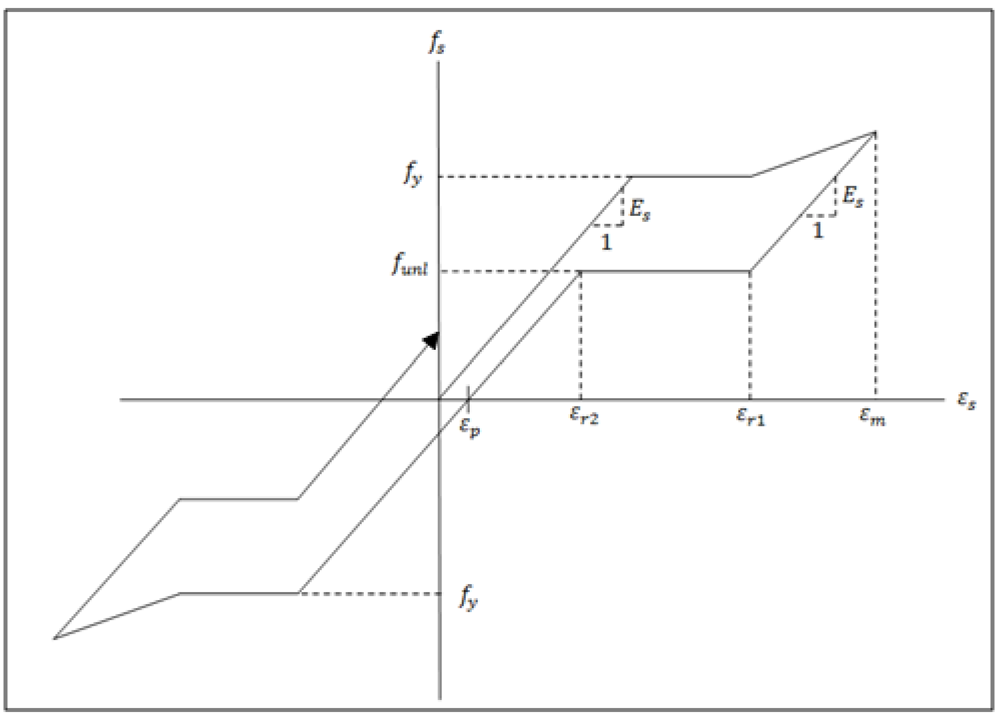


Figure 3-18 – The stress-strain response for Shape Memory Alloy 2

Figure 3-18 illustrates the stress-strain curve and hysteresis for SMA 2.

Where,

f_y = Yield Stress

f_{unl} = Unloading Stress

ε_p = Strain Offset

$\varepsilon_{r1}, \varepsilon_{r2}$ = Reference Strains

ε_m = Maximum Strain

3.3.2 Structural Steel

Structural Steel is modeled in VecTor2 as a linear-elastic material up to the point of yielding, after which plastic deformation and strain hardening occur. Figure 3-19 shows the stress-strain curve for steel material.

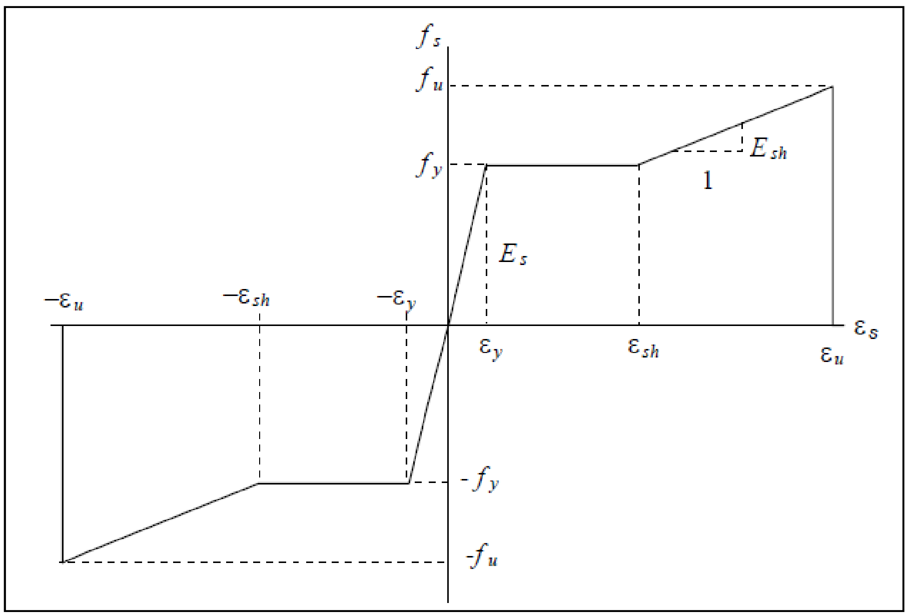


Figure 3-19 –The stress-strain curve for steel material (Collins and Mitchell, 1997).

The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-20 when **Structural Steel** is selected as the main material type. Enter the following properties in the **Concrete Properties** group. Properties marked by * are assigned default values when '0' are entered in their fields.

- **Thickness, T:**
  Enter the out of plane thickness of the steel, in millimetres.

- **Yield Strength, F_y:**

  Enter the stress of the yield plateau, in MPa.

- **Ultimate Strength, F_u:**

  Enter the maximum stress the steel can attain before rupturing, in MPa. The ultimate strength must be greater than the yield strength. The default value is $F_u = 1.5F_y$ MPa.

- **Elastic Modulus, E_s:**

  Enter the stiffness of the initial linear-elastic branch of the stress-strain response, in MPa. The default value is $E_s = 200\ 000$ MPa.

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

  Enter the strain at which the steel stress-strain response begins to ascend from the yield plateau to the ultimate strength, in millistrain. The value must be greater than or equal to the yield strain, Fy/Es. The default value is $e_{sh} = 5$ me.

- **Ultimate Strain, ε_u:**

  Enter the strain at which the steel ruptures, in millistrain. The default value is $e_u = 150$ me.

- **Thermal Expansion Coefficient, C_s:**

  Enter the steel strain increase per temperature increase of $1\ °C$. The default value is 10×10^{-6} /°C.

- **Poisson's Ratio, ν:**

  Enter the initial Poisson's ratio, ν_o, of the steel as a positive value. The default value is $\nu = 0.30$.

- **Density:**

  Enter the mass density of the steel, in kg/m^3. The default value is $\rho = 7850\ \text{kg/m}^3$.

- **Thermal Diffusivity, K_s:**

  Enter the thermal diffusivity of the steel, in mm^2/s.

- **Unsupported Length Ratio, b/t:**

  Enter the unsupported length ratio of the steel.

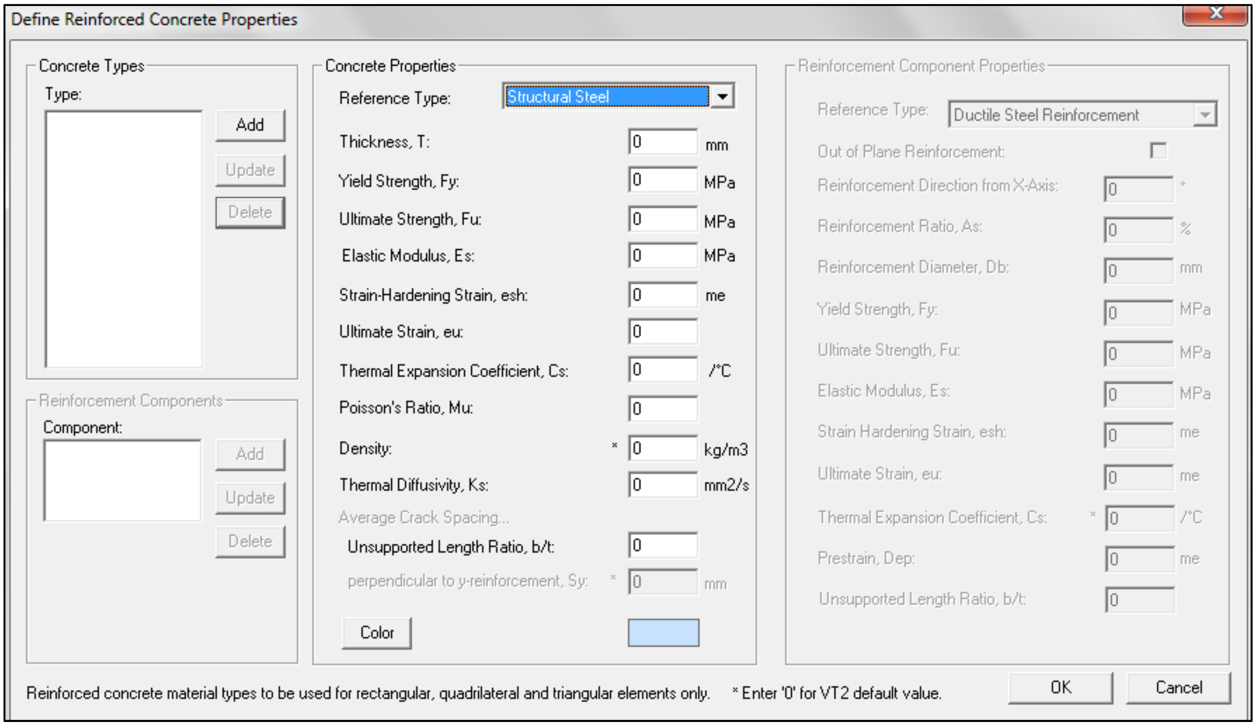


Figure 3-20 – Define Reinforced Concrete Properties window when Structural Steel type is selected as the main material type

3.3.3 Masonry

Masonry is a composite material consisting of masonry units and mortar joints. Masonry is an orthotropic material, due to the geometry and different mechanical properties of the units and joints, as shown in the Figure 3.10. As with the smeared crack approach to the analysis of cracked concrete, for sufficiently large masonry structures, the masonry can be modeled as a continuum with average properties where joint failures are smeared across the single finite element (Lourenco, 1996).

The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-21 when **Masonry** is selected as the main material type. Enter the following properties in the **Concrete Properties** group. Properties marked by * are assigned default values when '0' are entered in their fields.

- **Thickness, T:**
  Enter the out of plane thickness of the masonry, in millimetres. Note that it is the thickness of gross section.

- **Cylinder Compressive Strength, f'_c:**

  Enter the peak compressive stress, in MPa. As explained in the VecTor2 manual, the maximum compressive masonry strength, f_{my}, is entered as the cylinder compressive strength, f_c'.

- **Tensile Strength, f'_t:**

  Enter the uniaxial cracking strength of the masonry, in MPa. The default value is $f_t = 0.33\sqrt{f_c'}$ MPa.

- **Initial Tangent Elastic Modulus, E_c:**

  Enter the tangent stiffness of the masonry stress-strain response at zero-strain, in MPa. According to the Eurocode 6, the default value is $E_c = \frac{2000 f_c'}{\varepsilon_o}$ MPa (if $\varepsilon_o > 0$); $E_c = 1000 f_c'$ MPa (if $\varepsilon_o = 0$).

- **Cylinder Strain at f'_c, ε_o:**

  Enter the compressive strain, ε_o, corresponding to f_c', as a positive value, in millistrain. The default value is $\varepsilon_0 = \frac{2000 f_c'}{E_c}$ (if $E_c > 0$); $\varepsilon_0 = 1.8 + 0.0075 f_c'$ me (if $E_c = 0$).

- **Poisson's Ratio, ν:**

  Enter the initial Poisson's ratio, ν_o, of the masonry as a positive value. The default value is $\nu = 0.15$.

- **Thermal Expansion Coefficient, C_c:**

  Enter the masonry strain increase, per temperature increase of $1\,°C$. The default value is $C_c = 10 \times 10^{-6}$/°C.

- **Maximum Aggregate Size, a:**

  Enter the maximum aggregate size, in millimetres. The default value is a = 20 mm.

- **Density:**

  Enter the mass density of the masonry, in kg/m^3. The default value is $\rho = 2400$ kg/m^3.

- **Thermal Diffusivity, K_c:**

  Enter the thermal diffusivity of the masonry, in mm^2/s. The default value is $K_c = 1.20$ mm^2/s.

- **Joint Spacing:**

  Enter the joint spacing perpendicular to the x-axis for S_x, and perpendicular to the y-axis for S_y. S_x and S_y are defined as the spacings between the head joints and bed joints. See Figure 3-10 for more information.

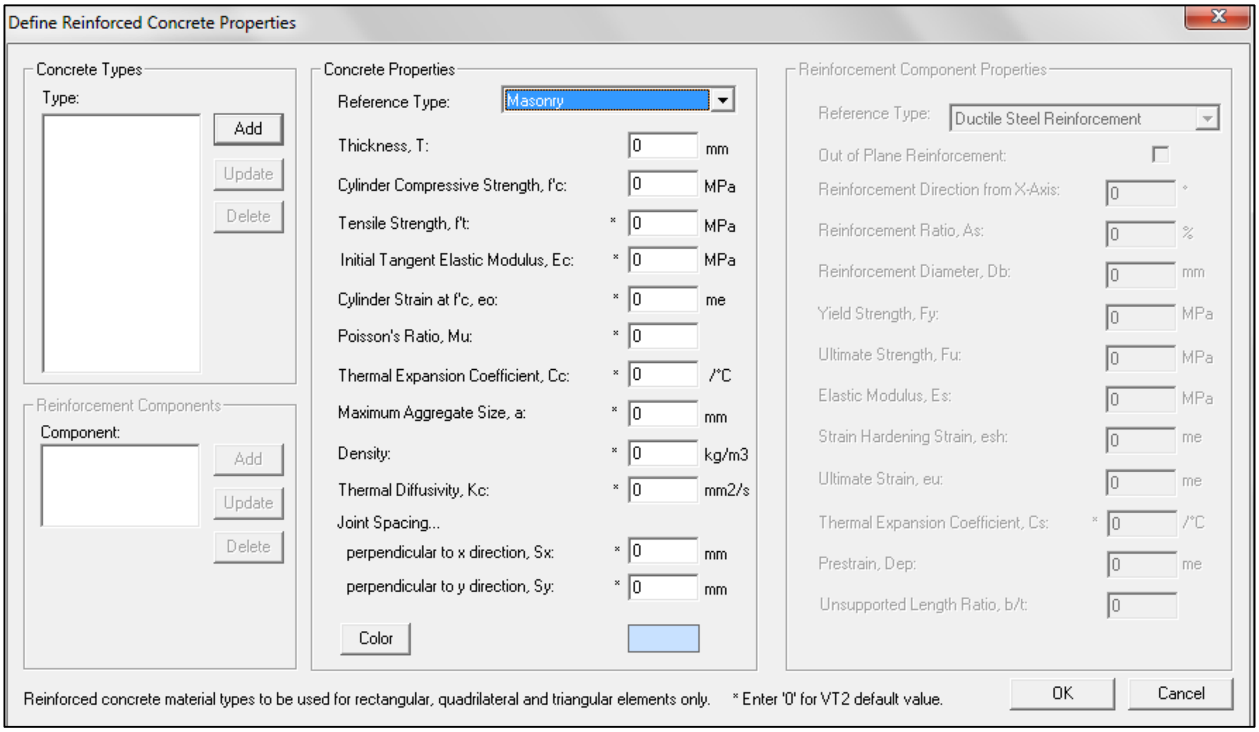


Figure 3-21 –Define Reinforced Concrete Properties window when Masonry type is selected as the main material type

3.3.4 Wood- Fixed Orthotropic

In VecTor2, wood is modeled as a fixed orthotropic material. The two directions of orthotropy are parallel to the grain and perpendicular to the grain. The longitudinal direction is defined as parallel to the grain; the transverse direction is perpendicular to the grain. When modeling wood in the FormWorks, the user must define the properties in both the longitudinal and transverse directions. No default values will be applied by VecTor2 if an input is left blank.

The stress-strain curve for wood is shown in Figure 3-22 (Hasebe & Usuki, 1989).

The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-23 when wood is selected as the main material type. Enter the following properties in the **Concrete Properties** group.

- **Thickness, T:**

  Enter the thickness of the wood in the out-of-plane direction, in millimetres.

- **Longitudinal Direction from x-axis:**

  Enter the longitudinal direction of the wood, defined counterclockwise from the x-axis, in degrees. In wood material models, the longitudinal direction is the direction of the grain of the wood.

- **Compressive Strength-Longitudinal, $\mathbf{f'_{c-l}}$:**

  Enter the compressive strength in the longitudinal direction, in MPa.

- **Compressive Strength-Transverse, $\mathbf{f'_{c-t}}$:**

  Enter the compressive strength in the transverse direction, perpendicular to the direction of the grain, in MPa.

- **Tensile Strength-Longitudinal, $\mathbf{f'_{t-l}}$:**

  Enter the tensile strength in the longitudinal direction, in MPa.

- **Tensile Strength-Transverse, $\mathbf{f'_{t-t}}$:**

  Enter the tensile strength in the transverse direction, perpendicular to the direction of the grain, in MPa.

- **Shear Strength:**

  Enter the shear strength, v_{lt}, in MPa. This is the shear strength parallel to the grain; that is, the longitudinal shear strength of the wood. The shear strength varies depending on the type of wood, and the water content of the wood.

- **Density:**

  Enter the density of the wood, in kg/m^3.

- **Elastic Modulus-Longitudinal, $\boldsymbol{E_l}$:**

  Enter the elastic modulus for the longitudinal direction, in MPa.

- **Elastic Modulus-Transverse, $\boldsymbol{E_t}$:**

  Enter the elastic modulus for the transverse direction, in MPa.

- **Poisson's Ratio-Long. Stress-Trans. Strain, $\boldsymbol{\nu_{lt}}$:**

  Specify the initial Poisson's ratio for longitudinal stress-transverse strain.

- **Poisson's Ratio-Trans. Stress-Long. Strain, $\boldsymbol{\nu_{tl}}$:**

Specify the initial Poisson's ratio for transverse stress-longitudinal strain.

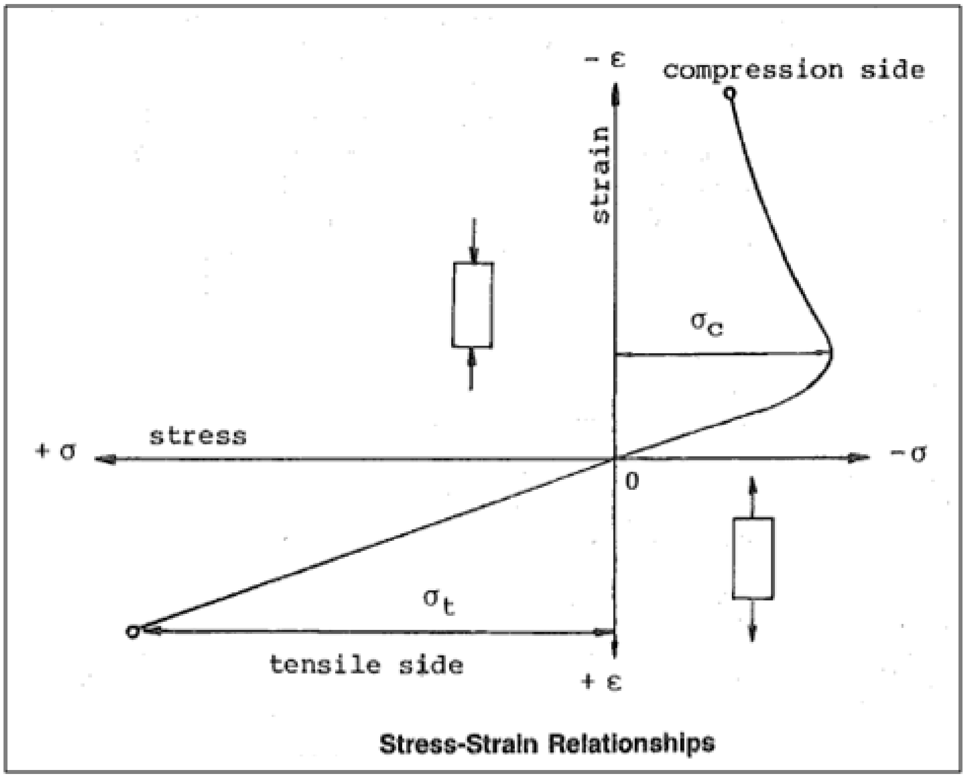


Figure 3-22 – The stress-strain curve for wood material (Hasebe & Usuki, 1989)**.**

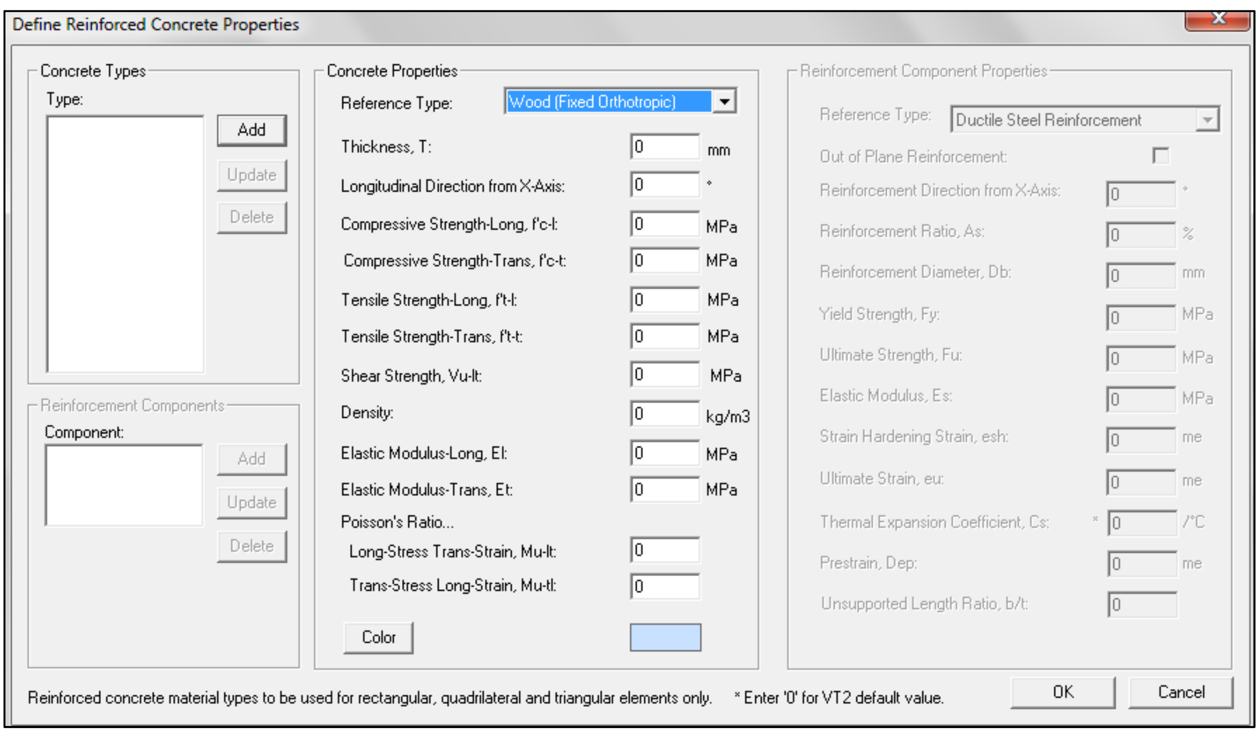


Figure 3-23 – The Define Reinforced Concrete Properties window when Wood type is selected as the main material type

3.3.5 Concrete-Steel Laminate

Steel-concrete composite wall elements typically consist of a thick concrete core integrated with two thin steel faceplates. Forces are generally transferred between the concrete and the steel mainly by shear studs. In VecTor2, the DSFM is the basis for the analysis of Concrete-Steel Laminates. As VecTor2 is a 2D finite element program, the different materials must be modeled as a smeared/combined element with properties representative of the two materials, as outlined below and described fully by Vecchio and McQuade (2011).

A typical Concrete-Steel Laminate is illustrated in Figure 3-24.

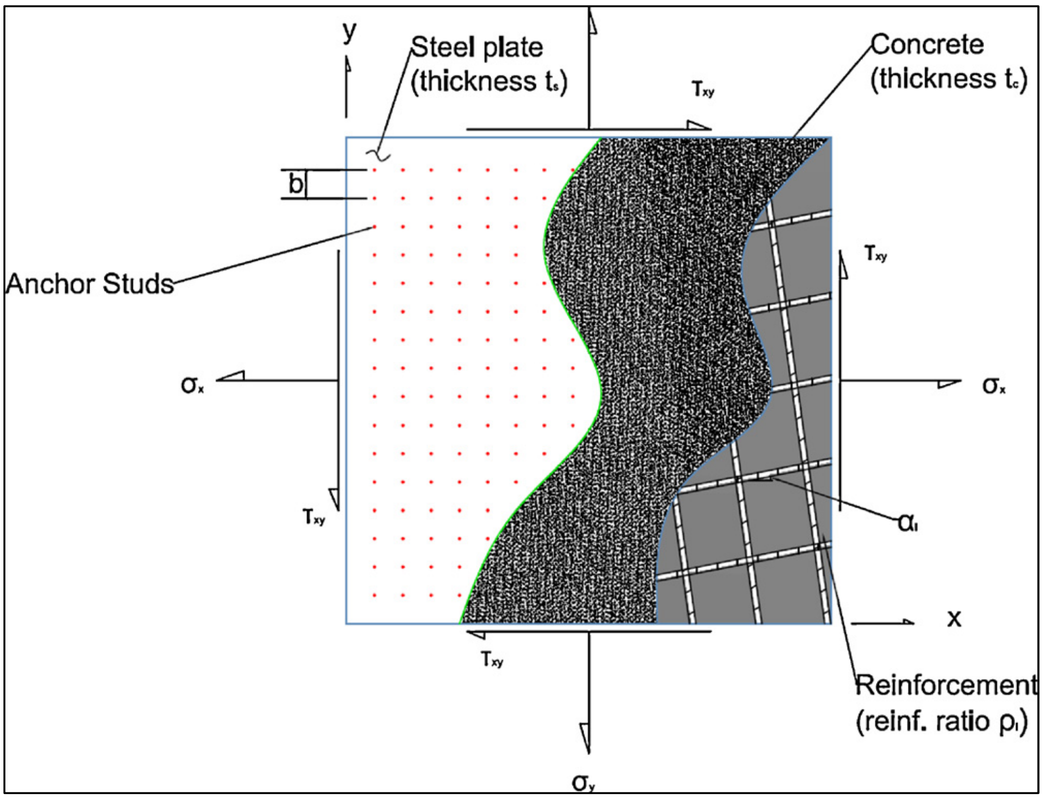


Figure 3-24 –A typical Concrete-Steel Laminate (Vecchio & McQuade, 2011)

When using this concrete type, the **Steel Skin Plate** must also be specified as one of the types of reinforcements. After the two materials, the concrete and reinforcement, are defined in FormWorks, VecTor2 creates a combined material stiffness which is used for the analysis. Regular **Ductile Steel Reinforcement** can also be specified as a reinforcement component for the concrete in the Concrete-Steel Laminate, although it is not required. If specified, the ductile steel reinforcing bars will also be incorporated into the combined material stiffness matrix. In general, when using the concrete-steel laminate material type, no embedded reinforcement is used.

The **Define Reinforced Concrete Properties** dialog box appears as shown below when **Concrete-Steel Laminate** is selected as the main material type.

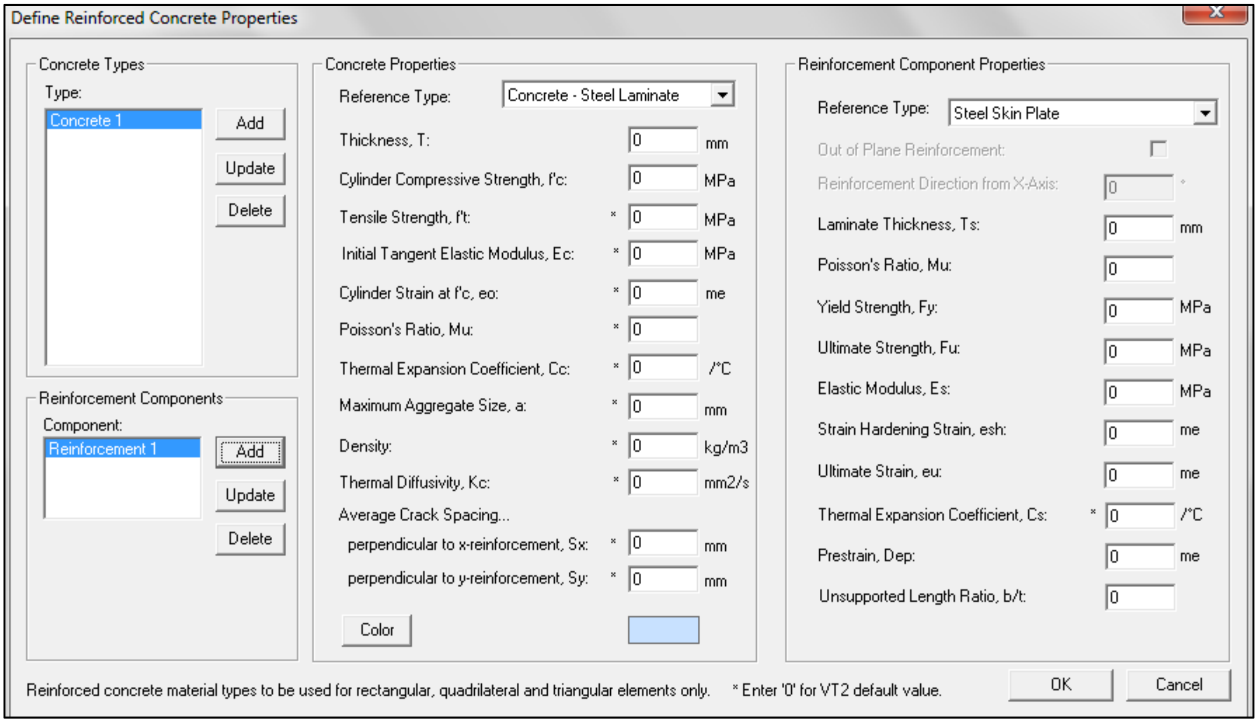


Figure 3-25 – The Define Reinforced Concrete Properties window when the main material type is Concrete Steel Laminate and its component Steel Skin Plate

The same concrete properties must be entered as are required to define regular Reinforced Concrete. Refer to Section 3.1 for the required concrete inputs. Note that that the thickness entered in the **Concrete Properties** section is the thickness of the concrete core only and does not include the Steel Skin Plate, the thickness of which is specified in the **Reinforcement Component Properties** section.

For the most part, the input properties required for the Steel Skin Plate are similar to the input requirements for the other Steel Reinforcement types. For further instruction on how to enter these parameters, see Section 3.1. The input parameters unique to the Steel Skin Plate reinforcement are:

- **Laminate Thickness, T_S:**

  Enter the thickness of one of the steel plates, in millimetres.

- **Poisson's Ratio, ν:**

  Enter the initial Poisson's ratio, ν_o, for the steel plate.

3.3.6 Concrete-SFRC Laminate

Similar to the Concrete-Steel Laminate, a Concrete-SFRC Laminate combines a concrete core with steel-fibre reinforced concrete faceplates. Using an SFRC laminate will result in a higher capacity and a more ductile response. A common application of the SFRC laminate is in strengthening of RC slabs, whereby a thin layer of SFRC is attached to an existing RC slab. There are also benefits in shear capacity. An overlay of SFRC allows for increased post-cracking residual stress as the steel fibres are efficient at controlling large cracks in most circumstances (Bonaldo, Barros, & Lourenco, 2005).

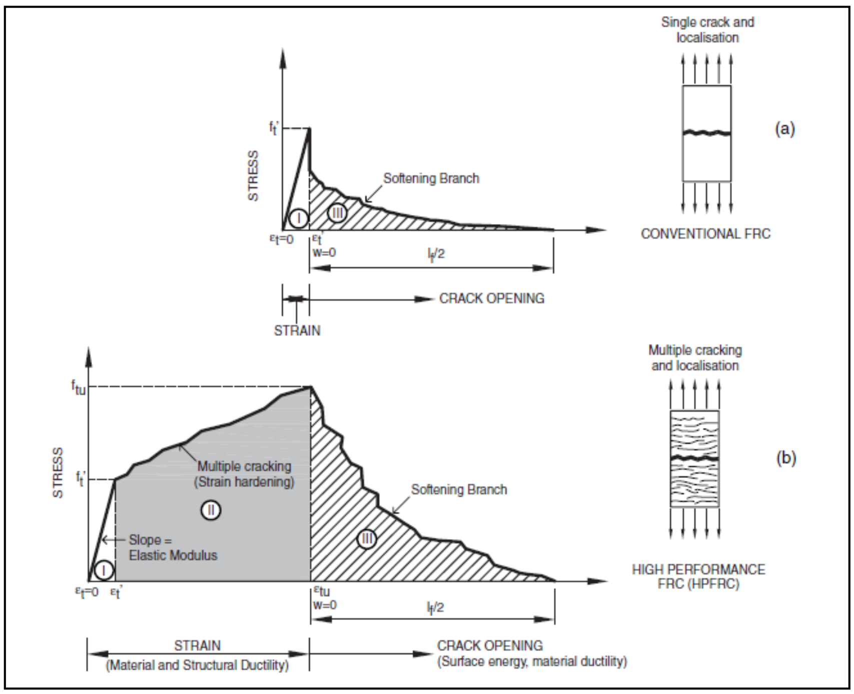


Figure 3-26 –Comparison of typical tensile stress-strain response of fibre reinforced concrete containing: (a) low fibre volume content, and (b) high fibre volume content (Naaman, 2003).

To define Concrete-SFRC Laminate properly in FormWorks, in the Define Reinforced Concrete Properties tab, in the Concrete Properties subgroup, the reference type must be set to **Concrete-SFRC Laminate**. In the Reinforcement Component Properties subgroup, the reference type must be set to either **SFRC Laminate-Hooked Fibre** or **SFRC Laminate-Straight Fibre**. The difference between the two fibre types is discussed briefly in Section 3.1 of this Chapter. The user can also choose to include normal ductile reinforcing steel in the concrete, although this is not required.

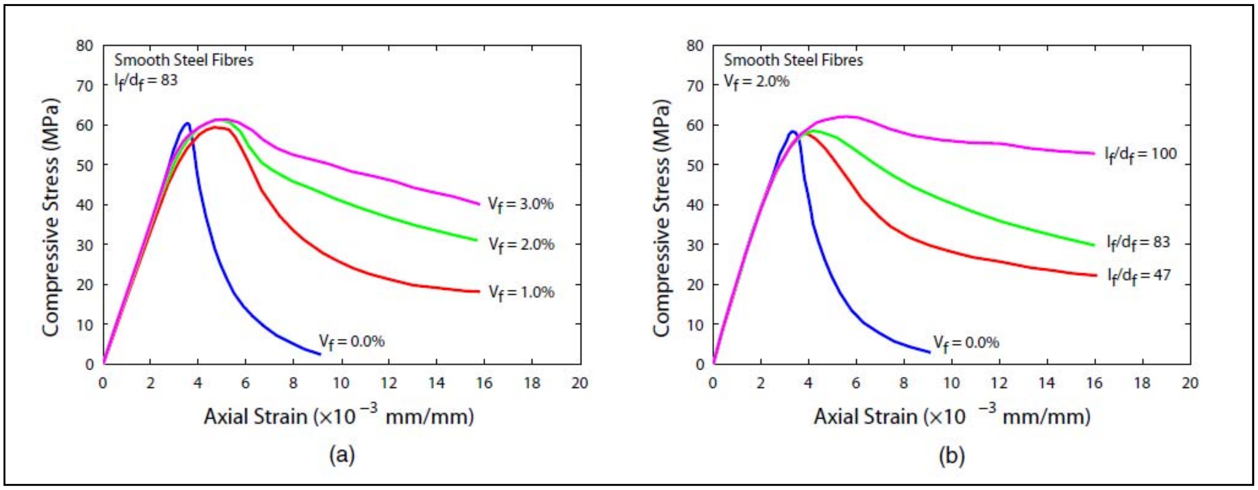


Figure 3-27 – The compressive stress-strain curve of steel fibre reinforced concrete: (a) influence of fibre content, and (b) influence of fibre aspect ratio (Fanella and Naaman, 1985).

The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-28 when Concrete-SFRC Laminate is selected as the main material type.

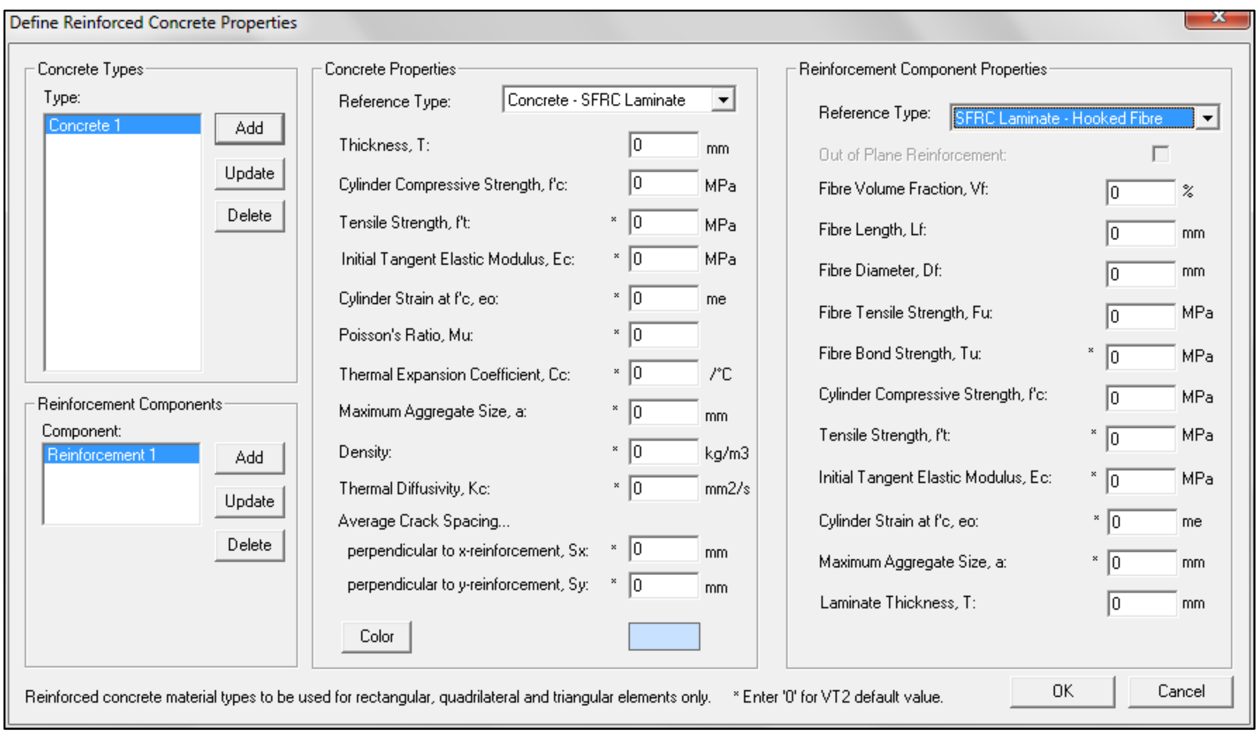


Figure 3-28 – Define Reinforced Concrete Properties window when the main material type is Concrete – SFRC Laminate

The same concrete properties must be entered as are required to define Reinforced Concrete. Refer to Section 3.1 for the required concrete inputs.

Enter the following properties in the **Reinforcement Component Properties** group. Properties marked by * are assigned default values when '0' are entered in their fields. The same properties must be entered regardless of whether a Hooked Fibre or Straight Fibre laminate is used.

- **Fibre Volume Fraction, V_f:**
  Enter the fibre volume fraction as a percentage (%).

- **Fibre Length, L_f:**
  Enter the fibre length used in the SFRC, in millimetres.

- **Fibre Diameter, D_f:**
  Enter the fibre diameter, in millimetres.

- **Fibre Tensile Strength, F_u:**
  Enter the fibre tensile strength, in MPa. The default value is $F_u = 1100$ MPa.

- **Fibre Bond Strength, T_u:**
  Enter the fibre bond strength, in MPa. Enter zero for the default value defined as Table 3-1.

- **Cylinder Compressive Strength, f'_c:**
  Enter the peak compressive stress, in MPa.

- **Tensile Strength, f'_t:**
  Enter the uniaxial cracking strength of the SFRC, in MPa. The default value is $f'_t = 0.33\sqrt{f'_c}$MPa.

- **Initial Tangent Elastic Modulus, E_c:**
  Enter the tangent stiffness of the SFRC stress-strain response at zero-strain, in MPa. The default value is $E_c = 3320\sqrt{f'_c} + 6900$ MPa.

- **Cylinder Strain at f'_c, ε_o:**
  Enter the compressive strain, ε_o, corresponding to f'_c, as a positive value, in millistrain. The default value is $\varepsilon_0 = 1.8 + 0.0075f'_c$ me.

- **Maximum Aggregate Size, a:**
  Enter the maximum aggregate size, in millimetres. The default value is $a = 20$ mm.

- **Laminate Thickness, T:**
  Enter the SFRC laminate thickness, in millimetres.

3.3.7 Masonry-SFRC Laminate

The Masonry-SFRC Laminate is defined in FormWorks in basically the same way as the Concrete-SFRC Laminate. In the Define Reinforced Concrete Properties tab, in the Concrete Properties subgroup, the reference type must be set to **Masonry-SFRC Laminate**. In the Reinforcement Component Properties subgroup, the reference type must be set to either **SFRC Laminate-Hooked Fibre** or **SFRC Laminate-Straight Fibre**. The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-29 when Masonry-SFRC Laminate is selected as the main material type.

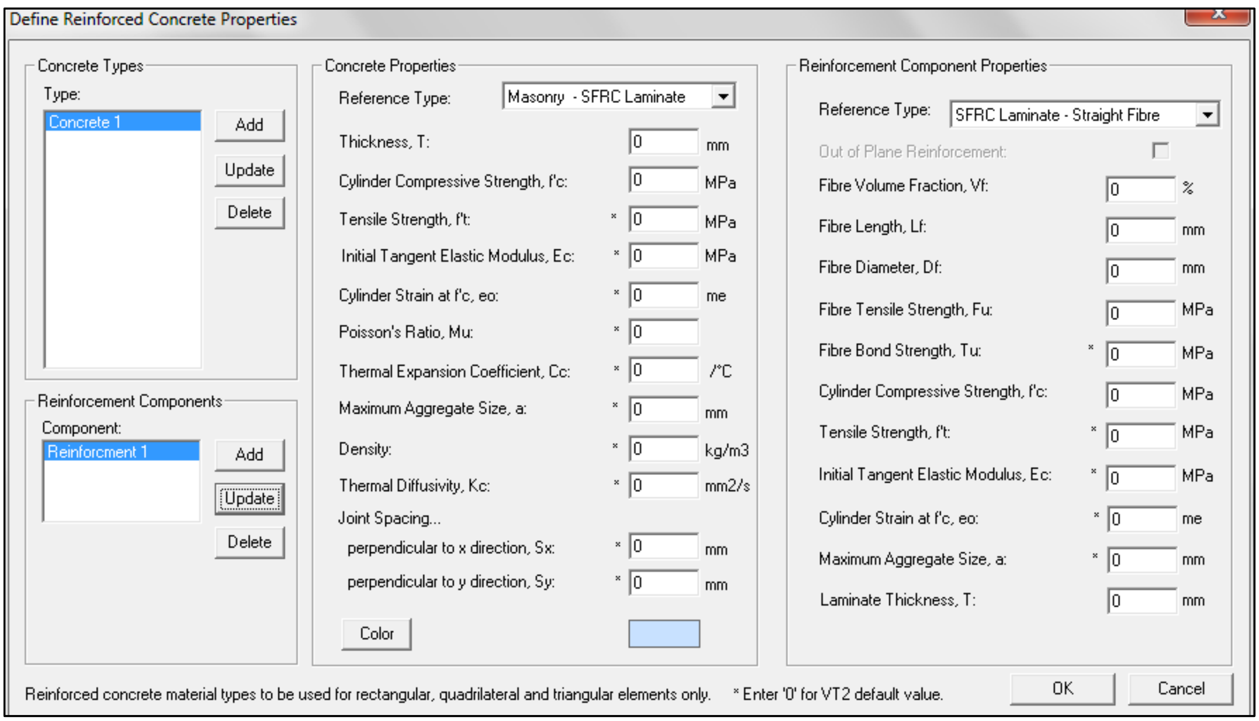


Figure 3-29 – Define Reinforced Concrete Properties window when the main material type is Masonry – SFRC Laminate

The same masonry properties must be entered as are required to define regular reinforced or unreinforced masonry. See Section 3.3 for the required inputs. The SFRC Laminate properties are entered as they are entered for a Concrete-SFRC Laminate. See Section 3.6 for more information about the required parameters.

3.3.8 Concrete-Ortho Laminate

The Concrete-Ortho Laminate is a combination of a concrete core and wood (or other orthotropic material) faceplates. Concrete-wood laminates are commonly used in both floor and beam construction. In new floor construction, solid concrete is typically placed on timber floor beams or a solid layer of wood. The wood layer functions to replace the cracked concrete-steel reinforcement section of a solid concrete slab, and also reduces the need for formwork. Similarly, deep beams benefit from concrete-wood composite construction, as wood can help reduce or eliminate the high tensile stresses in the concrete. Hence, bridges can also utilize composite concrete-timber decks. As with other laminates, forces must be transferred between the concrete and the wood, most likely through shear studs. For the stress-strain behaviour of orthotropic wood, see Figure 3-22 (Gutkowski, et al., 2010).

In the Define Reinforced Concrete Properties tab, in the Concrete Properties subgroup, the reference type must be set to **Concrete-Ortho Laminate**. In the Reinforcement Component Properties subgroup, the reference type must be set to **Orthotropic Laminate**. The **Define Reinforced Concrete Properties** dialog box appears as shown in Figure 3-30 when Concrete-Ortho Laminate is selected as the main material type.

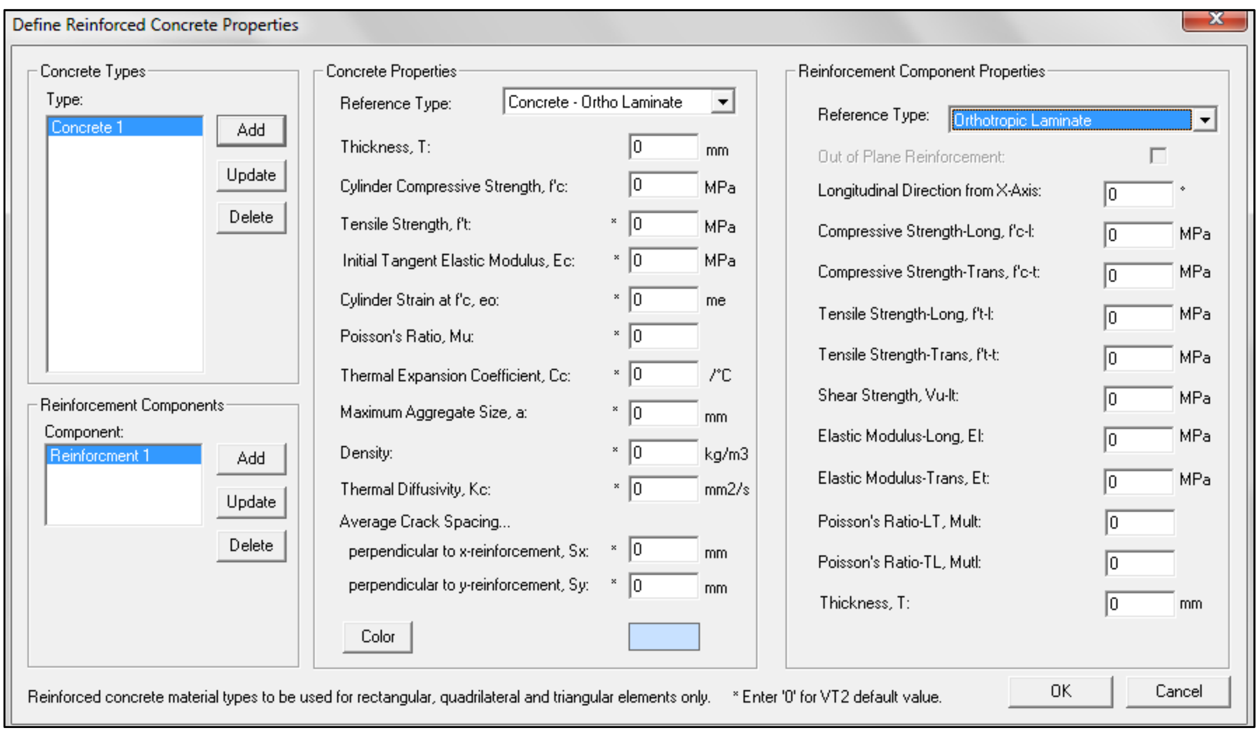


Figure 3-30 – Define Reinforced Concrete Properties window when the main material type is Concrete–Ortho Laminate

The same concrete properties must be entered as are required to define regular reinforced concrete. See Section 3.1 for the required concrete inputs.

In Reinforcement Component Properties group, the inputs required are the same as for the Wood (Fixed Orthotropic) material type, with the exception of density. The wood density is not a required input for Orthotropic Laminate. The required inputs are specified in Section 3.3.4.

Chapter 4

Modeling 3D Solid Structures

4.1 Introduction

VecTor3 is a nonlinear finite element analysis (NLFEA) program for modeling 3D reinforced concrete solid structures. The theoretical bases of VecTor3 are the Modified Compression Field Theory (Vecchio and Collins, 1986) and the Disturbed Stress Field Model (Vecchio, 2000). The program uses a smeared, rotating-crack formulation to analyze the structures. To produce a stable and robust nonlinear solution, VecTor3 uses a total-load iterative procedure with a secant stiffness formulation.

Incorporated into the program's analysis algorithms are material nonlinearity effects including compression softening due to transverse cracking, tension stiffening, shear slip along crack surfaces, and other mechanisms important in making a precise assessment of cracked reinforced concrete behaviour. In addition, the program is capable of modeling nonlinear expansion and confinement, cyclic loading, effects of slip distortion on element compatibility relations, and application to the analysis of repaired or rehabilitated structures. To consider each of the above mentioned constitutive responses and mechanisms properly, the program supports a wide range of material models (Selby, 1990).

To create finite element models, VecTor3 utilizes a fine mesh of low-powered elements. The program's element library includes an 8-node brick element (24 d.o.f.), a 6-node wedge element (18 d.o.f.), and a 2-node truss bar element (6 d.o.f.). The solid elements can be used to model reinforced concrete structures. Reinforcement can be modeled either as smeared within the solid elements, or as discrete bars using the truss elements. Calculating the stiffness matrix coefficients explicitly for each of the elements makes the computational time shorter and avoids numerical instabilities such as zero energy nodes or shear locking in the program analysis (Selby, 1990).

VecTor3 is currently configured to accommodate: 12000 elements, 15000 nodes, 45 concrete material types, 15 steel material types, and 4 smeared reinforcement components per material type.

Previous versions of FormWorks were only compatible with VecTor2. The remaining VecTor programs did not have pre-processors and they ran in a DOS environment from Fortran executable files; standard text editors were used to create input data files. A new version of FormWorks, FormWorks-Plus, has been developed to create finite element models compatible with the remaining VecTor programs including VecTor3. FormWorks-Plus is able to create 3D solid finite element models, apply different types of loads, assign material properties, and specify analysis parameters which are compatible with VecTor3. This chapter will discuss the different parts of FormWorks-Plus that are involved in modeling a 3D solid structure to be analyzed by VecTor3.

4.2 The Job Data

For the most part, the input properties required to define the Job Page for the 3D Solid Structures are similar to the input requirements for the 2D Plane Membrane Structures. For further instruction on how to enter these parameters, see Section 2 in Chapter 3. Figure 4-1 and 4-2 show the analysis parameters in the Job Control Page and Auxiliary Page unique to VecTor3.

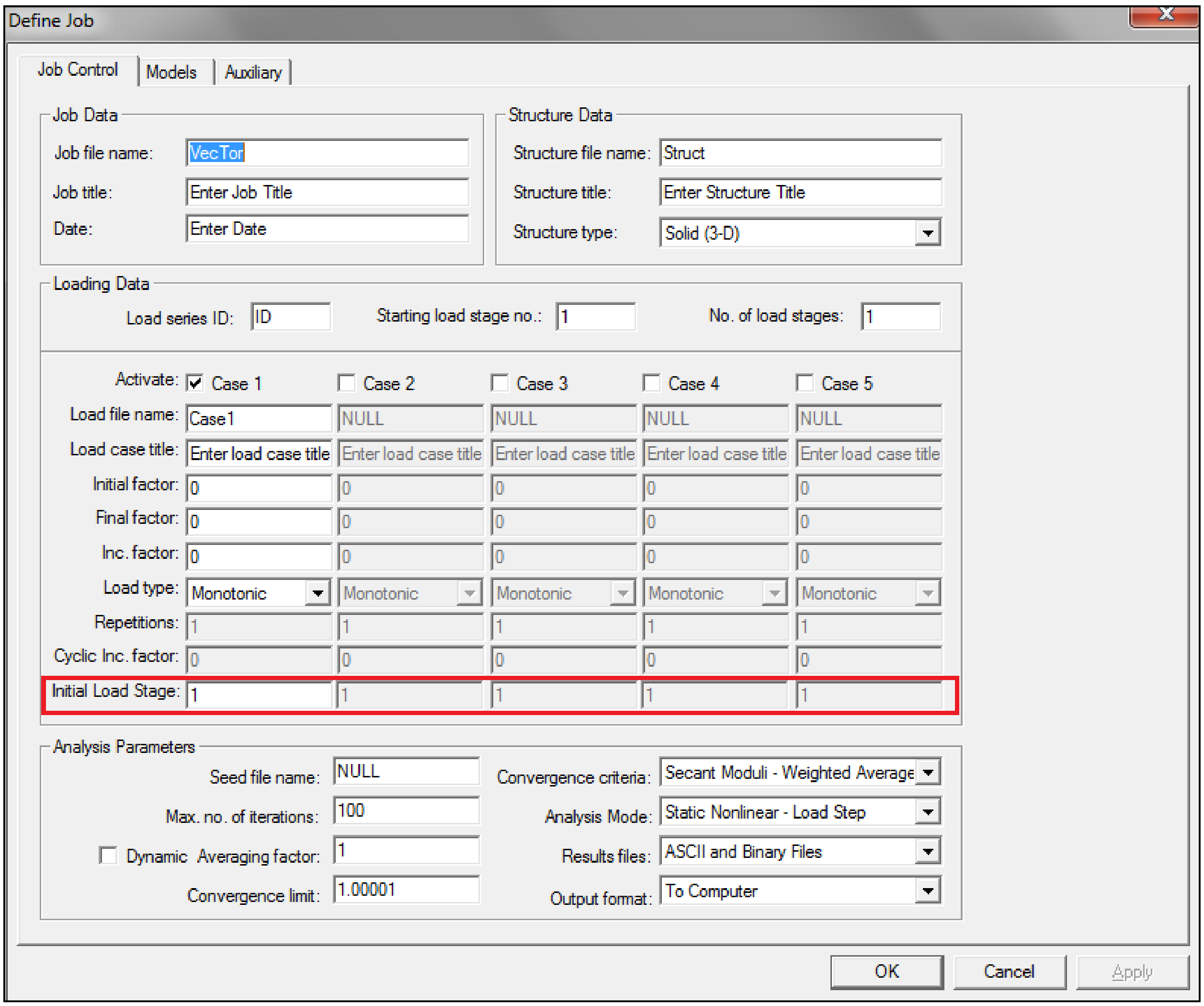


Figure 4-1 – The Job Control Page for 3D Solid Structures

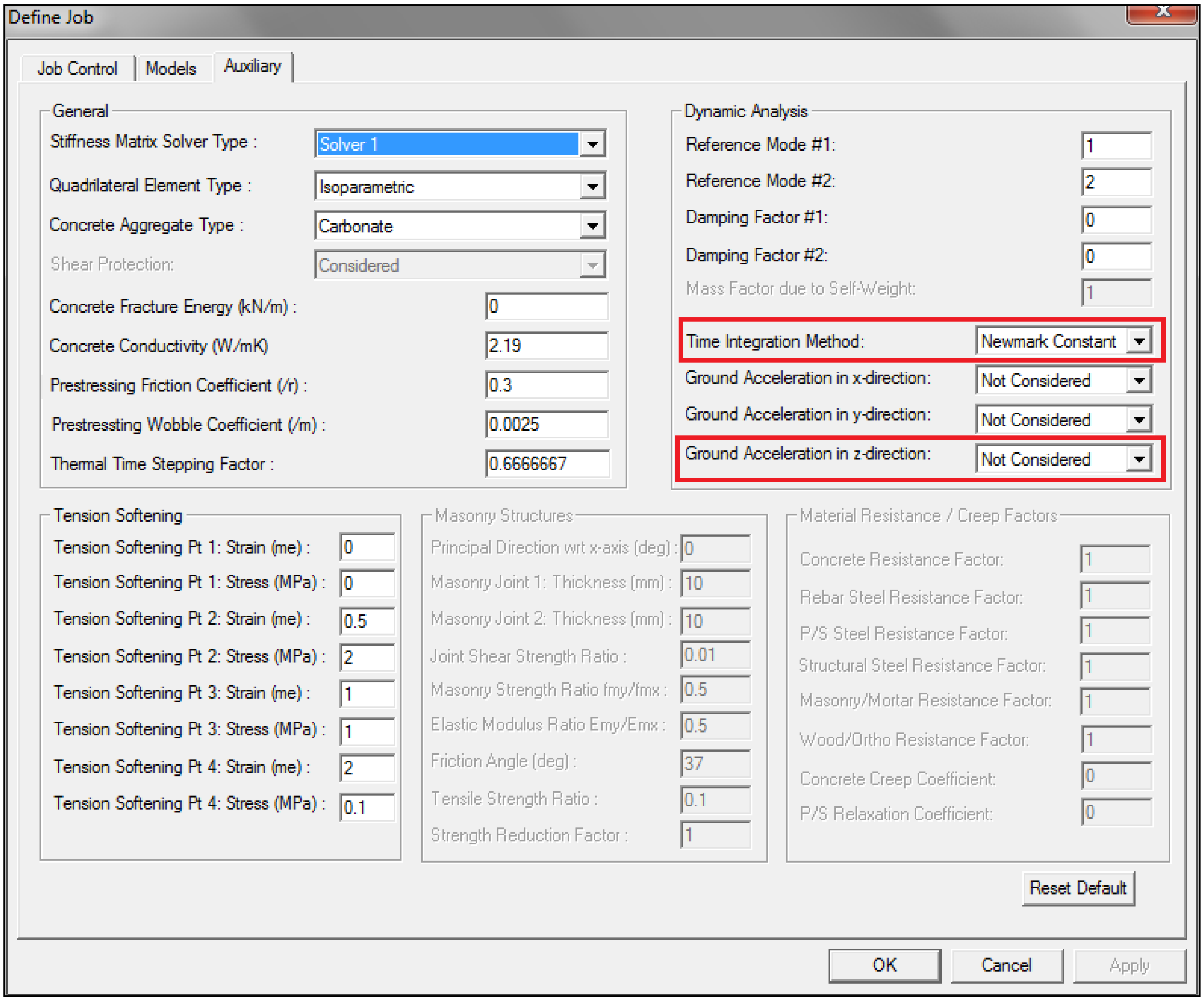


Figure 4-2 – The Auxiliary Page for 3D Solid Structures

The input parameters unique to the 3D Solid Structures are:

- **Initial Load Stage**

  In the Job Control Page, Enter the load stage number from which the load case should be activated. The default value is 0.

- **Time Integration Method**

  In the Auxiliary Page, Select one of the following Time Integration methods for Dynamic Analysis. As shown in the following, each method has its own Newmark Beta Factor (β) and Gamma Factor (γ). The default option is the Newmark Constant method.

1. Newmark Constant: $\beta = 0.25$ and $\gamma = 0.5$.
2. Newmark Linear: $\beta = 1/6$ and $\gamma = 0.5$.
3. Newmark Beta: $\beta = 0.25$ and $\gamma = 0.5$.

- **Ground Acceleration in z-direction**

  In the Dynamic Analysis part of the Auxiliary Page, Choose 'Considered' or 'Not Considered' for the ground acceleration in the z-direction. The Considered option is to be chosen when the Analysis Mode on the Job Control page is set to Dynamic Nonlinear—General.

Note that although all the input parameters in the Models Page are similar to the 2D Plane Membrane Structures, each drop list contains models which are only applicable to the 3D Solid Structures.

4.3 The Structure Data

After defining the analysis parameters and behavioural models in the Job Data Page, the next step is to define the Structure Data which creates the structure file with a *.S3R extension. This file contains all the information about the finite element mesh including material properties, node numbers and coordinates, elements, and restraints.

The first step is to simplify the actual structure so it can be modeled easily with FormWorks-Plus using cubic and wedge elements. In defining complex finite element models, drawing a sketch can help to have a better understanding of the model. The sketch should include the simplified geometry of structure, different reinforced concrete material types, the location of reinforcements and boundary conditions.

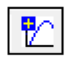
Currently FormWorks-Plus is not capable of meshing 3D Solid Structures automatically; only the manual method is available. Although defining the nodes and elements manually is time consuming, it gives complete control over the mesh topology and its computational characteristics to the user. To define the finite element mesh the following steps should be taken:

1. Define material properties
2. Specify nodal coordinates
3. Define elements
4. Assign materials to elements
5. Define restraints

There are several parameters that can affect the finite element mesh such as element types, location of loads, placement of restraints, number of material types, the required degree of accuracy and processing limits (Wong, 2002). One of the important parameters in meshing the structure is the size of elements. VecTor3 uses the formulation of the Modified Compression Field Theory to analyze structures. Elements should be sized so that the distributions of the stress and crack stay uniform within elements. Checking the analysis results and stress variations in elements may indicate the need for refining the mesh.

4.3.1 Specifying Material Properties

VecTor3 includes two types of materials: reinforced concrete and reinforcement.

4.3.1.1 Reinforced Concrete 

For the most part, the input properties required to define the reinforced concrete material in VecTor3 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, see the FormWorks manual. The input parameters unique to VecTor3 are:

- **Average Crack Spacing perpendicular to z-reinforcement:**
  Enter the crack control parameter indicating the spacing of cracks planes perpendicular to the z-axis for **Sz,** in millimeters. The default value is computed by the CEB-FIP model, unless the Tension Chord (Kauffmann) model is selected for tension stiffening.

- **Reinforcement Orientation:**
  In VecTor3, instead of defining the “Out-of-Plane Reinforcement” and “Reinforcement Direction from X-Axis”, direction cosines are used to define the orientation of smeared reinforcement.

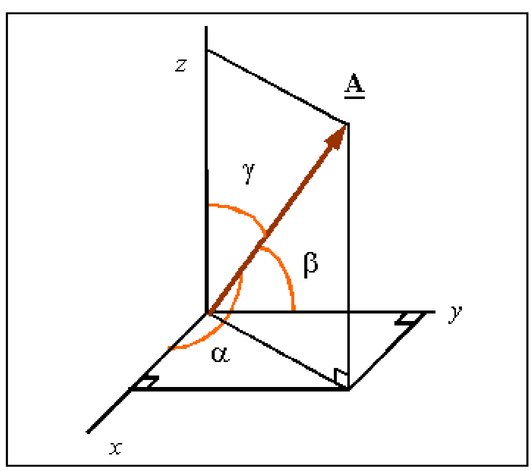


Figure 4-3 – Cosine directions for defining reinforcement orientation

k, **l**, and **m** are direction cosines, defined as the following:

$k = cos\ (\alpha)$

$l = cos\ (\beta)$

$m = cos\ (\gamma)$

Figure 4-3 demonstrates the definition of the cosine directions. For instance, to define a smeared reinforcement in the X-direction the input parameters are k = 1, l = 0, and m = 0.

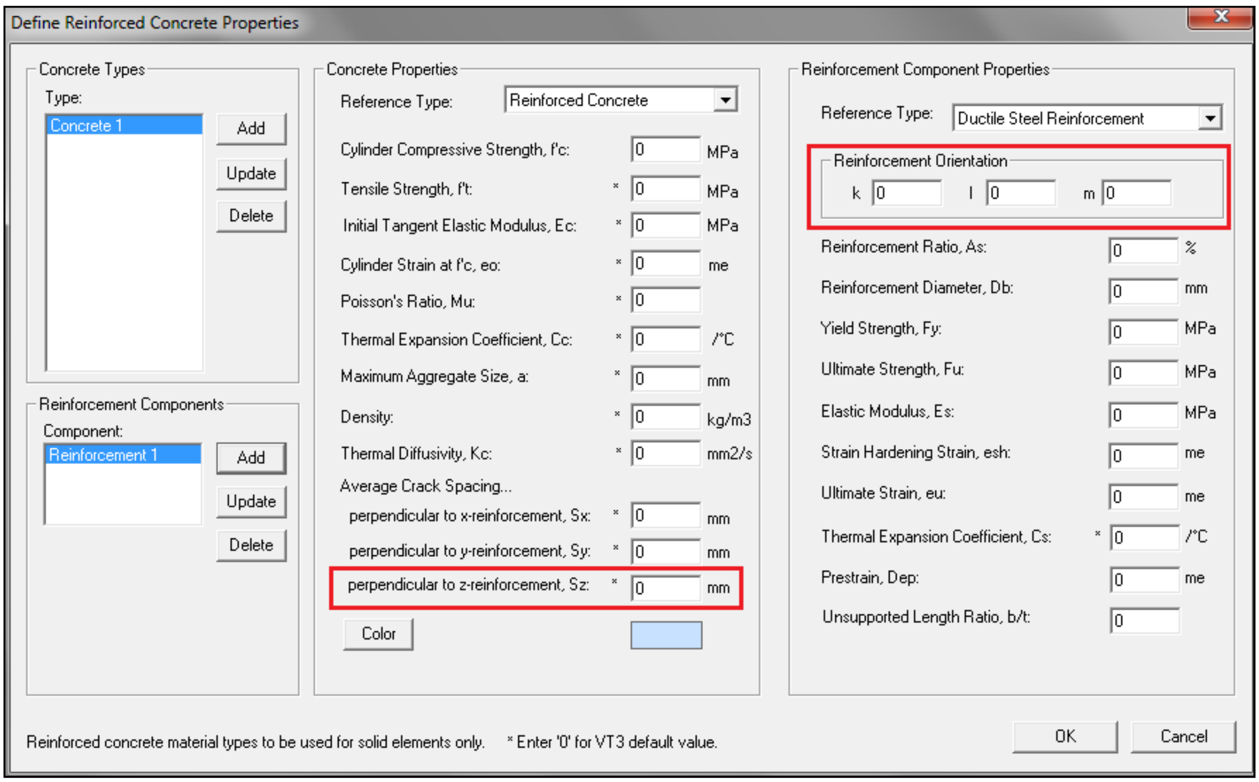
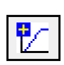
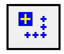


Figure 4-4 – Define Reinforced Concrete Properties window for 3D Solid Structures

4.3.1.2 Reinforcement 

The input properties required to define discrete reinforcement in VecTor3 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, see the FormWorks manual.

4.3.2 Defining Nodes 

Select the **Structure/Create Nodes** menu item or click the **Create Node**s toolbar button. The **Create Nodes** dialog box appears as shown in Figure 4-5. This section explains the procedures to add or delete nodes. There are two rules for defining the nodes in VecTor programs. The first is that all the nodes of a finite element mesh must be numbered consecutively starting from 1. The second rule is that all the nodes must be attached to at least one element.

FormWorks-Plus checks both rules to make sure all nodes are defined correctly. For the first rule the program allows nodes to be arbitrarily numbered, but when generating the structure file it checks the node numbers and if the rule is violated FormWorks-Plus renumbers the nodes automatically. For the second rule, if there is any node which is not attached to an element FormWorks-Plus sends a warning message to the user.

Renumbering the nodes is a useful feature which allows modifying an existing mesh instead of defining a new one. But it is recommended that the user follows the above mentioned rules in defining the nodes, and not rely on the renumbering feature whenever possible.

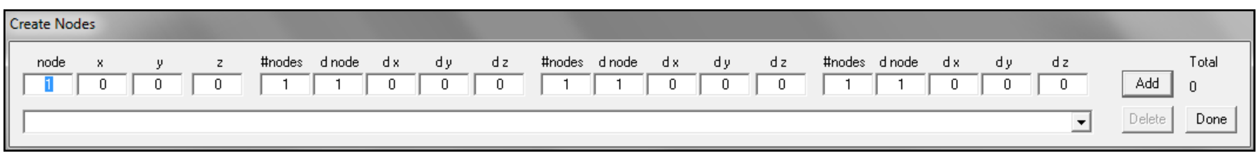


Figure 4-5 – Create Nodes window for 3D Solid Structures

The coordinates of each node must be defined in the Create Nodes window. Since creating nodes one by one takes considerable time, FormWorks-Plus has a node increment feature which helps to define several nodes in the X, Y, and Z directions simultaneously. The following example shows how to use node increment feature.

Node	X	Y	Z	# nodes	d nodes	dx	dy	dz	# nodes	d nodes	dx	dy	dz	# nodes	d nodes	dx	dy	dz
1	50	75	100	4	1	100	0	0	3	4	0	50	0	2	12	0	0	200

Figure 4-6 (A) – A simple example of node increment feature - input data

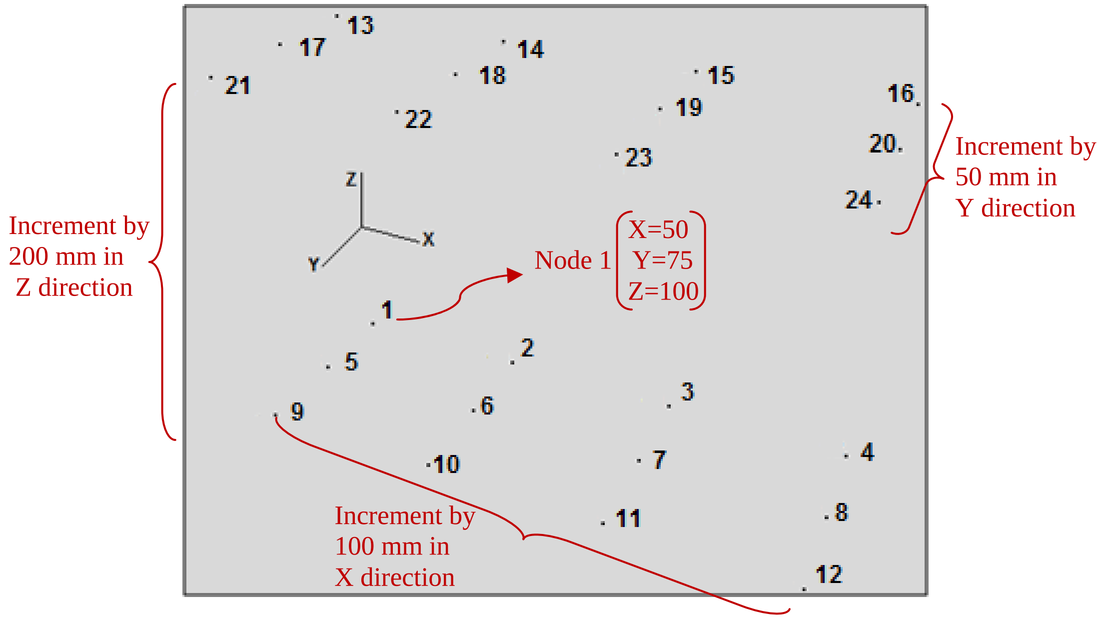


Figure 4-6 (B) – A simple example of node increment feature – 3D view

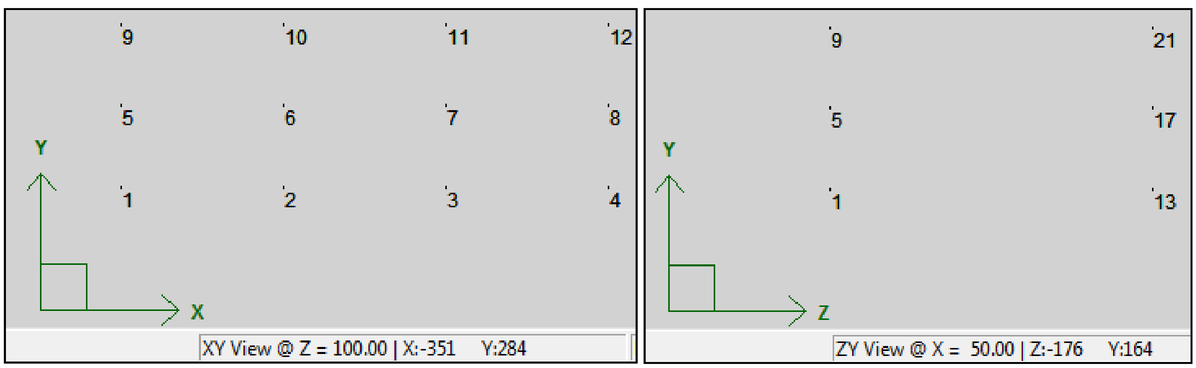
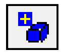


Figure 4-6(C) – A simple example of node increment feature – Sectional view

By adding or deleting the nodes, the total number of nodes and the drop list in the drop box, showing the list of the nodes, will be updated automatically. Note that by deleting the nodes, all the nodal loads assigned to them are also deleted.

4.3.3 Defining Elements

As explained before, VecTor3 has three types of elements: hexahedral elements, wedge elements, and truss bars. This section explains how to use the VecTor3 elements to create the finite element model in the FormWorks-Plus. Note that all the elements of a finite element mesh must be numbered consecutively starting from 1. FormWorks-Plus allows elements to be arbitrarily numbered but, when generating the structure file, it checks the element numbers and if the rule is violated FormWorks-Plus renumbers the elements automatically. Renumbering the elements is a useful feature which allows modifying an existing mesh instead of defining a new one.

- **Hexahedral Elements** 

The hexahedral element (or brick element) is an 8-node element with 24 degrees of freedom. Select the **Structure/Create Hexahedral Elements** menu item or click the **Create Hexahedral Elements** toolbar button. The **Create Hexahedral Elements** dialog box appears as shown in Figure 4-8 (C).

To generate an element, enter all 8 nodes and click the add button. The drop list in the drop box shows the list of all defined elements. To delete an element, select it from the drop list and click the delete button. The total number of elements and the drop list contents will be updated automatically by adding or deleting any element. The default colour for hexahedral elements is white.

FormWorks-Plus has two rules in defining hexahedral elements:

1. The first four nodes must have equal z coordinates and the value must be greater than the last four nodes z coordinates. This implies that, the first four nodes must be located at the front face and the last four nodes must be in the back face. (Z1=Z2=Z3=Z4 > Z5=Z6=Z7=Z8)

2. All the element nodes must be defined in a counter-clockwise order in the XY plane.

Figure 4-7 shows a sample hexahedral element defined in FormWorks-Plus:

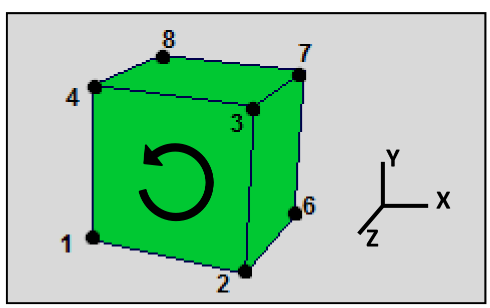


Figure 4-7 – A sample hexahedral element

Element's nodes must be entered manually in the dialog box. To save time, FormWorks-Plus has an element increment feature which allows defining several number of elements simultaneously in two directions. The following example shows how to use the increment feature for brick elements.

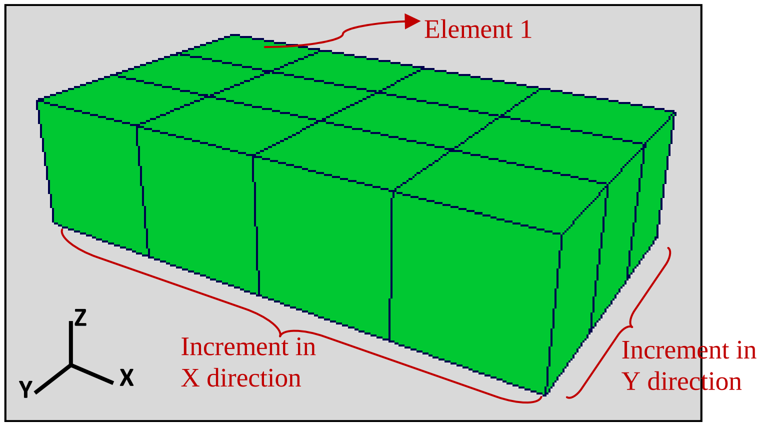


Figure 4-8 (A) – A simple example of creating brick elements using increment feature – 3D view

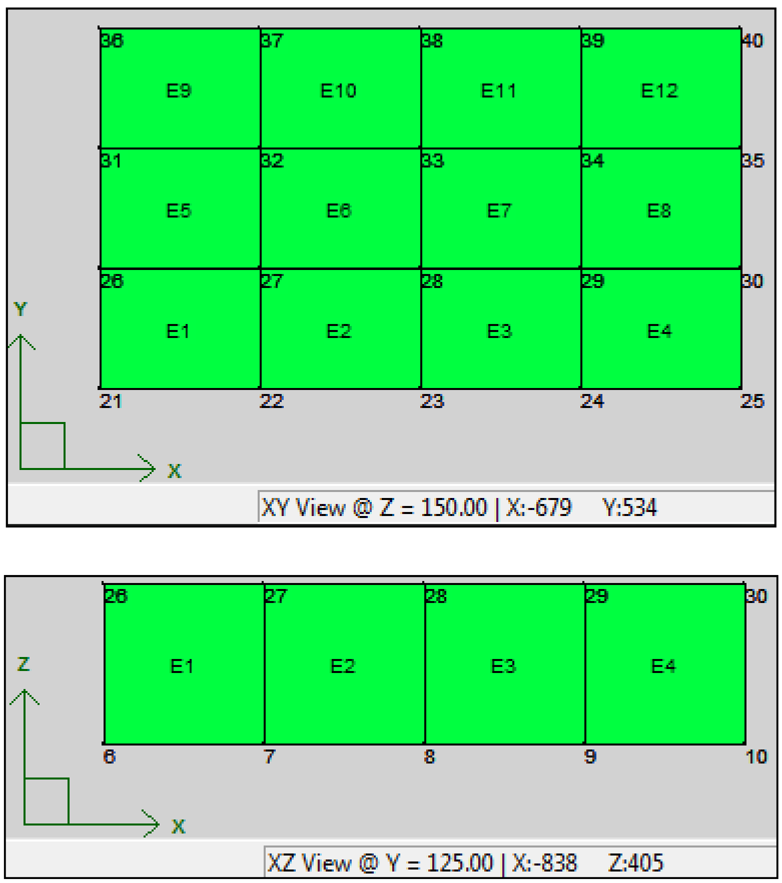


Figure 4-8 (B) – A simple example of creating brick elements using increment feature – Sectional view

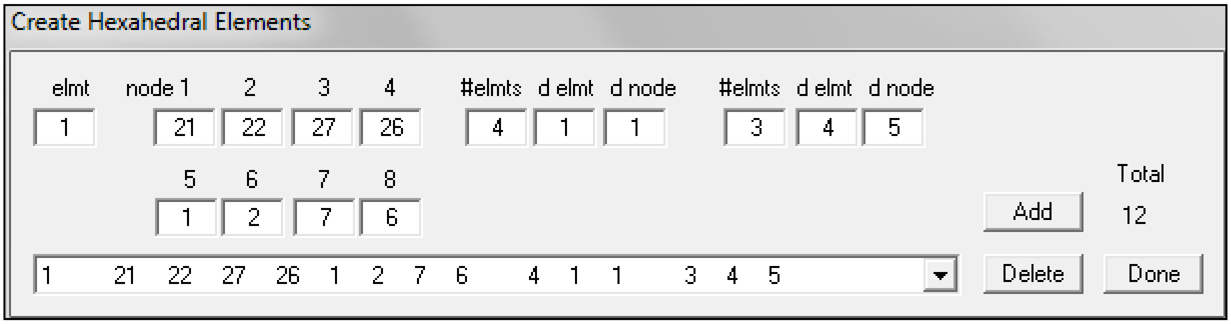


Figure 4-8 (C) – A simple example of creating brick elements using increment feature – Input data

- **Wedge Elements** 

The wedge element is a 6- node element with 18 degrees of freedom which assumes linear displacement fields similar to brick elements. Select the **Structure/Create Wedge Elements** menu item or click the **Create Wedge Elements** toolbar button. The **Create Wedge Elements** dialog box appears as shown in Figure 4-10 (A).

To generate a wedge element, enter all 6 nodes and click the add button. The drop list in the drop box shows the list of all the defined elements. To delete an element, select it from the drop list and click the delete button. The total number of elements and the drop list contents will be updated automatically by adding or deleting any element. The default colour for wedge element is white.

FormWorks-Plus allows defining three types of wedge elements. Table 4-1 describes the different types of wedge elements available and the rules to generate these elements. Note that, same as with hexahedral elements, in wedge elements all the nodes must be defined in a counter-clockwise order.

Wedge Element Types		
Type	**Rules**	
1	1. In XY plane, The first three nodes must be located at front face and last three nodes must be in back face.	(Z1=Z2=Z3) > (Z4=Z5=Z6)
	2. The first node in each face must have the maximum Y coordinate.	(Y1>Y2&Y3) , (Y4>Y5&Y6)
2	1. In XZ plane, The first three nodes must be located at front face and last three nodes must be in back face.	(Y1=Y2=Y3) > (Y4=Y5=Y6)
	2. The first node in each face must have the maximum Z coordinate.	(Z1>Z2&Z3) , (Z4>Z5&Z6)
3	1. In ZY plane, The first three nodes must be located at front face and last three nodes must be in back face.	(X1=X2=X3) > (X4=X5=X6)
	2. The first node in each face must have the maximum Z coordinate.	(Z1>Z2&Z3) , (Z4>Z5&Z6)

Table 4-1 –Different types of wedge element

As explained in Table 4-1, the triangular faces of elements must be parallel to one of the main planes. The current version of FormWorks-Plus is not capable of modeling an arbitrary shape of wedge element. Figure 4-9 illustrates the three types of wedge elements available in FormWorks-Plus.

An element's nodes must be entered manually in the dialog box. To save time, FormWorks-Plus has an element increment feature which allows defining several number of elements simultaneously in two directions. The following example shows how to use increment feature for wedge elements.

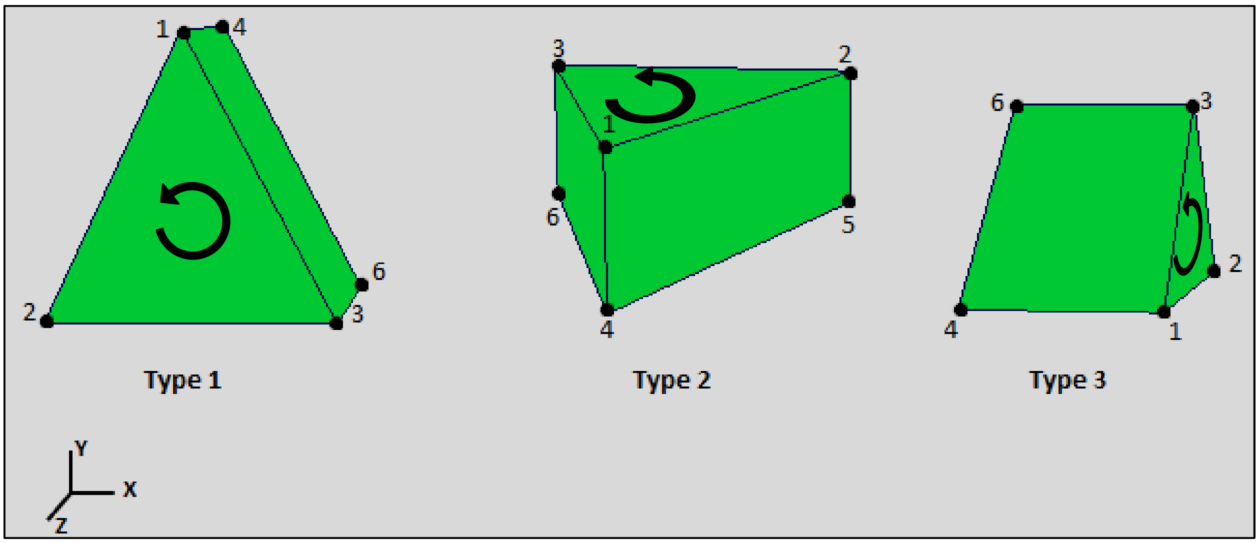


Figure 4-9 –Different types of wedge element

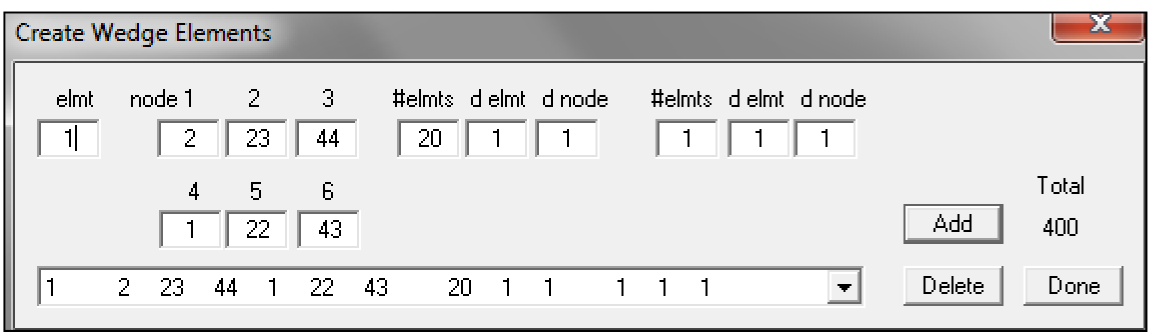


Figure 4-10 (A) – Create Wedge Elements window

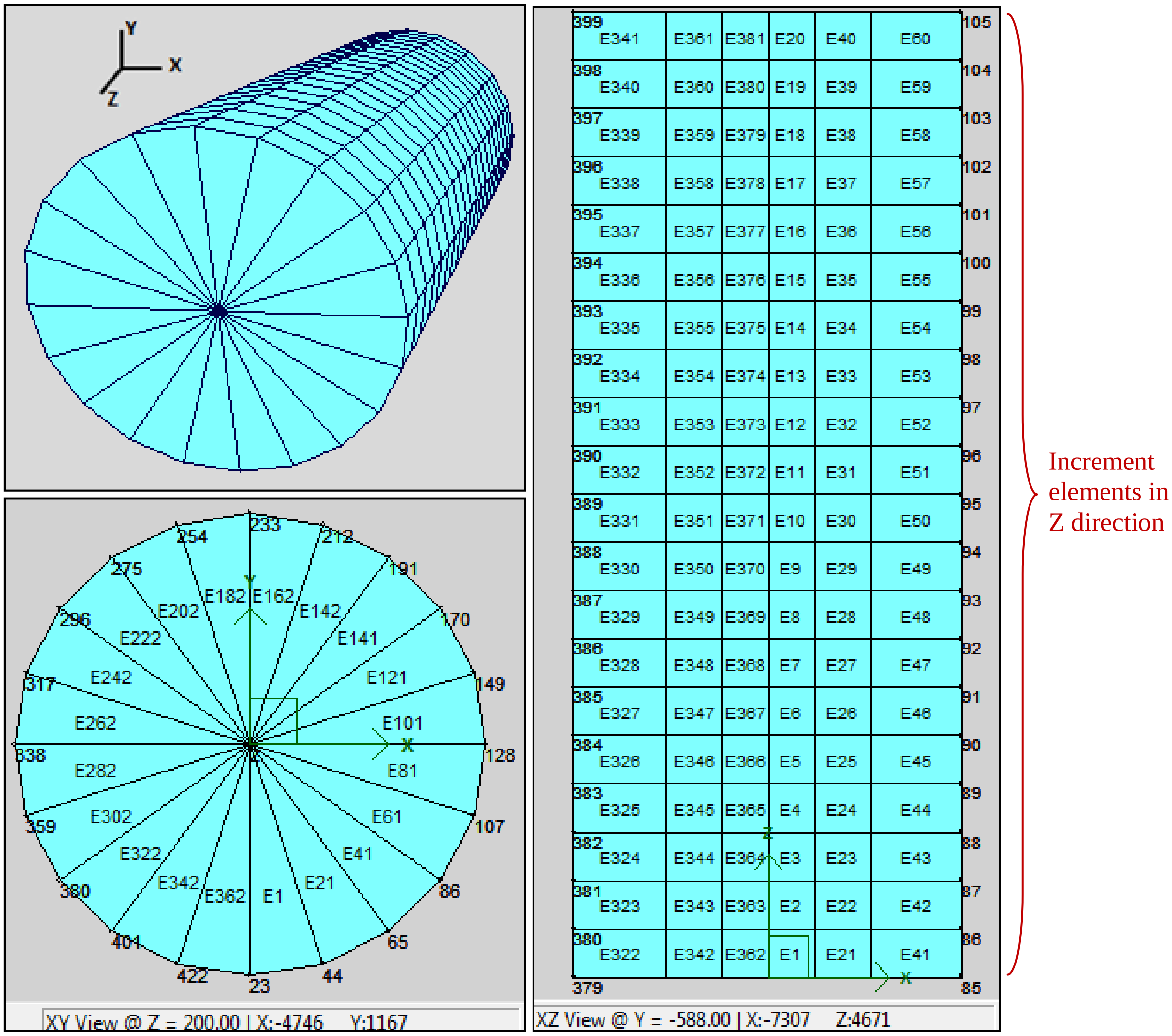
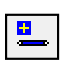


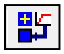
Figure 4-10 (B) – The Increment feature for wedge elements – Different views

- **Truss Elements** 

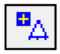
The third type of element in VecTor3 is the Truss Bar. Truss elements are mainly used to model reinforcement discretely. Select the **Structure/Create Truss Elements** menu item or click the **Create Truss Elements** toolbar button to define truss bars.

To generate a truss element enter two node numbers that define the end nodes of element and click the add button. The drop list in the drop box shows the list of all the defined elements. To delete an element, select it from the drop list and click the delete button. The total number of elements and the drop list contents will be updated automatically by adding or deleting any element. The default colour for truss elements is magenta.

As with hexahedral and wedge elements, the element increment feature can be used to define multiple truss bars in two directions simultaneously.

4.3.4 Assigning Materials 

The input properties required to assign material types in VecTor3 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, see the FormWorks manual.

4.3.5 Support Restraints 

Without restraining the structure, the finite element model is not complete and FormWorks-Plus will not allow generating the structure file. To add or delete restraints, select the **Structure/Create Support Restraints** menu item or click the **Create Support Restraints** toolbar button. The **Create Support Restraints** dialog box appears, as shown in Figure 4-11.

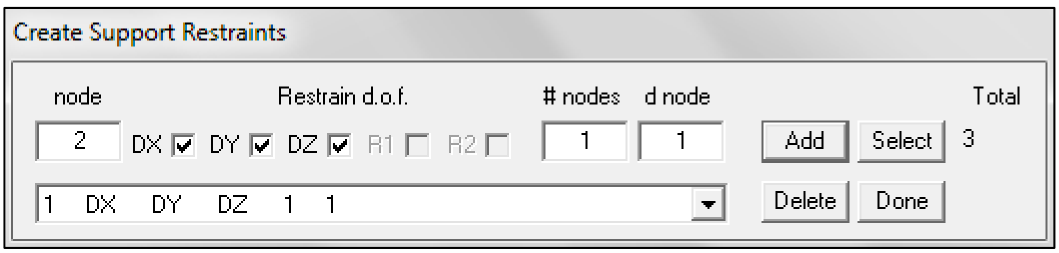


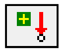
Figure 4-11 – Create Support Restraints window

For the most part, the input properties required to define the support restraints in VecTor3 are similar to the input requirements for VecTor2. For further instruction on how to select the nodes and enter these parameters, refer to the FormWorks manual. The only unique input parameter for 3D Solid Structures is **DZ** which restraints the selected nodes against displacement in the Z-direction.

4.4 The Load Case Data

The last step in defining the finite element model is assigning the loads. VecTor3 supports six types of nodal loads, three types of element loads, and ground acceleration load. This section will discuss all different types of loads and required input parameters.

4.4.1 Nodal Loads

4.4.1.1 Joint Loads 

Select the **Load/Apply Nodal Loads** menu item or click the **Nodal Loads** toolbar button. The **Apply Nodal Loads** dialog box appears as shown in Figure 4-12.

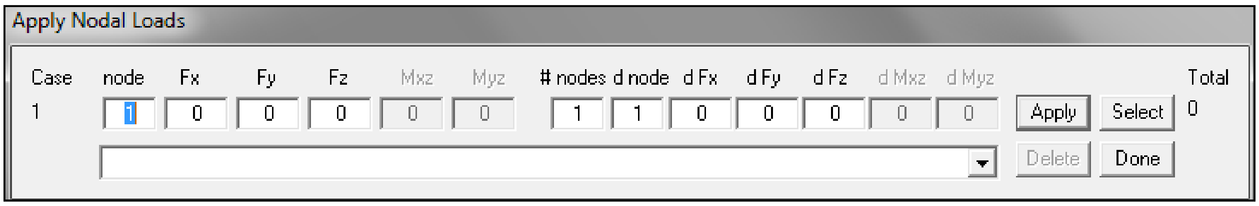


Figure 4-12 – Apply Nodal Loads window

The procedure for adding Joint Loads in VecTor3 is similar to that in VecTor2. For further instruction on how to select the nodes and enter the parameters, refer to the FormWorks manual. The unique entry fields are described as follows:

- **Fz:**
  Enter the joint force acting in Z-direction, in kN. Positive values represent the force in the direction of positive X-axis, while negative values are in the opposite direction.

- **dFz:**
  Enter the increment of the force in Z-direction for the selected nodes, in kN.

4.4.1.2 Support Displacements 

Select the **Load/Apply Support Displacement** menu item or click the **Support Displacement** toolbar button. The **Apply Support Displacement** dialog box appears as shown in Figure 4-13.

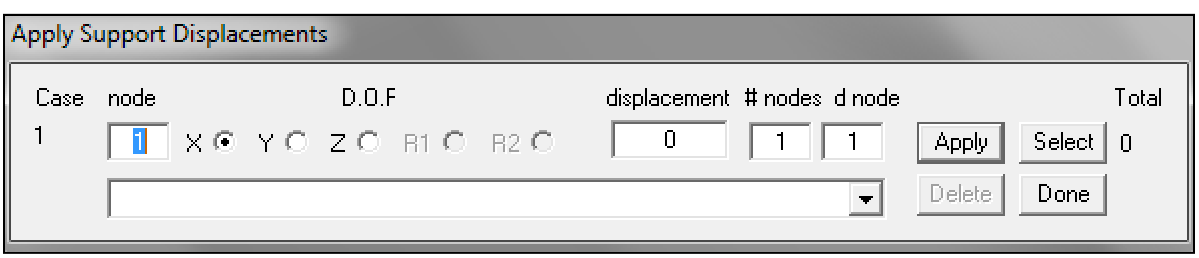


Figure 4-13 – Apply Support Displacements window

The procedure for adding Joint Loads in VecTor3 is similar to that in VecTor2. For further instruction on how to select the nodes and enter the parameters, refer to the FormWorks manual. The only unique entry field is degree of freedom (D.O.F) in the Z-direction. Select the X, Y or Z direction to apply nodal displacement to the selected nodes. To apply displacements in two or three directions, separate entries are required.

4.4.1.3 Vapour Content 

Select the **Load/Apply Nodal Vapour Content** menu item or click the **Nodal Vapour Content** toolbar button. The **Apply Nodal Vapour Content** dialog box appears as shown in Figure 4-14.

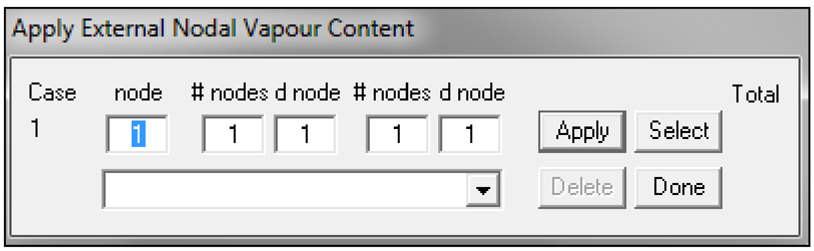


Figure 4-14 – Apply External Nodal Vapour window

Enter the node number or use the select button to specify the nodes with external vapour content. To delete a vapour content select the entry either from the drop list or using the select feature and click the delete button.

4.4.1.4 Nodal Thermal Loads 

Select the **Load/Apply Nodal Thermal Loads** menu item or click the **Nodal Thermal Loads** toolbar button. The **Apply Nodal Thermal Loads** dialog box appears as shown in Figure 4-15.

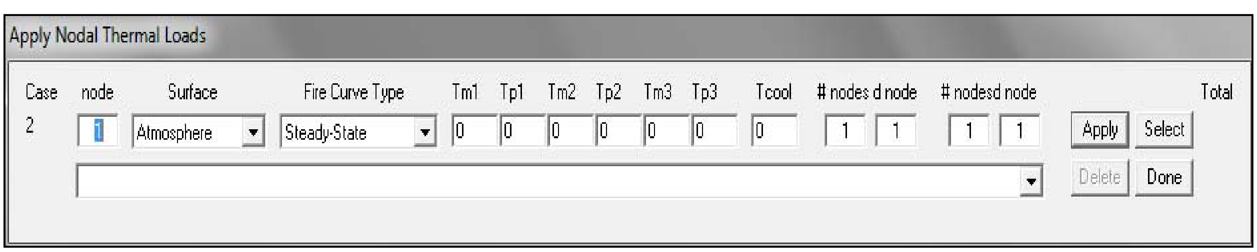


Figure 4-15 – Apply Nodal Thermal Loads window

Enter the node number or use the select feature to specify the nodes with thermal loads. To define a thermal load, complete the following input parameters.

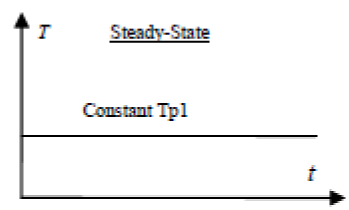
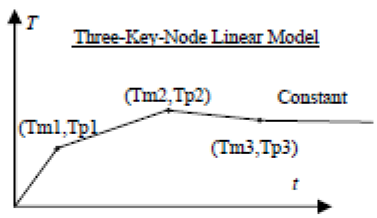
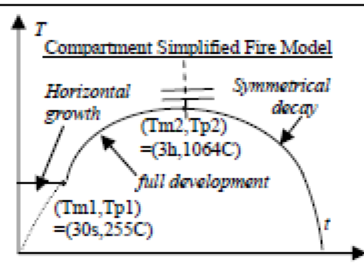
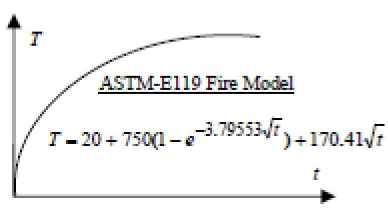
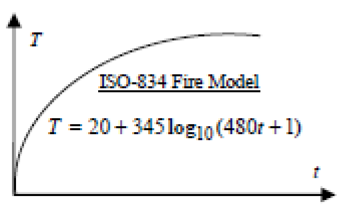
Load Type	Entry Fields in Load File								Load Plots
	Node	Type	Tm1	Tp1	Tm2	Tp2	Tm3	Tp3	
1	+	1		+					
2	+	2	+	+	+	+	+	+	
3	+	3	+	+	+	+			
4	+	4							
5	+	5							
Notes:	1. "+" indicates the field that is used in load definition; 2. Load types 2-5 are transient analyses while type 1 is steady-state one. 3. Full development curve in load type 3 is based on ASTM-E119 during the range of 255C-1064C; and the decay curve is symmetrical with development one about the peak point. 4. Standard fire curves in load types 4 and 5 have no ends.								

Table 4-2 – Different types of Standard Fire Curves (Zhou, 2004)

- **Surface:**

  Specify the surface for which the temperature parameters are going to be defined.

  1. Atmosphere
  2. Element Surface

  The default option is to apply the temperature parameters of the atmosphere.

- **Fire Curve Type:**

  Enter one of the following Standard Fire Curves:

  1. Steady- State
  2. Three-Key-Node Linear Model
  3. Compartment Simplified Fire Model
  4. ASTM-E119 Fire Model
  5. ISO-834 Fire Model

  The default model is number 1, Steady-State. For more information about the different types of Standard Fire Curves see Table 4-2.

- **Tm (n):**

  Enter the time, in seconds, at which the temperature **Tp (n)** occurs.

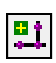
- **Tp (n):**

  Enter the temperature, in °C, corresponding to **Tm (n)**.

- **Tcool:**

  Enter the time, in seconds, at which the cooling process starts.

To delete a thermal load select the entry either from the drop list or using the select feature and click the delete button.

4.4.1.5 Lumped Masses 

Select the **Load/Apply Lumped Masses** menu item or click the **Lumped Masses** toolbar button. The **Apply Lumped Masses** dialog box appears as shown in Figure 4-16. Note that this load type is only applicable when performing dynamic analysis.

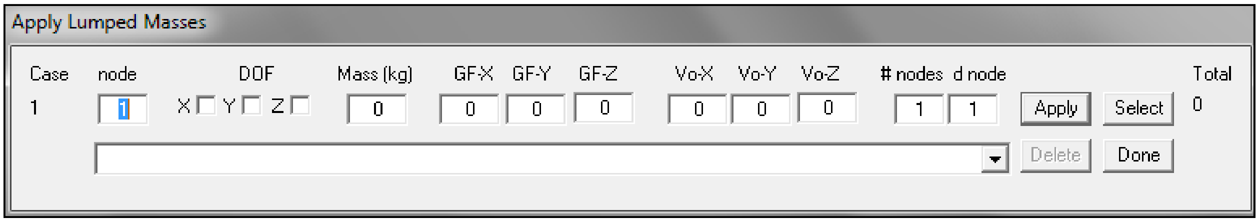


Figure 4-16 – Apply Lumped Masses window

Enter the node number or use the select feature to specify the nodes with lumped mass. To define a lumped mass load complete the following input parameters:

- **DOF:**

  Select the Degree Of Freedom (DOF) in which Lumped Mass is applied. It is possible to select more than one direction.

- **Mass:**

  Enter the lumped mass, in kg, acting on the selected nodes.

- **GF-X, Y or Z:**

  Enter the factor of ground acceleration acting in X, Y, and Z direction, expressed in g (9.81 m/s^2).

  Note: In VecTor4, the ground acceleration values are in m/s^2, not in terms of g. For instance to apply 4g ground acceleration in the X-direction, the input parameter in VecTor3 must be 4, while in VecTor4 it must be $4 \times 9.81 = 39.24\ m/s^2$.

- **Vo-X, Y or Z:**

  Enter the initial velocities, in m/s, acting on the selected nodes in X, Y, and Z direction.

4.4.1.6 Impulse Forces 

Select the **Load/Apply Impulse Forces** menu item or click the **Impulse Forces** toolbar button. The **Apply Impulse Forces** dialog box appears as shown in Figure 4-17.

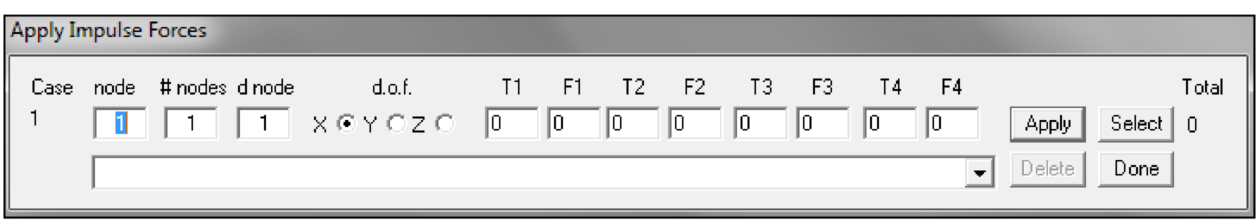
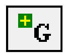


Figure 4-17– Apply Impulse Forces window

The procedure for adding the impulse forces in VecTor3 is similar to that in VecTor2. For further instruction on how to select the nodes and enter the parameters, refer to the FormWorks manual. The only unique entry field is the Degree Of Freedom (D.O.F.) in the Z-direction. Select one of the X, Y, or Z directions to apply impulse forces to the selected nodes. To apply impulse forces in two or three directions at the same node, separate entries are required.

4.4.2 Element Loads

4.4.2.1 Gravity Loads 

Select the **Load/Apply Gravity Loads** menu item or click the **Gravity Loads** toolbar button. The **Apply Gravity Loads** dialog box appears as shown in Figure 4-18.

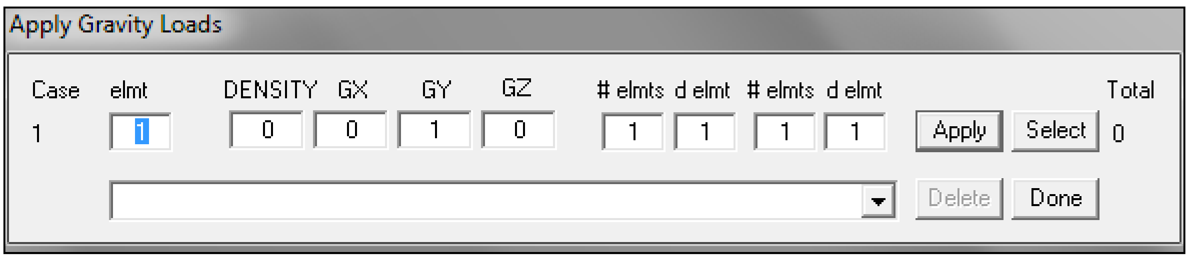


Figure 4-18 – Apply Gravity Loads window

The procedure for adding the gravity loads in VecTor3 is similar to that in VecTor2. For further instruction on how to select the nodes and enter the parameters, refer to the FormWorks manual. The only unique entry field is the G-force acting in the Z-direction, **GZ**. For gravity forces acting in the positive direction of Z-axis enter positive values. The GZ parameter can be used when the coordinate axes have been rotated and a component of the gravitational load is acting in the Z-direction.

4.4.2.2 Temperature Loads 

The input properties required to apply temperature loads in VecTor3 are similar to the input requirements for VecTor2 program. For further instruction on how to enter these parameters, refer to the FormWorks manual.

4.4.2.3 Concrete Prestrains 

The input properties required to apply concrete prestrains in VecTor3 are similar to the input requirements for VecTor2 program. For further instruction on how to enter these parameters, refer to the FormWorks manual.

4.4.3 Ground Accelerations 

Select the **Load/Apply Ground Acceleration** menu item or click the **Ground Acceleration** toolbar button. The **Apply Ground Acceleration** dialog box appears as shown in Figure 4-19.

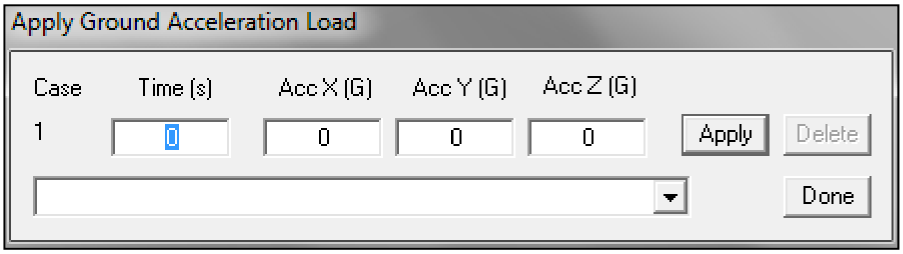


Figure 4-19 – Apply Ground Acceleration window

The procedure for adding the ground acceleration in VecTor3 is similar to that in VecTor2. For further instruction on how to select the nodes and enter the parameters, see the FormWorks manual. The only unique entry field is **Acc Z (G)**. Acc Z (G) is the ground acceleration in the Z direction, in m/s^2, occurring at the specified time.

4.5 A Simple Example in VecTor3

In this section, a simple example illustrating the FormWorks-Plus modeling procedure for 3D solid structures is presented. The objective of the example is to create a finite element model for a reinforced concrete beam with hollow section, and investigate the response of the structure under combined flexure and torsion. The beam to be modeled is one tested by Onsongo in 1978. Figure 4-20 and 4-21 demonstrate the testing arrangement and the cross sectional properties, respectively.

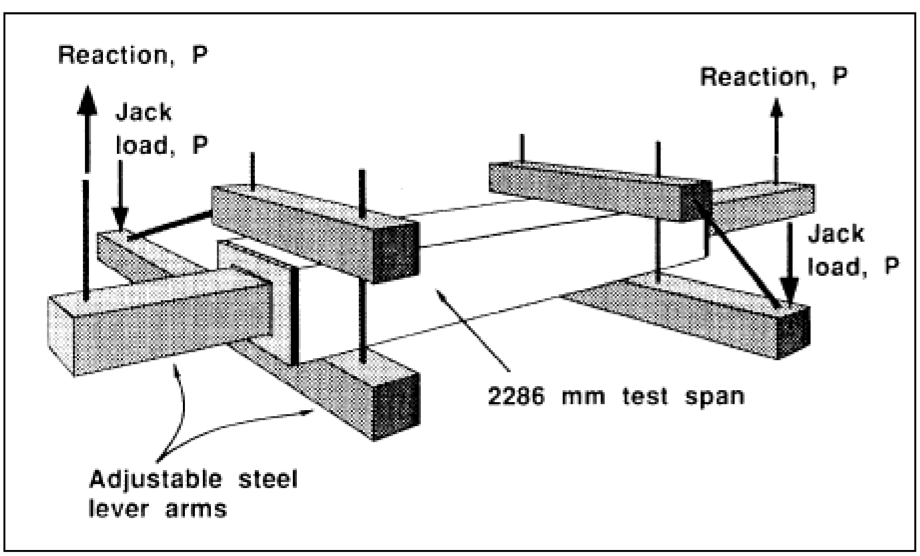


Figure 4-20 – Test arrangement for bending-torsion beam (Osongo, 1978).

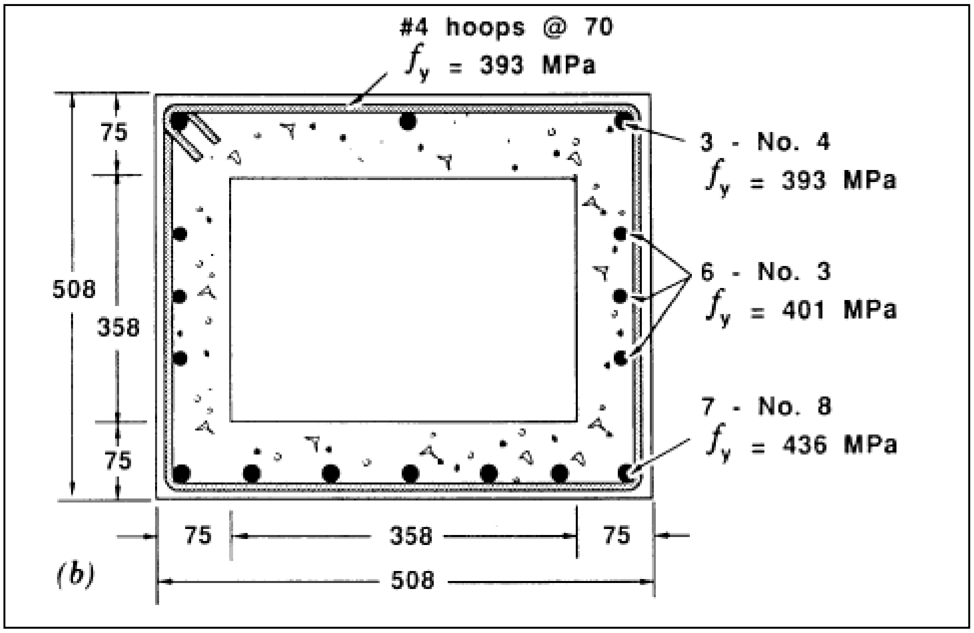


Figure 4-21 – The cross sectional properties

To save time in performing the analysis, only a part of the test beam considered in the finite element model. Since the beam is subjected to constant moment and torque along its length this simplification does not affect the results. Figure 4-22 shows the detailed meshing of the structure created by FormWorks-Plus. For modeling the transverse reinforcements properly, three elements were used through the thickness of the webs and flanges. To maintain the size of elements reasonable in all directions, 10 layers of elements were used along the length of the structure.

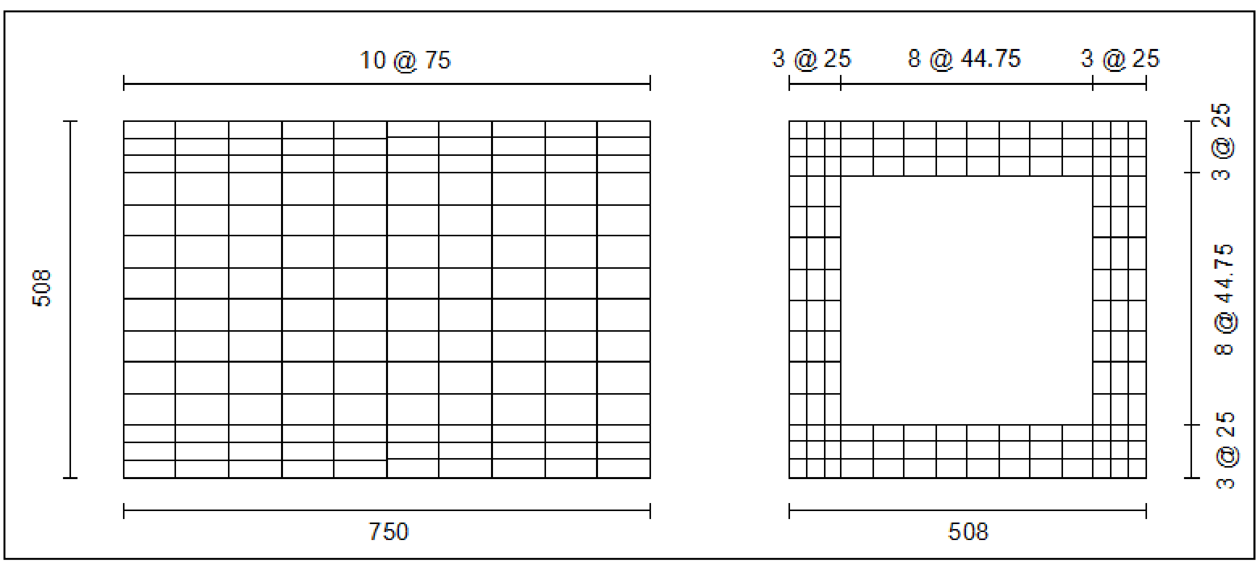


Figure 4-22 – The mesh used for the finite element model

Both the longitudinal and hoop reinforcement were modeled as smeared. To make the theoretical model more realistic, two-thirds of the hoop reinforcement was smeared in the outermost layer of elements. The remaining area of the reinforcement was smeared over the inner two layers of elements (Vecchio, et al., 1990). Table 4-3 and Figure 4-23(a) show the material type's properties and the location of each material type in the structural model, respectively.

Following figure demonstrates the finite element mesh, generated by FormWorks-Plus.

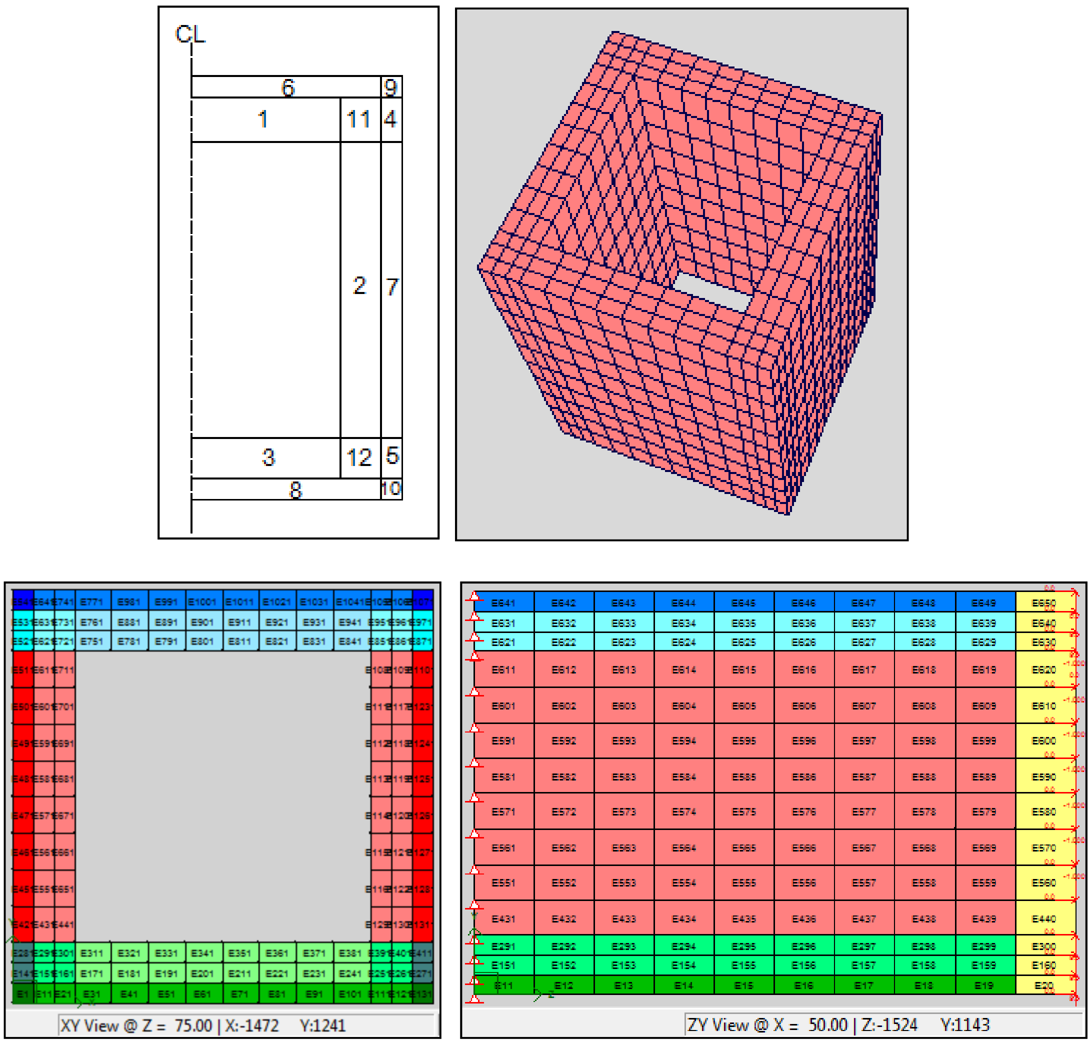


Figure 4-23 – The finite element mesh generated by FormWorks-Plus and the location of each material type

Material Type	Concrete Properties						Reinforcement Component Properties										
	Colour	F'c	F't	Ec	e0	Other Properties	Comp. #	Orientation			ρ	Db	Fy	Fu	Es	esh	eu
		MPa	MPa	MPa	me			k	l	m	%	mm	MPa	MPa	MPa	me	me
1		34.8	1.95	32445	3.1	default	1	1	0	0	1.229	12.7	393	600	200000	10	100
							2	0	0	1	1.016	12.7	393	600	200000	10	100
2		34.8	1.95	32445	3.1	default	1	0	1	0	1.229	12.7	393	600	200000	10	100
							2	0	0	1	0.793	9.525	401	600	200000	10	100
3		34.8	1.95	32445	3.1	default	1	1	0	0	1.229	12.7	393	600	200000	10	100
							2	0	0	1	9.352	25.4	436	600	200000	10	100
4		34.8	1.95	32445	3.1	default	1	0	1	0	4.914	12.7	393	600	200000	10	100
							2	0	0	1	1.016	12.7	393	600	200000	10	100
5		34.8	1.95	32445	3.1	default	1	0	1	0	4.914	12.7	393	600	200000	10	100
							2	0	0	1	9.352	25.4	436	600	200000	10	100
6		34.8	1.95	32445	3.1	default	1	1	0	0	4.914	12.7	393	600	200000	10	100
							2	0	0	1	1.016	12.7	393	600	200000	10	100
7		34.8	1.95	32445	3.1	default	1	0	1	0	4.914	12.7	393	600	200000	10	100
							2	0	0	1	0.793	9.525	401	600	200000	10	100
8		34.8	1.95	32445	3.1	default	1	1	0	0	4.914	12.7	393	600	200000	10	100
							2	0	0	1	9.352	25.4	436	600	200000	10	100
9		34.8	1.95	32445	3.1	default	1	1	0	0	4.914	12.7	393	600	200000	10	100
							2	0	1	0	4.914	12.7	393	600	200000	10	100
							3	0	0	1	1.016	12.7	393	600	200000	10	100
10		34.8	1.95	32445	3.1	default	1	1	0	0	4.914	12.7	393	600	200000	10	100
							2	0	1	0	4.914	12.7	393	600	200000	10	100
							3	0	0	1	9.352	25.4	436	600	200000	10	100
11		34.8	1.95	32445	3.1	default	1	1	0	0	1.229	12.7	393	600	200000	10	100
							2	0	1	0	1.229	12.7	393	600	200000	10	100
							3	0	0	1	1.016	12.7	393	600	200000	10	100
12		34.8	1.95	32445	3.1	default	1	1	0	0	1.229	12.7	393	600	200000	10	100
							2	0	1	0	1.229	12.7	393	600	200000	10	100
							3	0	0	1	9.352	25.4	436	600	200000	10	100

Table 4-3 – Material properties

The bending moments and torques were applied as nodal forces at the end of the beam. The ratio of the torsion to moment was 1.47. To avoid end failure and facilitate load transfer, a stiffer material was assigned to the elements in the final layer. Two load cases were defined in FormWorks-Plus, the Torsion load case and the Moment load case. The load factor was increased monotonically from zero to failure in increments of 1. Following is the VecTor3 analysis results:

$\mathbf{Mu = 182\ kNm\ \ \&\ \ Tu = 267\ kNm}$ **(**Note: T/M factor equals to 1.47)

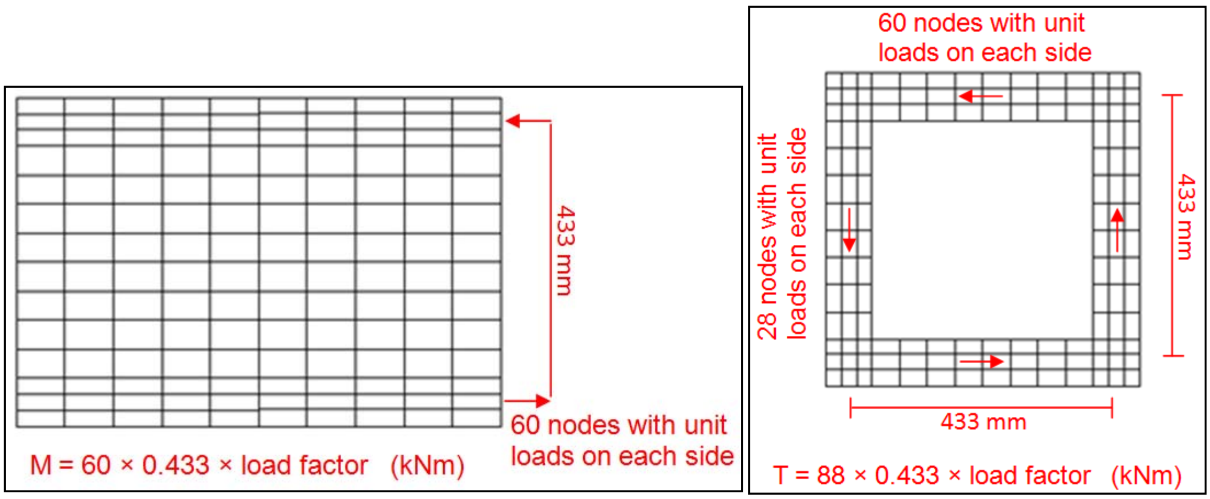


Figure 4-24 –Torsion and moment acting on the structural model

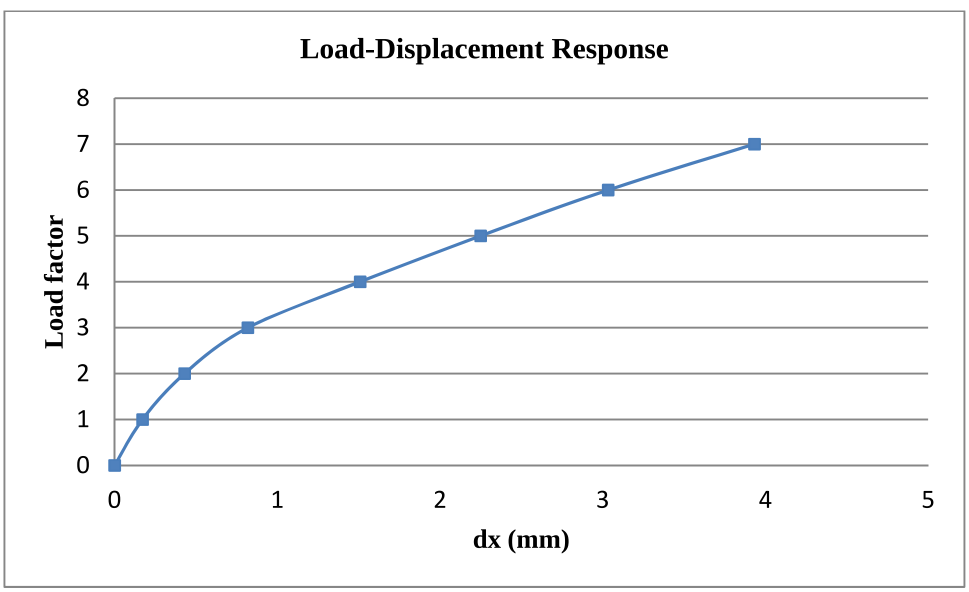


Figure 4-25 –Load-deflection response of the beam

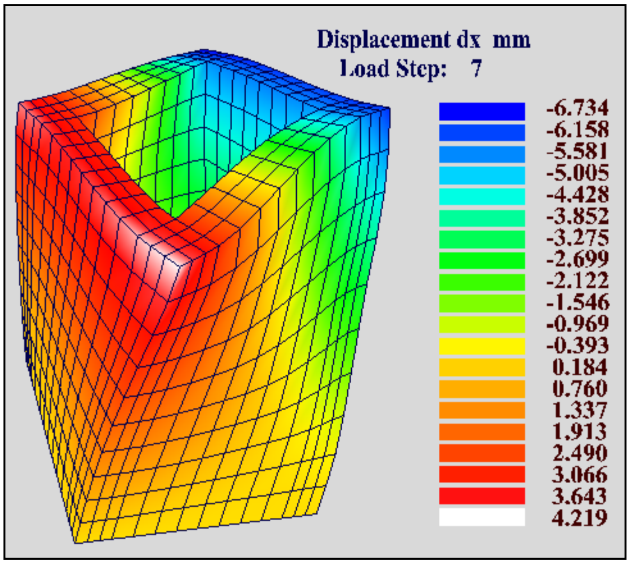


Figure 4-26 –Deformed shape of structure under combined torsion and moment loads

Chapter 5

Modeling Shell Structures

5.1 Introduction

VecTor4 is a nonlinear finite element analysis (NLFEA) program for reinforced concrete shell and plate structures. The theoretical basis for VecTor4 is the Disturbed Stress Field Model (Vecchio, 2000) which is derived from the Modified Compression Field Theory (Vecchio and Collins, 1986). The program uses a smeared rotating-crack formulation to analyze concrete structures. VecTor4 assumes a constant normal shear strain through the thickness of the elements to account for transverse shear deformations.

To generate a finite element model, VecTor4 utilizes degenerated quadratic isoparametric shell elements; specifically, a 9-node heterosis element with 42 degrees of freedom. Using quadratic shape functions allows the program to model curved surfaces. All side and corner nodes have three translational and two rotational degrees of freedom; the central ninth node has only the two rotational degrees of freedom. A principal advantage of the Heterosis element is that it exhibits good behaviour for both thick and thin shells (Seracino, 1995). Since it is one of the Mindlin type elements, its formulation is based on the following assumptions:

1. 'Normals' to the mid-surface remain straight but not necessarily normal after deformation, and
2. Resultant stresses normal to the mid-surface are negligible.

Figure 5-1 shows a typical 9-node heterosis element used in the program formulation.

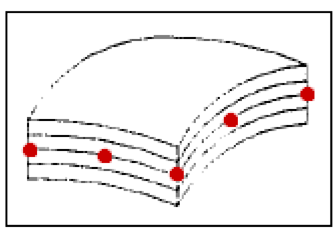
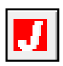


Figure 5-1 – A 9-node heterosis element

Incorporated into the program's analysis algorithms are material nonlinearity effects such as compression softening due to transverse cracking, tension stiffening, shear slip along crack surfaces, and other mechanisms important in accurately representing the behaviour of cracked reinforced concrete structures. To model material properties, VecTor4 employs a layered element formulation. Each material type can have up to 50 layers of concrete, where each layer has the same thickness and concrete properties. Steel layers are superimposed to model the in-plane reinforcement. Out-of-plane (transverse shear) reinforcement is defined as a property of the concrete layers.

The program is currently configured to accommodate: 600 elements, 3000 nodes. 35 concrete/steel material types, 50 concrete layers per material type, and 10 smeared reinforcement components per material type.

VecTor4 does not have pre-processor; it runs in a DOS environment from Fortran executable files. Creating finite element models without any graphical tool, especially with the complex heterosis element, is time consuming and can cause meshing problems. A user-friendly pre-processor, FormWorks-Plus, has been developed to create finite element models compatible with the analysis program. FormWorks-Plus is able to create shell element models, apply different types of loads, assign material properties, and specify analysis parameters that are compatible with VecTor4. This chapter will discuss the different parts of FormWorks-Plus that are involved in modeling a shell structure to be analyzed by VecTor4.

5.2 The Job Data 

For the most part, the input properties required to define the Job Page for the Shell Structures are similar to the input requirements for the 3D Solid Structures. For further instruction on how to enter these parameters, see Section 2 in Chapter 4. The Auxiliary page is the only part that has input parameters unique to Shell Structures:

- **Shear Analysis Mode**

  Select one of the following models for the shear analysis mode:

  1. Uniform Shear Strain
  2. Parabolic Shear Strain
  3. No Out-of-Plane Shear

  The default model is option 2, Parabolic Shear Strain.

- **Mass Factor due to Self-Weight**

  In dynamic analysis, to consider the nodal lumped masses due to the self-weight of the structure, a nonzero factor must be applied. Note that self-masses are only used in dynamic analyses; they are not converted to static forces. The default value is 1.0 which corresponds to the unfactored self-mass of the structure.

Figure 5-1 shows unique analysis parameters for VecTor4 in the Auxiliary Page.

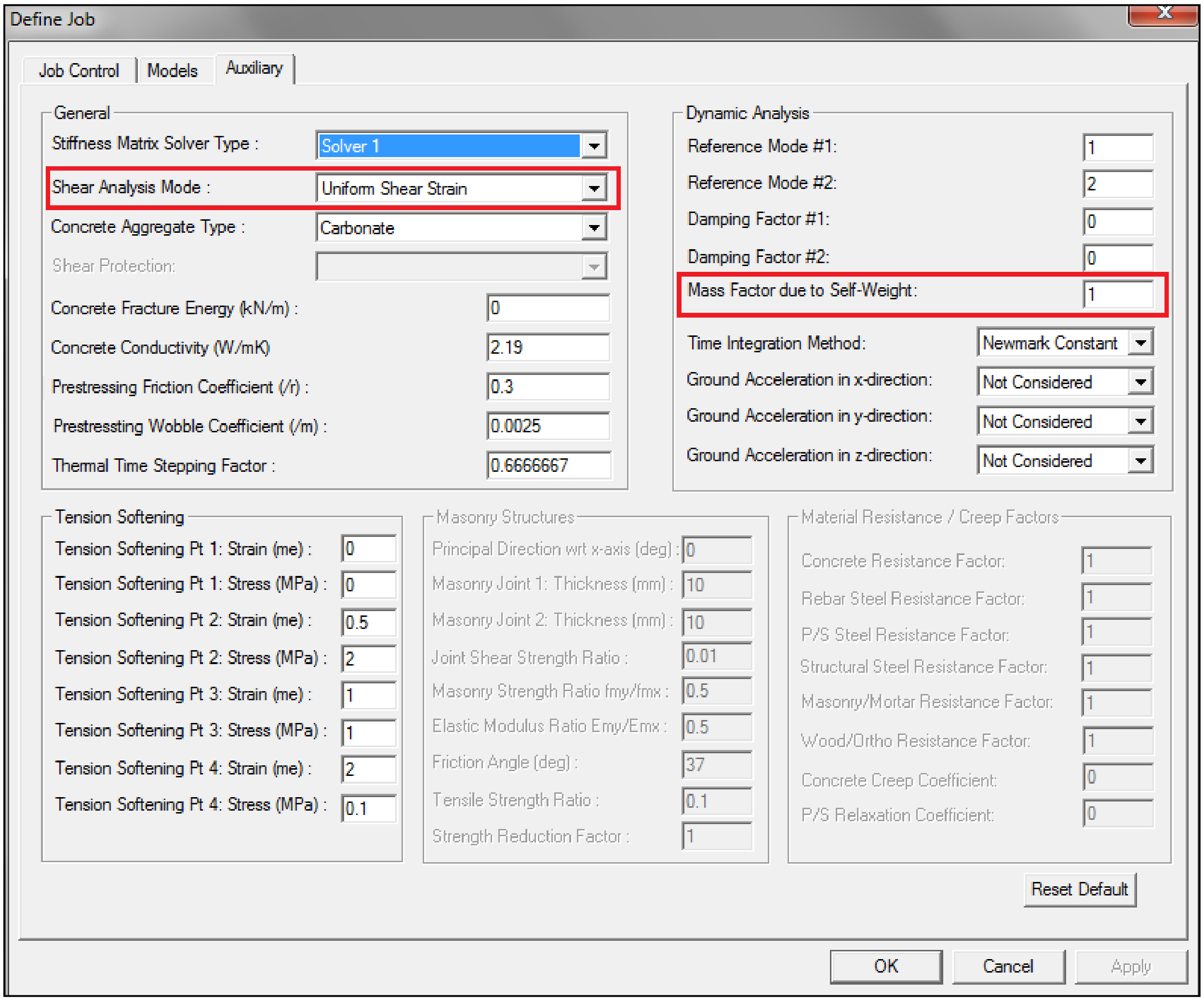


Figure 5-2 – The Auxiliary Page for Shell Structures

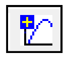
Note that although all the input parameters in the Models Page are similar to the inputs for the previous types of structures discussed in Chapter 3 and 4, each drop list contains models which are only applicable to Shell Structures.

5.3 The Structure Data

After defining all analysis parameters and behavioural models in the Job Data Page, the next step is to define the Structure Data which creates a structure file with the *.S4R extension. This file contains information about the finite element mesh including material properties, node numbers and coordinates, elements, and restraints. Note that all the steps explained in Chapter 4 for creating a proper finite element mesh must be considered in modeling shell structures, as well.

5.3.1 Specifying Material Properties

VecTor4 includes two types of materials: reinforced concrete and reinforcement.

5.3.1.1 Reinforced Concrete 

The input properties required to define the reinforced concrete material in VecTor4 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual. The input parameters unique to VecTor4 are:

- **Centerline Offset, OS:**
  Enter the centerline offset of the element, in mm, in the direction of the local Z-axis (The out-of-plane axis of the element). Positive values represent an offset in the positive local Z-direction, while negative values represent an offset in the opposite direction. The default value is 0.

  Note: In VecTor4, the local coordinate system (x y z) is a Cartesian System which is defined for each element. The z-direction is oriented perpendicular to the mid-plane of the element. The x-axis is parallel to the global X-Z plane. However, if z-axis is in the direction of the global Y-axis, then x-axis is oriented in the direction of global X-axis. Figure 5-3 demonstrates the local coordinate system definition (Seracino, 1995).

- **Number of Concrete Layers:**
  Enter the number of concrete layers for each material type. The default value is 10.

- **Distance from Top, d:**
  Enter the distance from the top of the element to the centroid of the reinforcement layer, in mm. The default value is 0 mm.

Note that out-of-plane (transverse shear) reinforcement is defined as a property of the concrete layers. The input parameters are same as Reinforcement Component Properties.

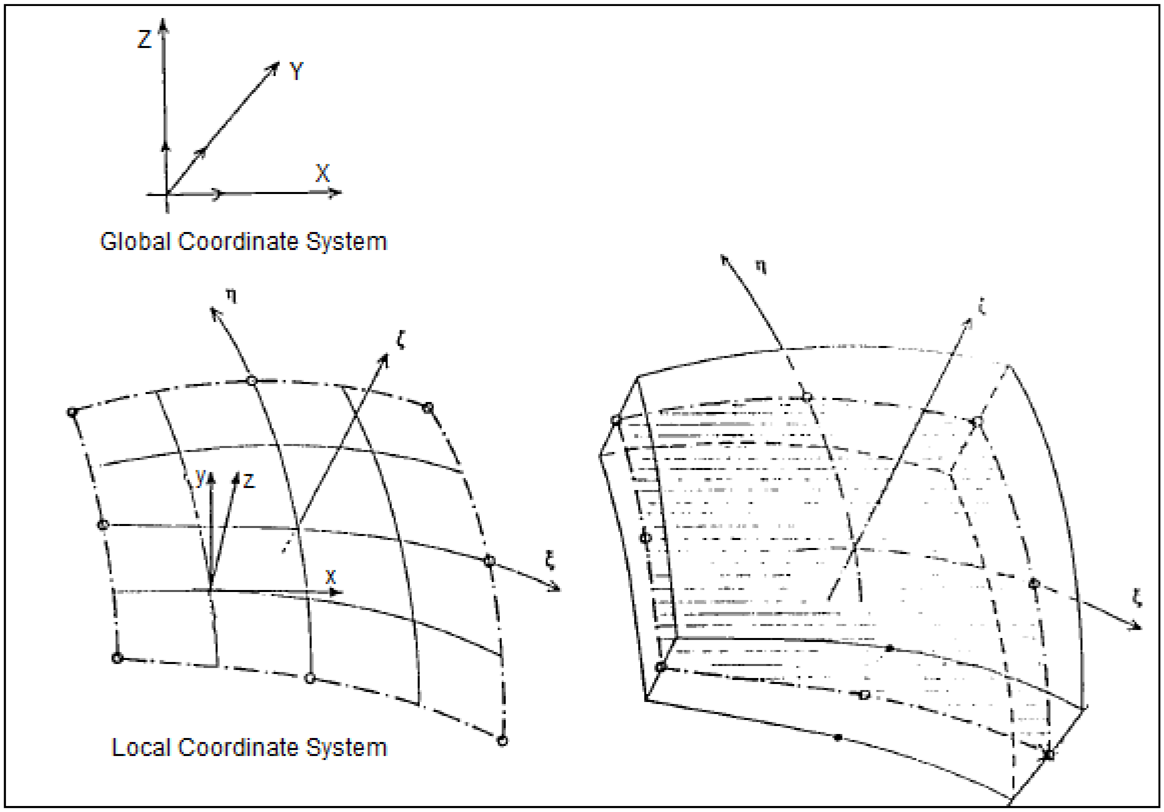


Figure 5-3 – Local coordinates in VecTor4 (Seracino, 1995).

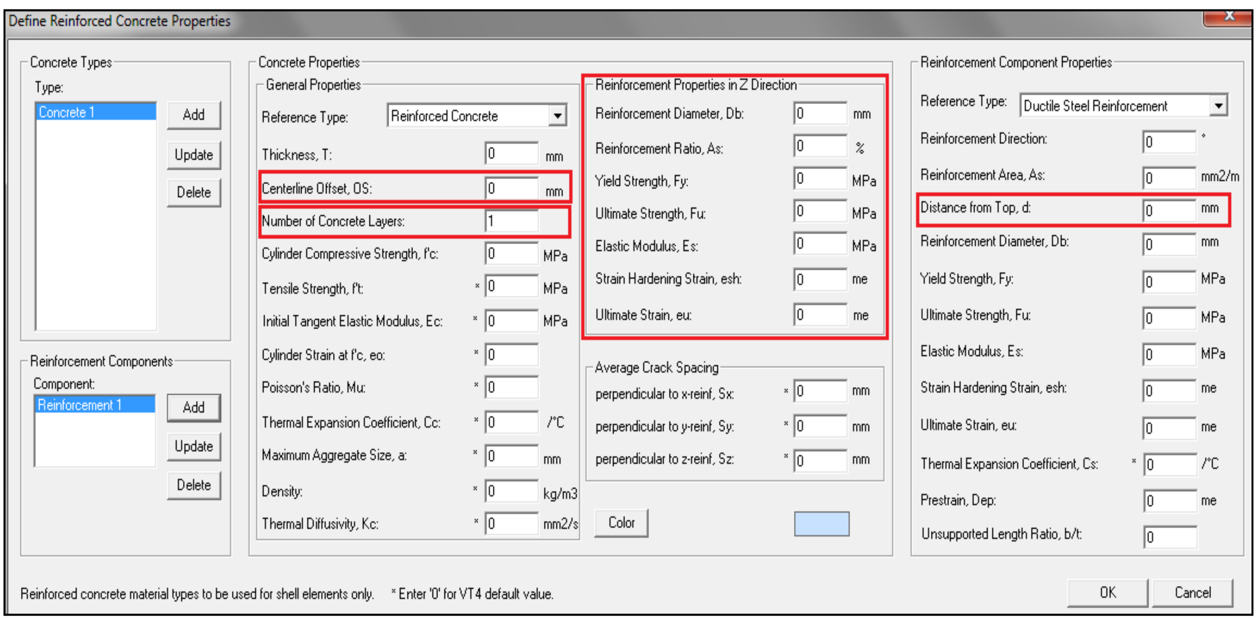
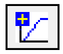


Figure 5-4 – Define Reinforced Concrete Properties window for Shell Structures

5.3.1.2 Reinforcement 

For the most part, the input properties required to define the reinforcement material in VecTor4 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual. The only unique input parameter to VecTor4 is **Centerline Offset (OS)** which has been explained in Section 3.1.1.

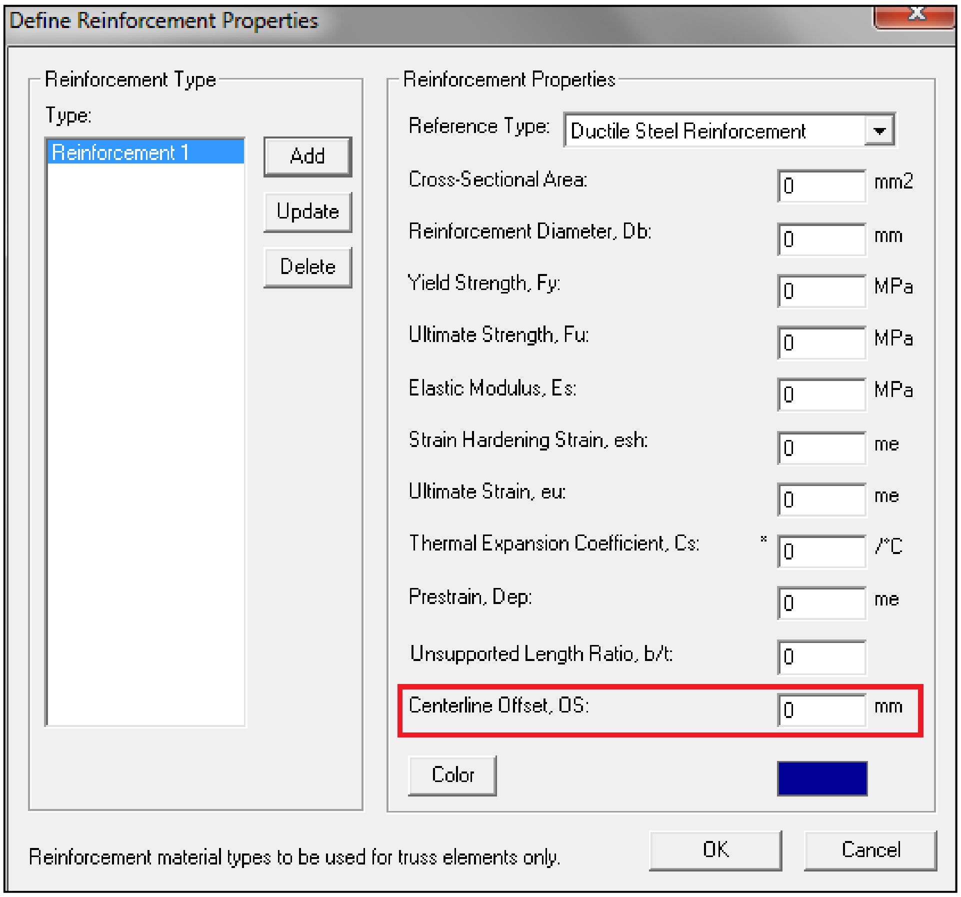
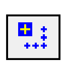


Figure 5-5 – Define Reinforcement Properties window for Shell Structures

5.3.2 Defining Nodes 

Select the **Structure/Create Nodes** menu item or click the **Create Node**s toolbar button. The **Create Nodes** dialog box appears as shown in Figure 5-6. This section explains the procedures for adding or deleting nodes. Note that the rules for defining the nodes, discussed in Section 3.2 of Chapter 4, must be considered for Shell Structures as well. FormWorks-Plus checks the rules to make sure all nodes are defined correctly.

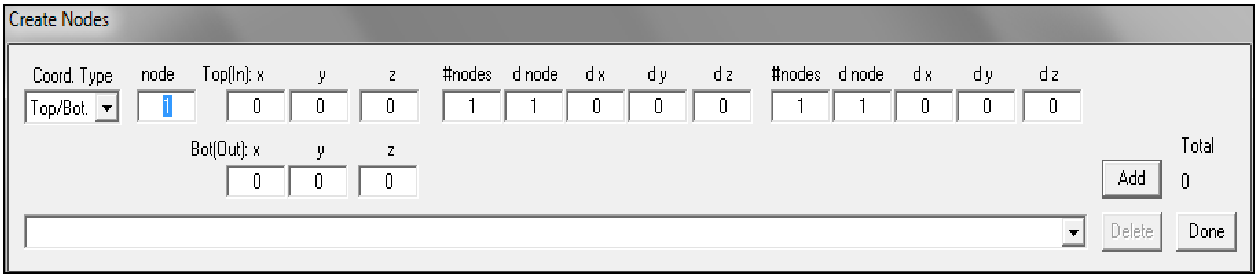


Figure 5-6 – Create Nodes window for Shell Structures

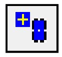
As discussed in the first chapter, FormWorks-Plus has two options for specifying node coordinates: Top (In) and Bottom (Out) or Centerline node coordinates. The default option is Top/Bottom coordinate type. To display the bottom coordinates of the nodes on the screen, check the Node Numbers (Out/Bot.) box in the Display Options window. By selecting Centerline coordinate type from the drop list, the input parameters for coordinates will reduce from 6 to 3 automatically. The Top (In) and Bottom (Out) coordinates option is mainly used for modeling curved structures; Centerline coordinates are suitable for modeling slabs and beams.

For Shell Structures, the coordinates of each node must be defined manually in the Create Nodes window. Since creating nodes one by one is a time consuming process, as with other structural types, FormWorks-Plus allows the user to define the nodes by a node increment feature.

By adding or deleting the nodes, the total number of nodes and the drop list in the drop box, showing the list of the nodes, will be updated automatically. Note that by deleting nodes, all the nodal loads assigned to them are also deleted.

5.3.3 Defining Elements

VecTor4 has two types of elements: heterosis shell elements and truss bars. This section explains how to use these elements to create a finite element model in FormWorks-Plus. Note that all the elements of a finite element mesh must be numbered consecutively starting from 1. As with other structural types, FormWorks-Plus allows heterosis elements to be arbitrarily numbered. However, when generating the structure file, FormWorks-Plus checks the element numbers and if the rule is violated, it renumbers the elements automatically.

- **Heterosis Elements** 

The heterosis element is a 9-node element with 42 degrees of freedom. Select the **Structure/Create Heterosis Elements** menu item or click the **Create Heterosis Elements** toolbar button. The **Create Heterosis Elements** dialog box appears as shown in Figure 5-8.

To generate an element, enter all 9 nodes and click the add button. The drop list in the drop box shows the list of all defined elements. To delete an element, select it from the drop list and click the delete button. The total number of elements and the drop list contents will be updated automatically by adding or deleting any element. The default colour for hexahedral elements is white.

All the element nodes must be defined in a counter-clockwise order. Figure 5-7 shows a sample heterosis element defined in FormWorks-Plus:

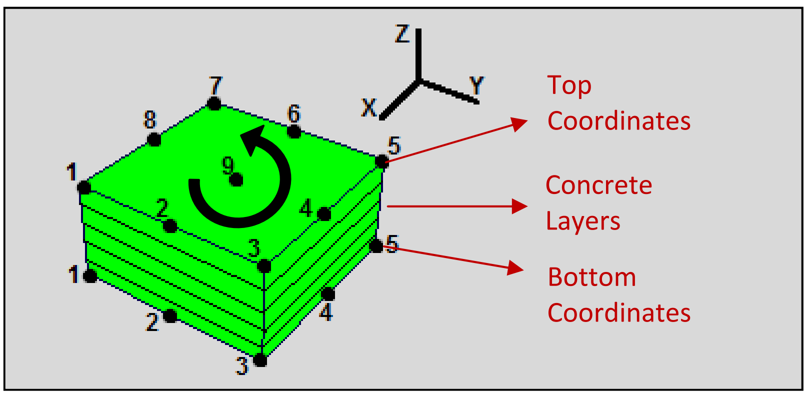


Figure 5-7 – A sample heterosis element

An element's nodes must be entered manually in the dialog box. As with the other structural types, FormWorks-Plus has an element increment feature which allows defining several elements simultaneously in two directions.

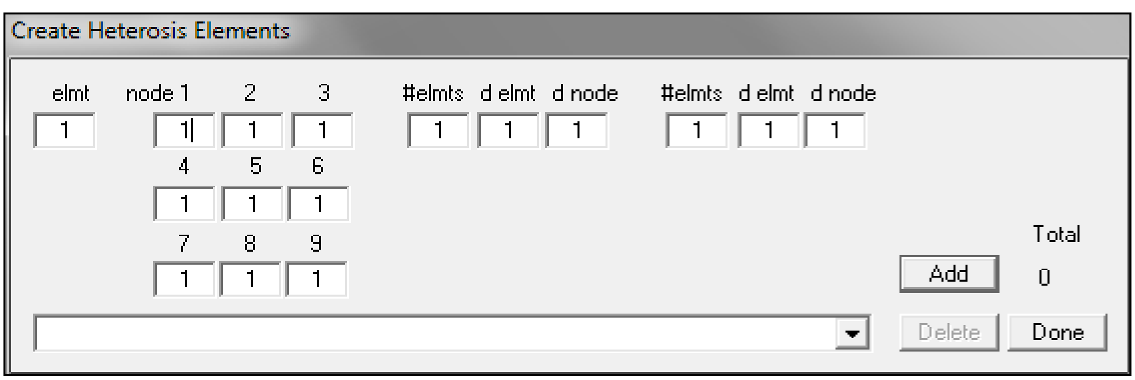
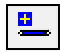
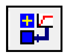


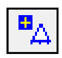
Figure 5-8– A Create Heterosis Elements window

- **Truss Elements** 

The second type of element in VecTor4 is the Truss Bar. Truss elements are mainly used to model reinforcement discretely. They can also be used to model external “springs”. Select the **Structure/Create Truss Elements** menu item or click the **Create Truss Elements** toolbar button to define truss bars.

5.3.4 Assigning Materials 

The input properties required to assign material types in VecTor4 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual.

5.3.5 Support Restraints 

Without restraining the structure, the finite element model is not complete and FormWorks-Plus will not allow generating the structure file. To add or delete restraints, select the **Structure/Create Support Restraints** menu item or click the **Create Support Restraints** toolbar button. The **Create Support Restraints** dialog box appears as shown in Figure 4-11.

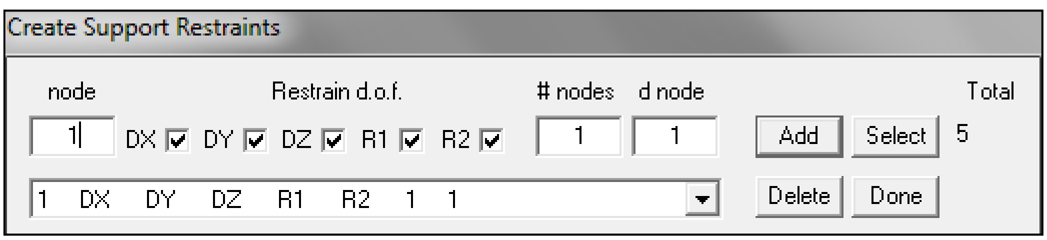


Figure 5-9 – Create Support Restraints window

For the most part, the input properties required to define the support restraints in VecTor4 are similar to the input requirements for VecTor3. The unique input parameters for Shell Structures are rotational restraints. Check the **R1** and **R2** boxes to restraint the selected nodes against rotations about the local v_{2k} and v_{1k} directions.

Note: In VecTor4, $(v_{1k}\ v_{2k}\ v_{3k})$ is the nodal coordinate system defined at each nodal point. Its origin lies on the midsurface of the element, where the subscript k indicates node k. Vector v_{3k} is defined by the line joining the bottom coordinates of the node k to the top coordinates of node k. Vector v_{1k} is oriented perpendicular to v_{3k} and parallel to the XZ global plane. However, if v_{3k} is oriented in the global Y-direction, then v_{1k} is oriented in the global X-direction. Finally, vector v_{2k} is oriented perpendicular to the plane formed by vector v_{3k} and v_{1k}. Figure 5-10 demonstrates the nodal coordinate system defined in VecTor4.

R1 and R2 are the rotations about the local v_{2k} and v_{1k} directions, respectively. By calculating the components of rotations, R1 and R2 can be expressed for each element in terms of local coordinate system, where R1 is the rotation about local y-direction and R2 is the rotation about local x-direction. Refer to Figure 5-3 for the local coordinate system definition.

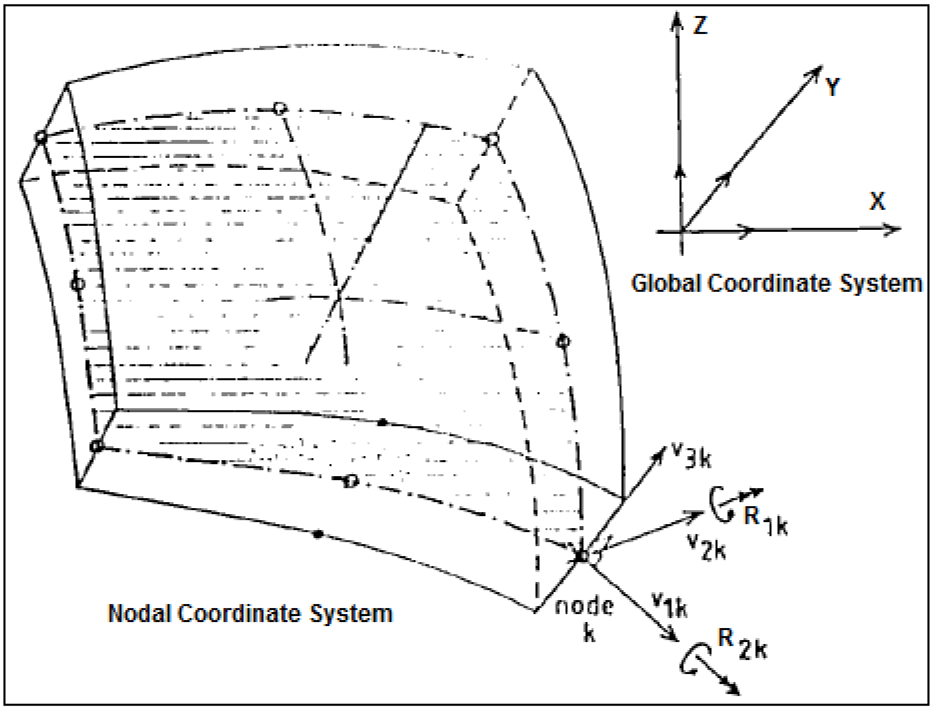
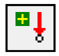


Figure 5-10 – Nodal coordinate system in VecTor4 (Seracino, 1995).

5.4 The Load Case Data

The last step in defining the finite element model is assigning the loads. VecTor4 supports four types of nodal loads, five types of element loads, and ground acceleration load. This section will discuss the different types of loads and required input parameters. Note that the load types, including Ground Acceleration, Impulse Forces, Lumped Masses, and Concrete Prestrains, and the input parameters that are similar to VecTor3 are not described in this section. For instruction on how to define these load types and input parameters, see Section 4 in Chapter 4.

5.4.1 Nodal Loads

5.4.1.1 Concentrated Loads 

Select the **Load/Apply Concentrated Loads** menu item or click the **Concentrated Loads** toolbar button. The **Apply Concentrated Loads** dialog box appears as shown in Figure 5-11.

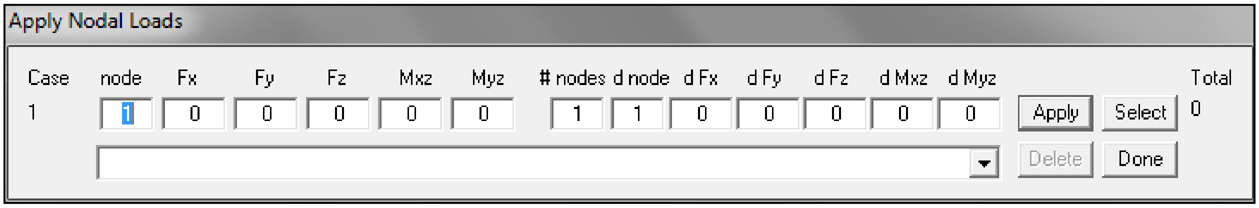


Figure 5-11 – Apply Nodal Loads window

The unique input parameters to Shell Structures are:

- **Mxz:**

  Enter the magnitude of the concentrated bending moment about v_{2k} direction, in kNm. It implies a bending moment about the local y-direction.

- **Myz:**

  Enter the magnitude of the concentrated bending moment about v_{1k} direction, in kNm. It implies a bending moment about the local x-direction.

5.4.1.2 Prescribed Node Displacements 

Select the **Load/Apply Concentrated Loads** menu item or click the **Concentrated Loads** toolbar button. The **Apply Concentrated Loads** dialog box appears as shown in Figure 5-12.

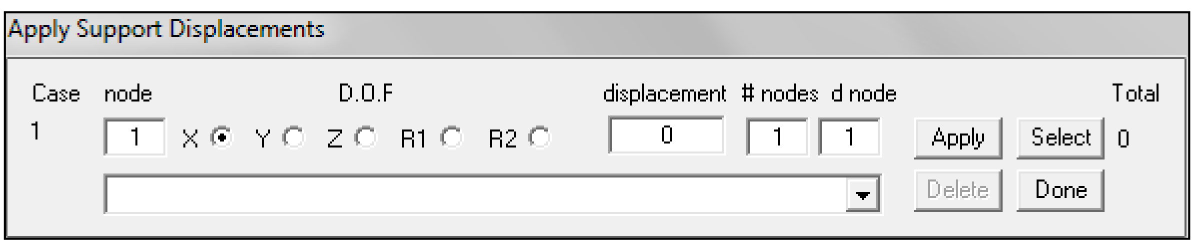


Figure 5-12 – Apply Nodal Loads window

The unique input parameters to Shell Structures are **R1** and **R2** which are described in Section 3.5.

5.4.2 Element Loads

5.4.2.1 Uniformly Distributed Loads 

Select the **Load/Apply Uniformly Distributed Loads** menu item or click the **Uniformly Distributed Loads** toolbar button. The **Apply Uniformly Distributed Loads** dialog box appears as shown in Figure 5-13. Enter the element number or use the select feature to specify the elements with uniformly distributed loads. To delete a load, select the entry either from the drop list or using the select feature and click the delete button.

The **pressure** input parameter is the magnitude of the uniformly distributed load applied normal to the elements surface, in MPa, in the direction of local z-axis. A negative value represents a pressure acting from the top to the bottom surface of the element, while a positive value is a pressure acting in the opposite direction.

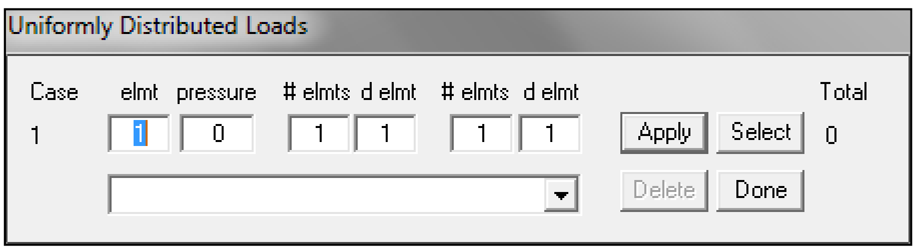


Figure 5-13 – Apply Uniformly Distributed Loads window

5.4.2.2 Hydrostatic Loads 

Select the **Load/Apply Hydrostatic Loads** menu item or click the **Hydrostatic Loads** toolbar button. The **Apply Hydrostatic Loads** dialog box appears as shown in Figure 5-14.

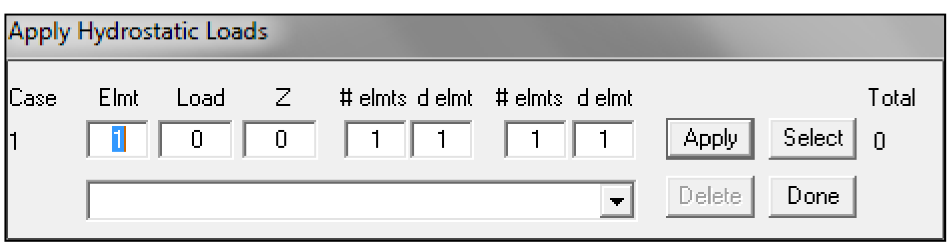
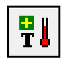


Figure 5-14 – Apply Uniformly Distributed Loads window

Enter the following input parameters for defining the load:

- **Load:**
- Enter the maximum value of the hydrostatic load, in MPa, in the direction normal to the surface of the structure.
- **Z:**

  Enter the global coordinate of zero hydrostatic pressure, in mm, in the Z-direction.

5.4.2.3 Temperature Loads 

Select the **Load/Apply Temperature Loads** menu item or click the **Temperature Loads** toolbar button. The **Apply Temperature Loads** dialog box appears as shown in Figure 5-15.

The unique input parameters to VecTor4 are:

- **Final Bottom:**

  Enter the final temperature change of the bottom surface of the element, in °C.
- **Initial Bottom:**

  Enter the initial temperature change of the bottom surface of the element, in °C.
- **Final Top:**

  Enter the final temperature change of the top surface of the element, in °C.
- **Initial Top:**

  Enter the initial temperature change of the top surface of the element, in °C.
- **Elapsed:**

  Enter the elapsed time between the initial and final temperatures, in hours.

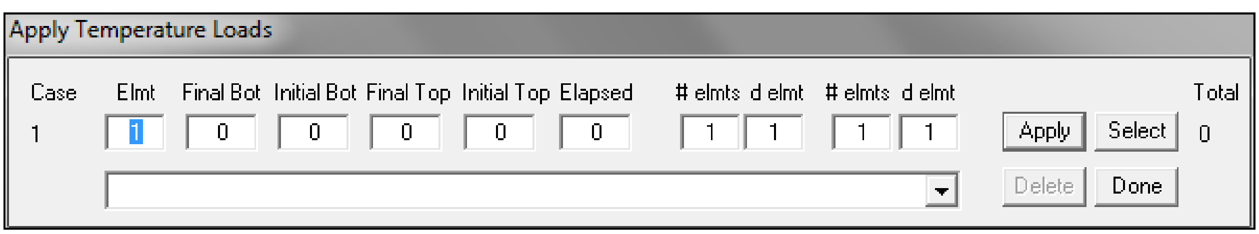
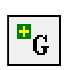


Figure 5-15 – Apply Temperature Loads window

5.4.2.4 Gravitational Loads 

Select the **Load/Apply Gravitational Loads** menu item or click the **Gravitational Loads** toolbar button. The **Apply Gravitational Loads** dialog box appears as shown in Figure 5-16. The only unique input parameter to Shell Structures is **Gamma**, the angular velocity with respect to the global Z-direction, in m/s.

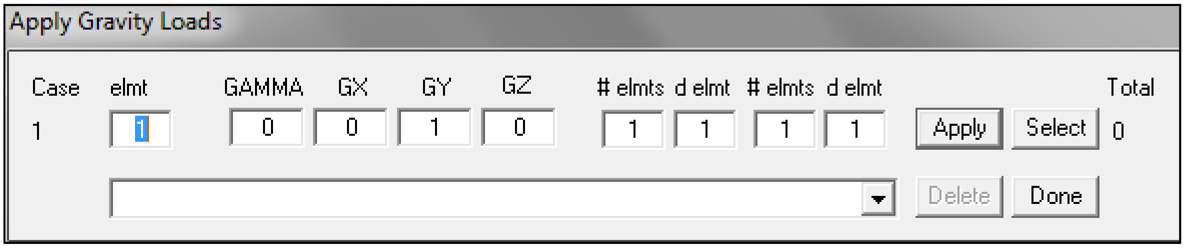


Figure 5-16 – Apply Gravity Loads window

5.5 A Simple Example in VecTor4

In this section, a simple example illustrating the FormWorks-Plus modeling procedure for Shell Structures is presented. The purpose is to analyze the failure of a reinforced concrete tank used to supply agricultural water for farms in India. The problem was referred to the Department of Civil Engineering of Uttar Pradesh for the remedial measures. Figure 5-17 demonstrates deterioration of concrete due to environmental effects at the ground level of the tank (Mital, et al., 2008).

Figure 5-17 – A view of the tank at the ground level, showing deterioration of concrete due to environmental effects (Mital, et al., 2008).

Unfortunately, the exact detailed design of the reinforced concrete tank is not available. Figure 5-18 and 5-19 shows the geometry and the cross sectional properties of a water tank which is similar to the one that failed in India.

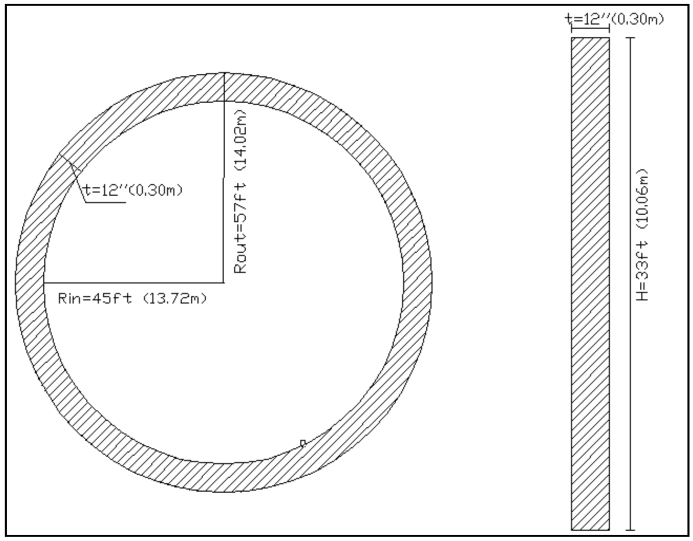


Figure 5-18 – Dimensions of the structure

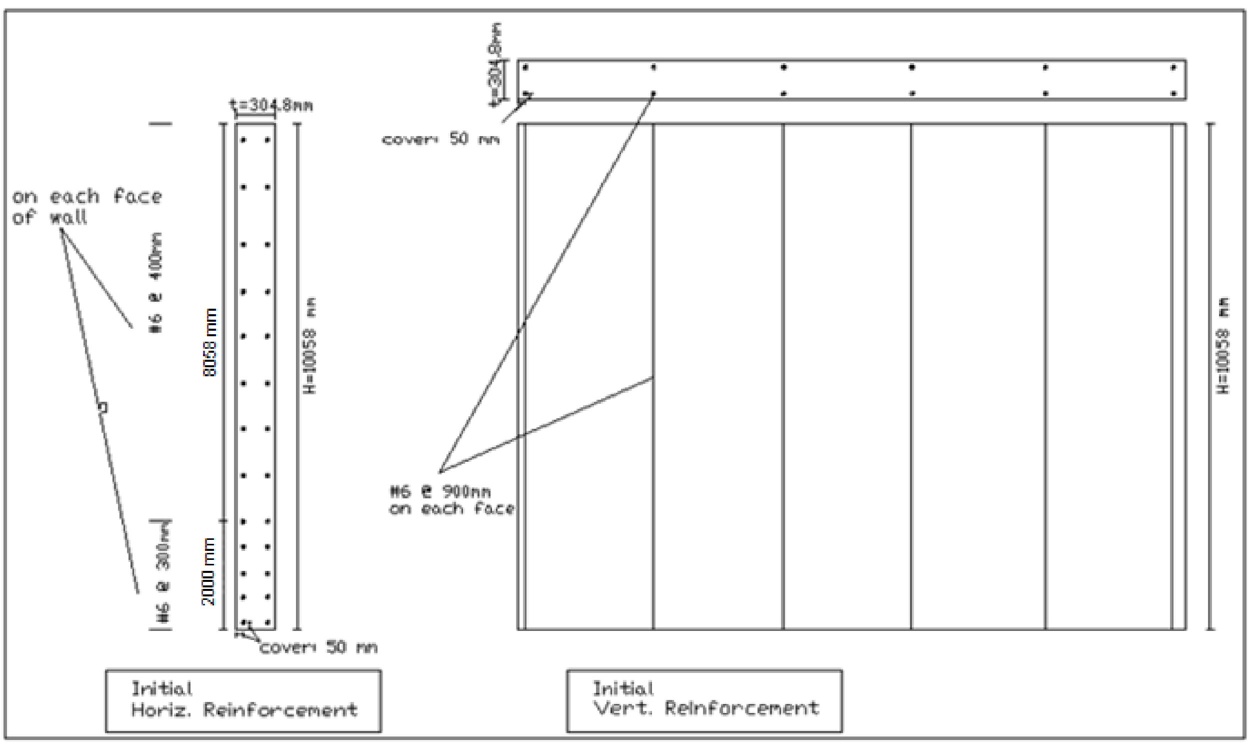


Figure 5-19 – The cross sectional properties

The finite element model created by FormWorks-Plus includes 60 nodes and 30 elements for each circumferential band, with 10 bands of elements used to model the entire height. This results in a total number of 1260 nodes and 300 shell elements. Figure 5-20 and 5-21 show the element numbering and node numbering in the finite element model, respectively.

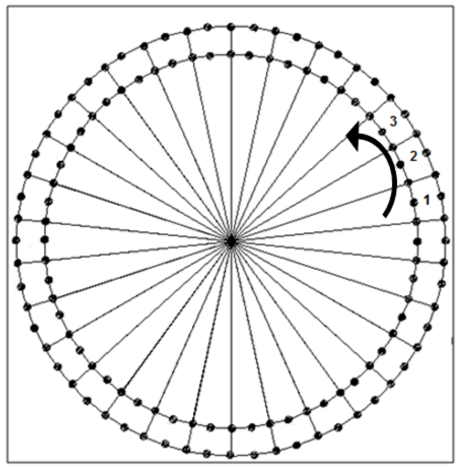


Figure 5-20 – The mesh used for the finite element model-element numbering

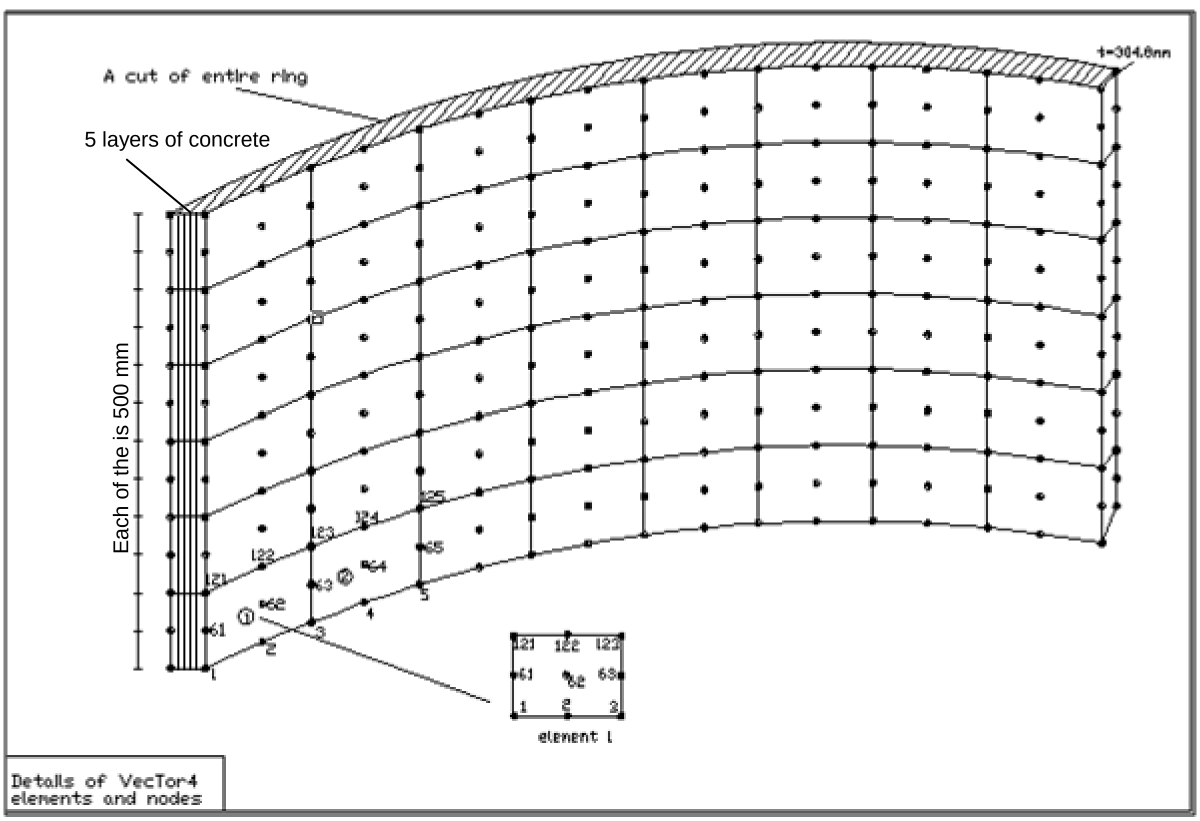


Figure 5-21 – The mesh used for the finite element model-node numbering

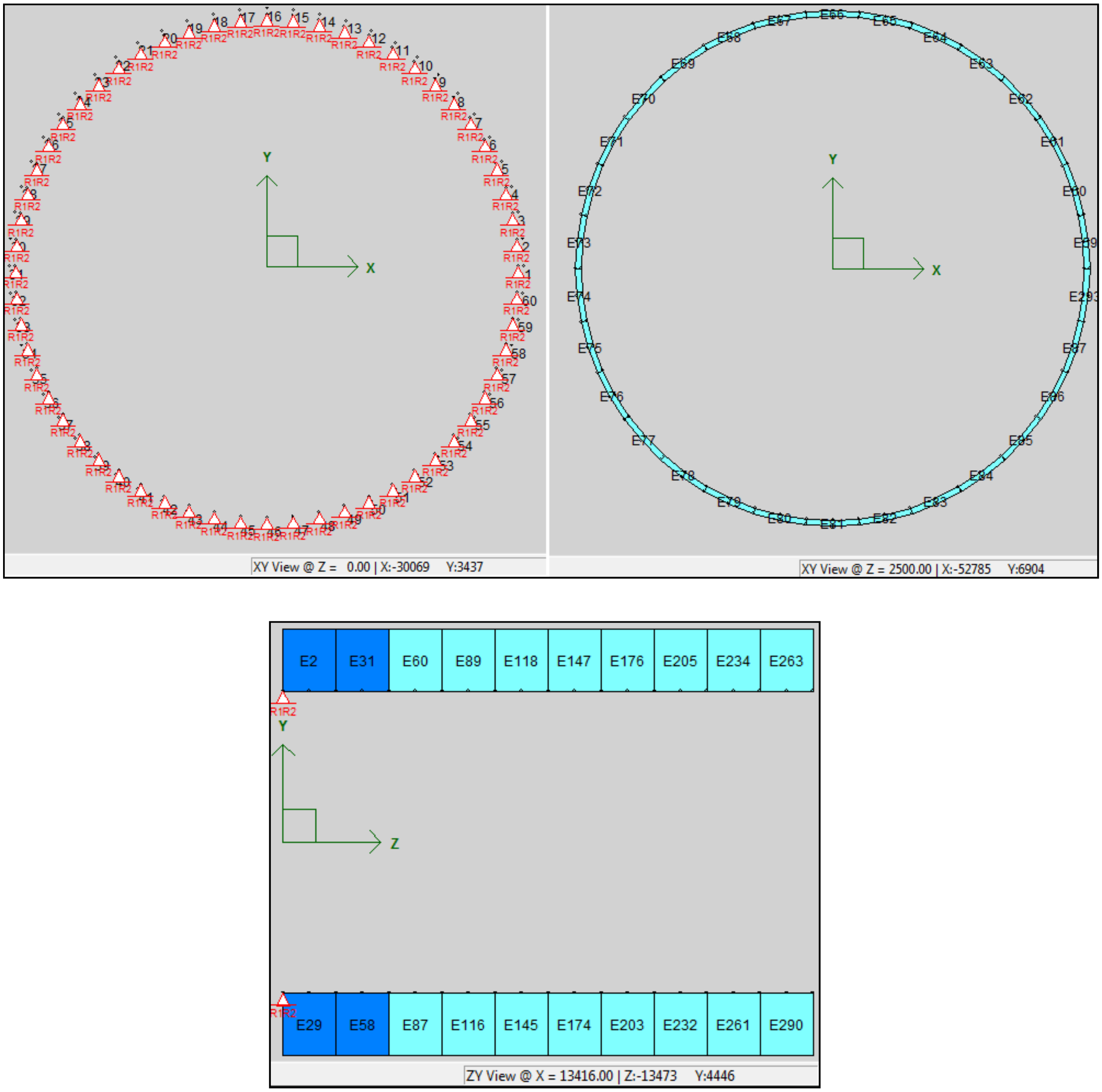


Figure 5-22 – The finite element model created with FormWorks-Plus

Material Type	Concrete Properties							
	Colour	T mm	# of Layers	F'c MPa	F't MPa	Ec MPa	e0 me	Out-of- Plane Reinforcement
1		304.8	5	27.6	1.73	27504	2.007	---
2		304.8	5	27.6	1.73	27504	2.007	---

Table 5-1 (A) – Material properties defined in FormWorks-Plus- Concrete Properties

Material Type	Reinforcement Component Properties									
	Comp. #	Direction	As	Dist. - Top	Db	Fy	Fu	Es	esh	eu
		Degree	mm^2/m	mm	mm	MPa	MPa	MPa	me	me
1	1	90	947	50.8	19.05	414	500	200000	20	200
	2	90	947	254	19.05	414	500	200000	20	200
	3	0	315	50.8	19.05	414	500	200000	20	200
	4	0	315	254	19.05	414	500	200000	20	200
2	1	90	710	50.8	19.05	414	500	200000	20	200
	2	90	710	254	19.05	414	500	200000	20	200
	3	0	315	50.8	19.05	414	500	200000	20	200
	4	0	315	254	19.05	414	500	200000	20	200

Table 5-1 (B) – Material properties defined in FormWorks-Plus- Reinforcement Components

The structure is analyzed under two load cases, self-weight and water pressure. The gravity load type with zero increment is used to apply the self-weight of structure. To define water pressure, the pressure corresponding to each layer of elements is calculated and applied as a uniformly distributed load in FormWorks-Plus. Figure 5-24 and 5-25 shows the deflected shape and load-deformation response of structure, respectively.

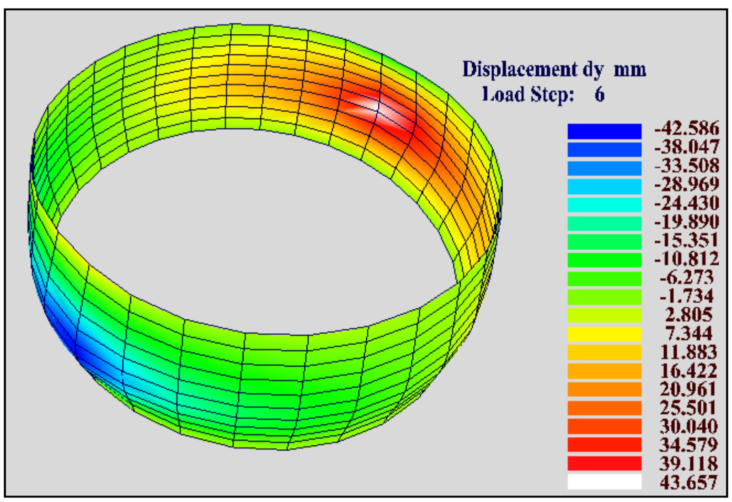


Figure 5-23 – Deflected shape of structure

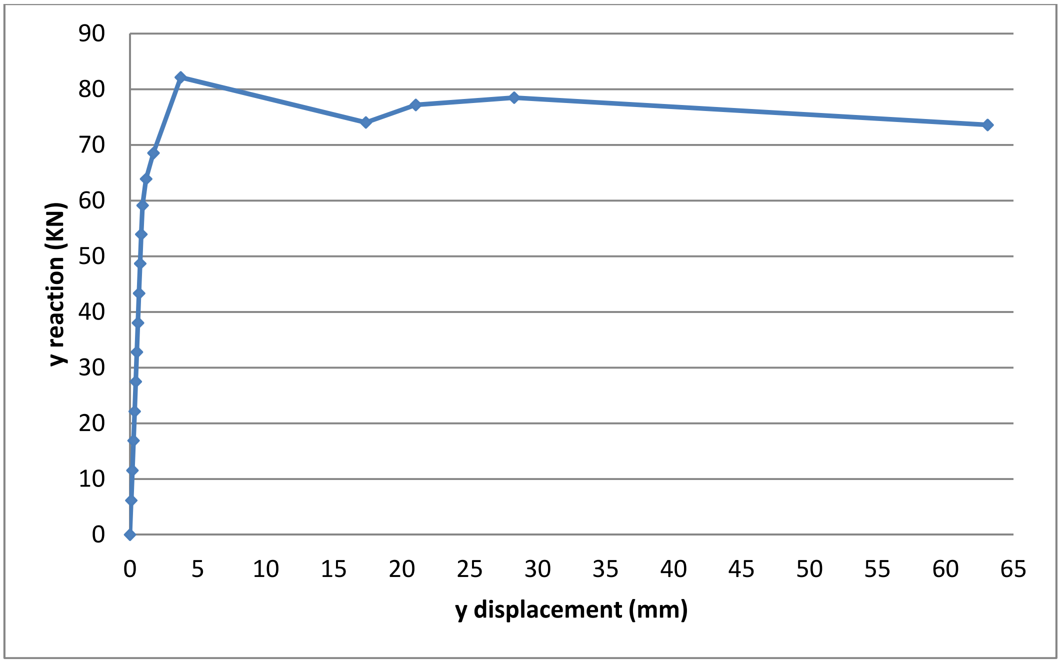


Figure 5-24 – Load-deflection response of structure

Chapter 6

Modeling 2D Frame Structures

6.1 Introduction

VecTor5 is a nonlinear sectional analysis program for reinforced concrete plane frame structures. Temperature, static and dynamic loading conditions are available in VecTor5. Temperature loads include nonlinear thermal gradients; static loads include monotonic, cyclic and reversed-cyclic load cases; dynamic loads include base accelerations (time-history analysis under an input accelerogram), impulse, impact and blast loads, initial velocity and constant acceleration load cases. Based on the Disturbed Stress Field Model (Vecchio, 2000) which is derived from the Modified Compression Field Theory (Vecchio and Collins, 1986), VecTor5 uses a smeared rotating crack approach for reinforced concrete based on a total load, secant stiffness formulation.

The computational algorithm performs two interrelated analyses. First, VecTor5 performs a global frame analysis based on direct stiffness method. In the next step, by using a distributed nonlinearity fibre model approach, rigorous sectional analyses of concrete member cross sections are performed at various sections along the lengths of the members. The computed responses are enforced with the use of an unbalanced force approach where the unbalanced forces are reduced to zero iteratively (Guner, 2008).

VecTor5 is capable of considering second order effects including material and geometric nonlinearities, time- and temperature-related effects, membrane action, nonlinear degradation of concrete and reinforcement under elevated temperatures, concrete compression softening, tension stiffening and tension softening, shear slip along crack surfaces, nonlinear concrete expansion, confinement effects, previous loading history, effects of slip distortions on element compatibility relations, concrete prestrains and reinforcement dowel action.

The program is currently configured to accommodate: 1200 elements, 1000 nodes. 50 material types, 110 concrete layers per material type, and 30 rebar components per material type.

Creating the structural model in text files without any graphical tool is a complex and time consuming process. FormWorks-Plus is a user-friendly pre-processor, capable of defining plane frame structures, applying different types of loads, assigning material properties, and specifying analysis parameters which are compatible with VecTor5. The program allows the user to define nodes and members based on the geometry, including joint regions, and specify the properties of concrete layers and rebar components for each material type. This chapter will discuss the different parts of FormWorks-Plus that are involved in the modeling of a 2D frame structure to be analyzed with VecTor5.

6.2 The Job Data

For the most part, the input properties required to define the Job Page for VecTor5 are similar to the input requirements for VecTor4. For further instruction on how to enter these parameters, see Section 2 in Chapter 5. The Auxiliary page is the only part that has input parameters unique to Plane Frame Structures:

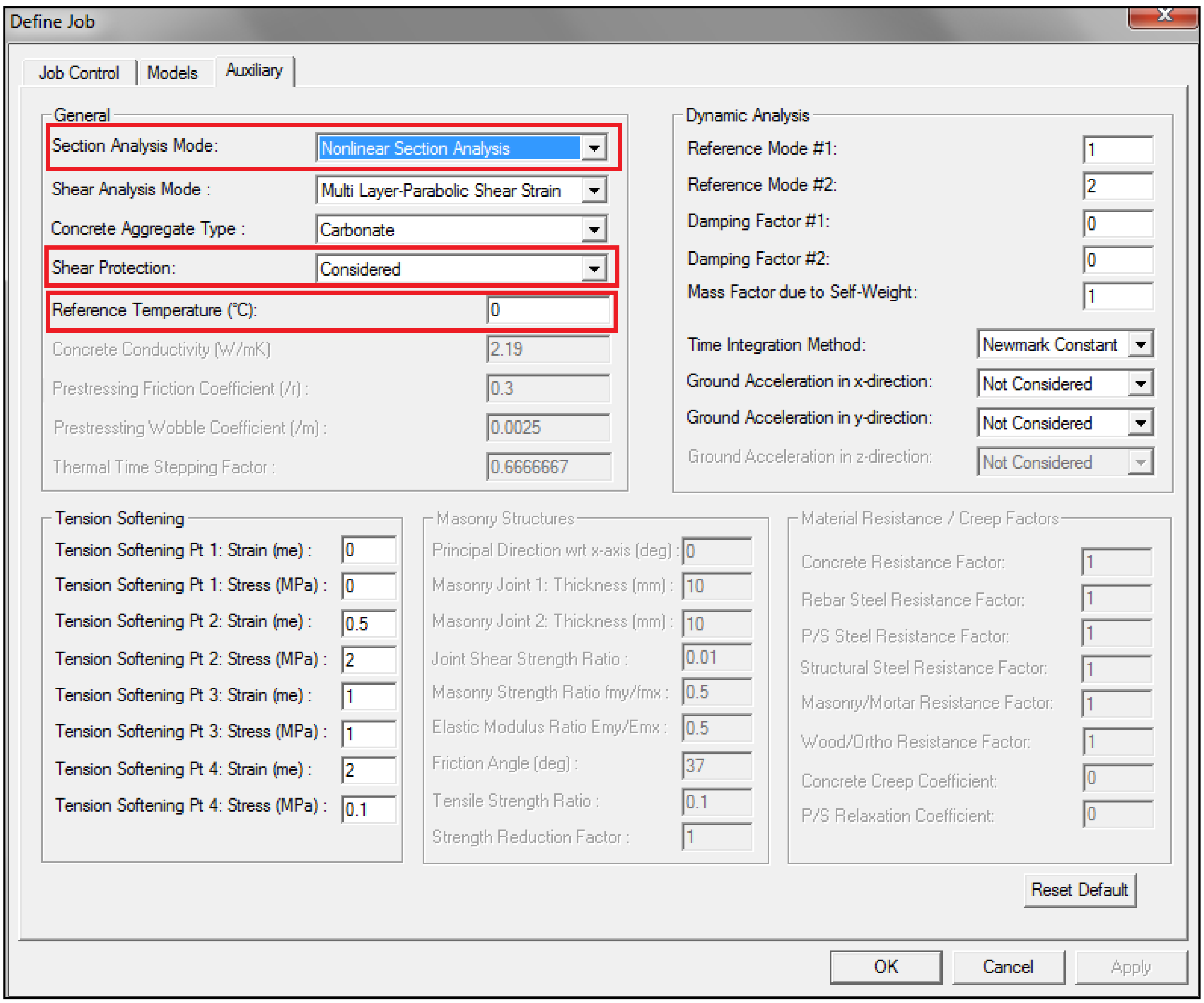


Figure 6-1 – The Auxiliary Page for Plane Frame Structures

- **Section Analysis Mode**

  Select one of the following models for the Section Analysis Mode:

  1. Nonlinear Section Analysis
  2. Effective Stiffness (Branson's formula)
  3. Cracked/Uncracked (ACI349)
  4. Uncracked (Using gross section stiffness)
  5. Fully Cracked (Using cracked section stiffness)

  The default model is option 1, Nonlinear Section Analysis.

- **Shear Protection**

  Experience with VecTor5 has shown that regions where strain distribution is significantly nonlinear such as near concentrated loads, corners, supports and other discontinuities are vulnerable to premature shear failures (Guner, 2008). To account for this phenomenon in discontinuity regions, commonly known as D-regions, an algorithm called shear protection was introduced into the proposed analysis procedure. Figure 6-2 shows D- and B-regions (where B refers to beam) for a reinforced concrete frame (Schlaich et al., 1987).

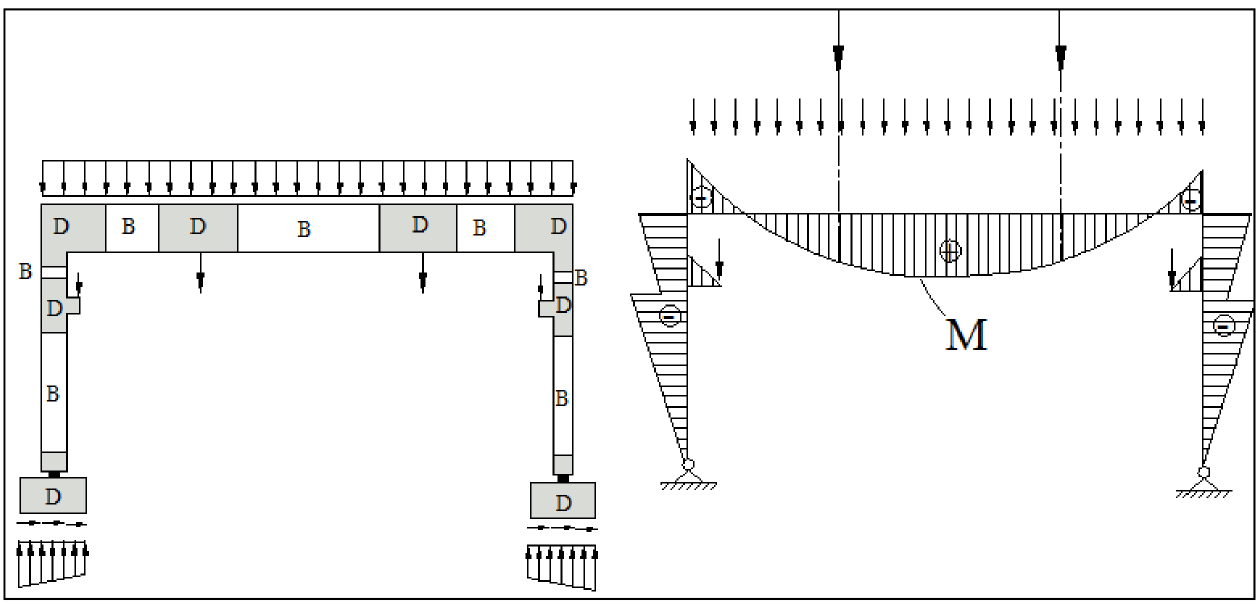


Figure 6-2 – A Reinforced Concrete Frame: (a) B- and D-Regions; (b) Bending Moment Distribution (Schlaich et al., 1987)

In FormWorks-Plus, the shear protection algorithm is considered by default. To turn this feature off, select "Not Considered" from the drop list.

- **Reference Temperature**

  The reference temperature is the ambient temperature, in °C, against which the top and bottom sectional temperatures are defined in a thermal analysis.

6.3 The Structure Data

Having the extension *.S5R, the Structure Data File contains information regarding nodal coordinates, member indices, support restraints, member specifications, concrete layers, rebar layers, and detailed member output list.

6.3.1 Specifying Material Properties

VecTor5 and FormWorks-Plus allow the user to define up to 50 material types. Each material type contains a number of concrete layers and reinforcement components which must be defined from the top of the section. Some of the input properties required to define the reinforced concrete material are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual. The input parameters unique to VecTor5 are shown in Figure 6-3 with highlighted boxes:

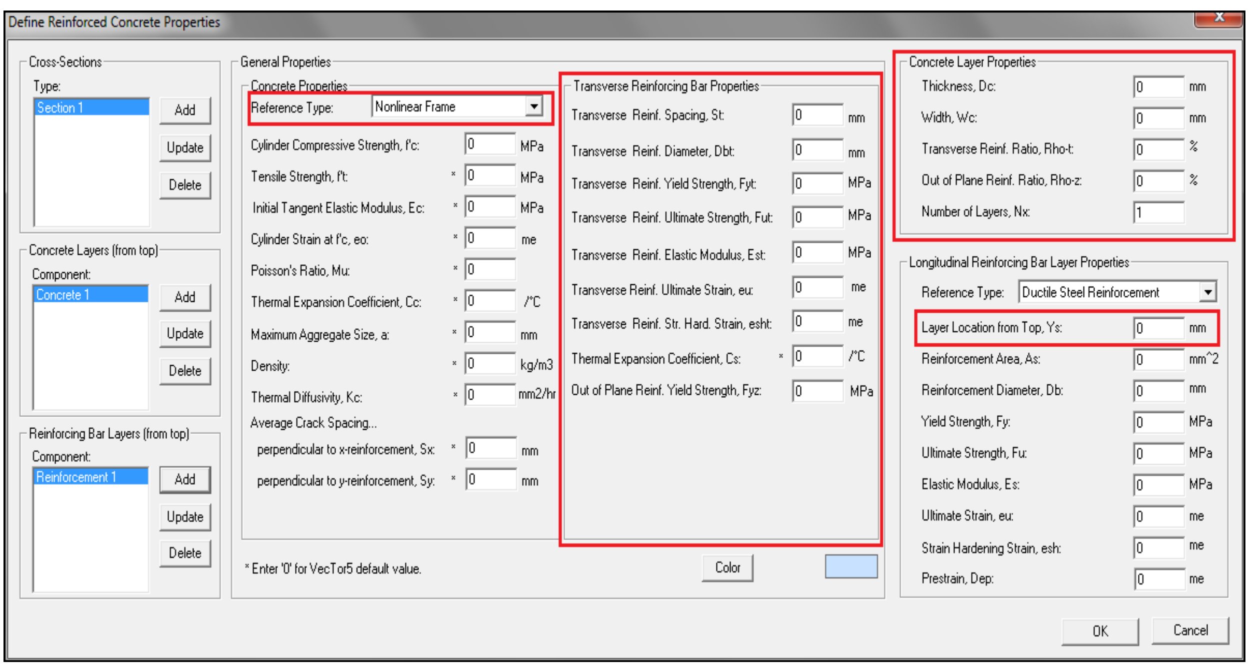


Figure 6-3 – Define Reinforced Concrete Properties window for Plane Frame Structures

6.3.1.1 General Properties

- **Reference type:**

  Select one of the reference types shown in Figure 6-4 to specify the member behaviour and nodal degrees-of-freedom.

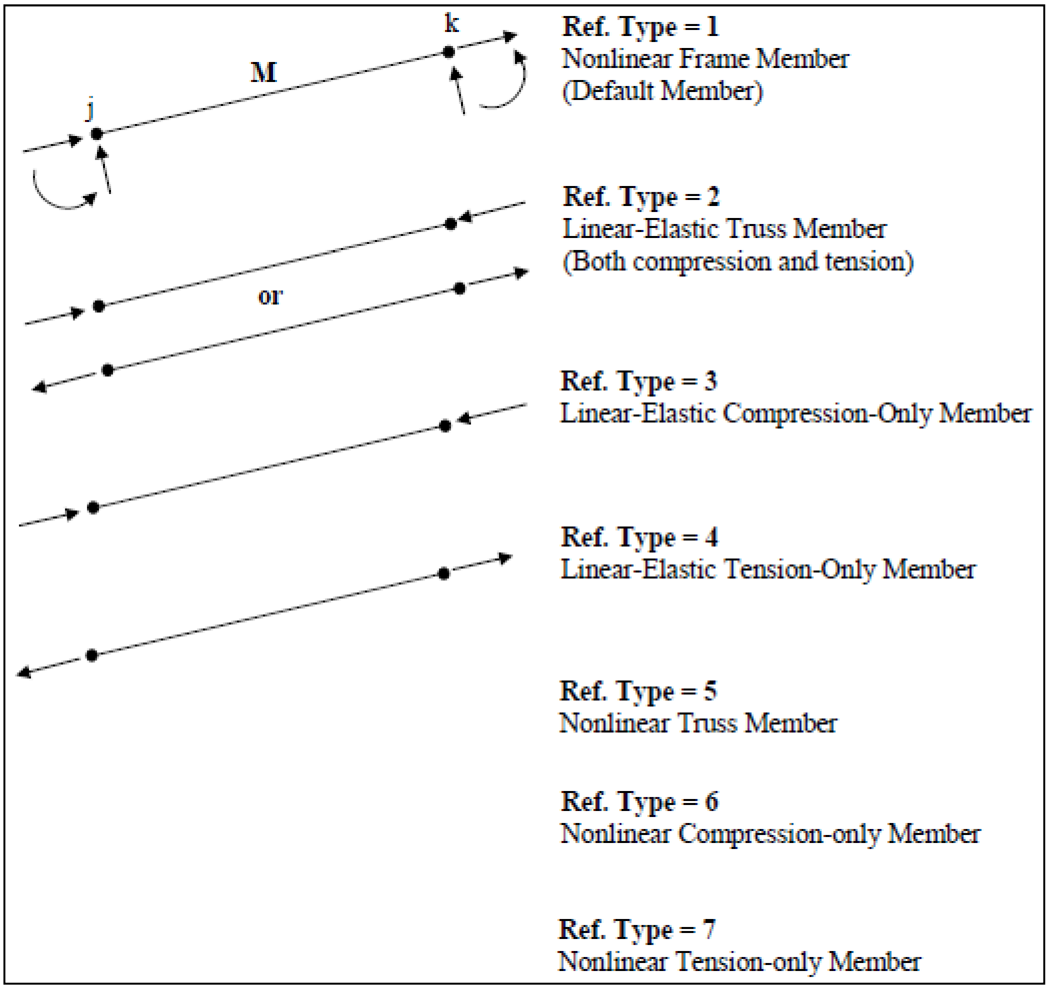


Figure 6-4 – Member reference types (Guner, 2008)

- **Transverse Reinforcing Bar Properties:**

  Enter the properties of the transverse and out-of-plane reinforcement.

6.3.1.2 Concrete Layer Properties

- **Thickness, Dc:**

  Enter the thickness of the layer, in mm.

- **Width, Wc:**
  Enter the width of the cross section, in mm.

- **Transverse Reinf. Ratio, Rho-t:**
  Enter the ratio of the transverse reinforcement in the corresponding concrete layer, in %.

- **Out-of-Plane Reinf. Ratio, Rho-z:**
  Enter the ratio of the out-of-plane reinforcement in the corresponding concrete layer, in %.

- **Number of Layers, Nx:**
  Enter the number of concrete layers with identical details. The default value is 1.

6.3.1.3 Longitudinal Reinforcing Bar Layer Properties

The only unique input parameter is **Ys,** the location of the longitudinal reinforcement layer measured from the top of the cross section, in mm.

6.3.2 Defining Nodes and Members

The procedure for defining nodes and members for 2D Frame Structures is similar to defining nodes and truss bars for Plane Membrane Structures. VecTor5 uses 2-node frame members which are defined based on orientations described in Figure 6-5, where 1 and 2 show the input order specified by the user for node numbers; NODE1 and NODE2 are the node numbers that the program uses.

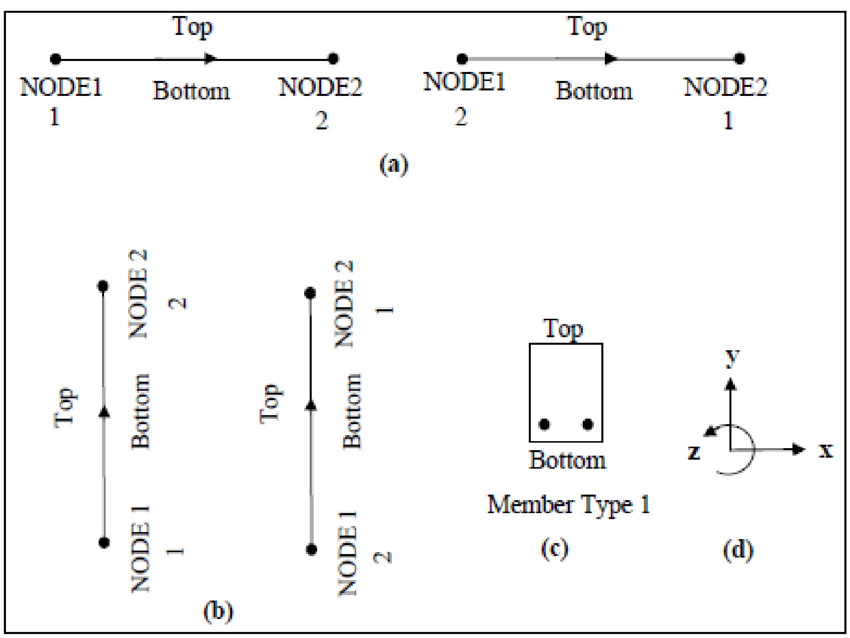
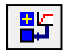
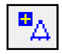


Figure 6-5 – Orientation of frame members (Guner, 2008)

6.3.3 Assigning Materials 

The input properties required to assign material types in VecTor5 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual.

6.3.4 Support Restraints 

The input properties required to assign material types in VecTor5 are similar to the input requirements for VecTor2. For further instruction on how to enter these parameters, refer to the FormWorks manual.

6.4 The Load Case Data

The last step in defining the finite element model is assigning the loads. VecTor5 supports four types of nodal loads, six types of element loads, and ground acceleration load. This section will discuss the different types of loads and required input parameters. Note that the load types, including Nodal Loads, Nodal Displacements, Gravity Loads, Temperature Loads, Ground Acceleration, Impulse Forces, Lumped Masses, and Concrete Prestrains, and the input parameters that are similar to the other types of structures are not described in this section. For instruction on how to define these load types and input parameters, see Section 4 in Chapter 4.

6.4.1 Member End Actions 

Select the **Load/Apply Member End Actions** menu item or click the **Member End Actions** toolbar button. The **Apply Member End Actions** dialog box appears as shown in Figure 6-6. Enter the element number or use the select feature to specify the elements with Member End Actions. To delete a load, select the entry either from the drop list or using the select feature and click the delete button.

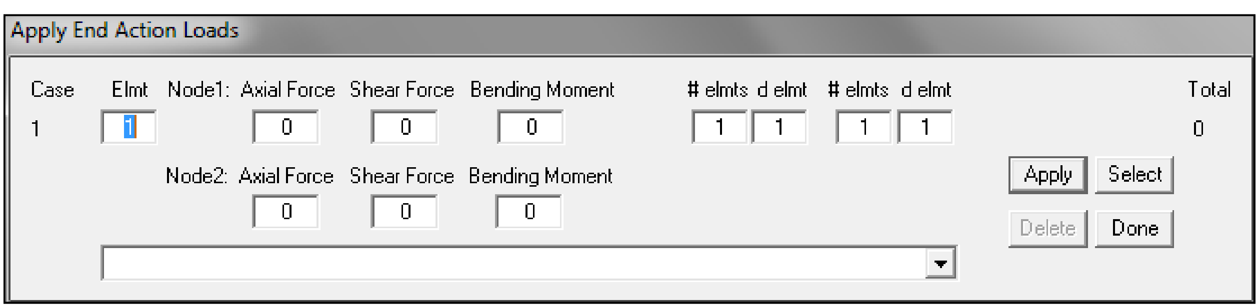


Figure 6-6 – Apply End Action Loads window

Enter axial forces and shear forces, in kN, and bending moments, in kNm, for Node1 and Node2. Positive directions of the member end actions are defined in Figure 6-7.

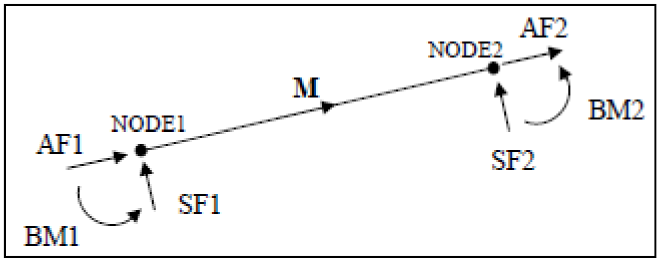


Figure 6-7 – Positive end actions directions for Node1 and Node2

6.4.2 Concentrated Loads 

Select the **Load/Apply Concentrated Loads** menu item or click the **Concentrated Loads** toolbar button. The **Apply Concentrated Loads** dialog box appears as shown in Figure 6-8.

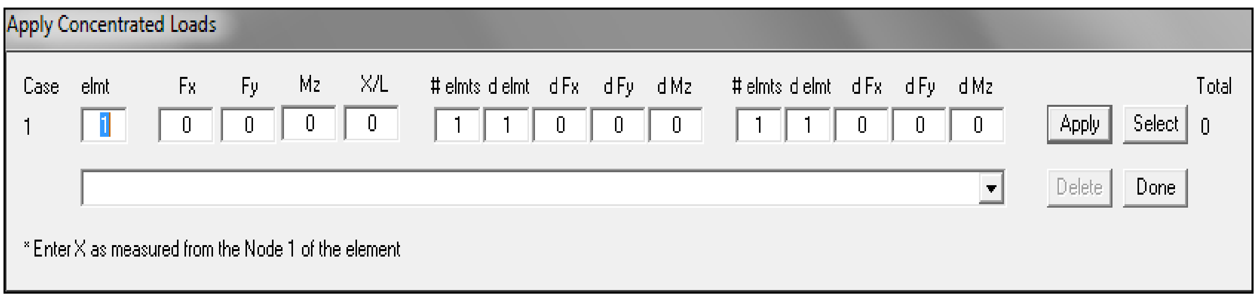


Figure 6-8 – Apply Concentrated Loads window

Concentrated Loads are defined relative to the member-oriented coordinate system as following:

- **Fx:**

  Enter the force acting in a direction oriented from the bottom to the top of the member, in kN.

- **Fy:**

  Enter the force acting in the orientation of the member, in kN.

- **Mz:**

  Enter the moment acting in the counterclockwise direction, in kNm.

- **X/L:**

  Enter the ratio of the distance from NODE1 of the member to the concentrated load application point to the length of the member.

Figure 6-9 shows the positive directions for concentrated loads.

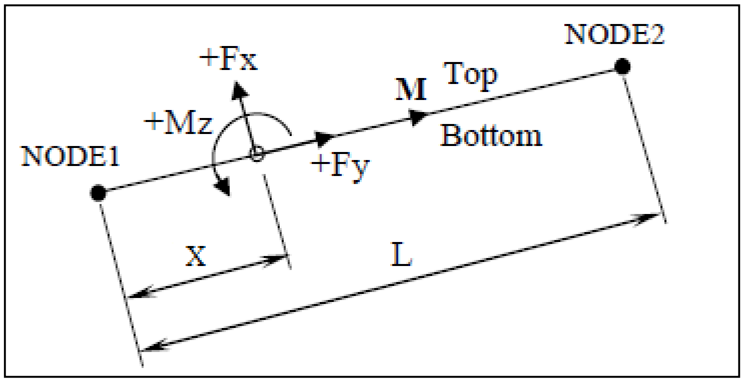


Figure 6-9 – Positive directions for concentrated loads

6.4.3 Uniformly Distributed Loads 

Select the **Load/Apply Uniformly Distributed Loads** menu item or click the **Uniformly Distributed Loads** toolbar button. The **Apply Uniformly Distributed Loads** dialog box appears as shown in Figure 6-10.

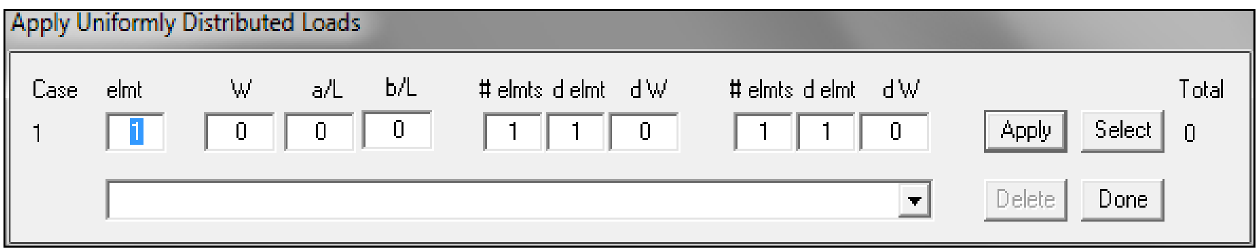


Figure 6-10 – Apply Uniformly Distributed Loads window

Enter the following input parameters for defining the load:

- **W:**

  Enter the uniform load acting in a direction oriented from the top to the bottom of the member, in kN/m.

- **a/L:**

  Enter the ratio of the distance from NODE1 of the member to the starting point of the uniformly distributed load, to the length of the member.

- **b/L:**

  Enter the ratio of the distance from NODE1 of the member to the ending point of the uniformly distributed load, to the length of the member.

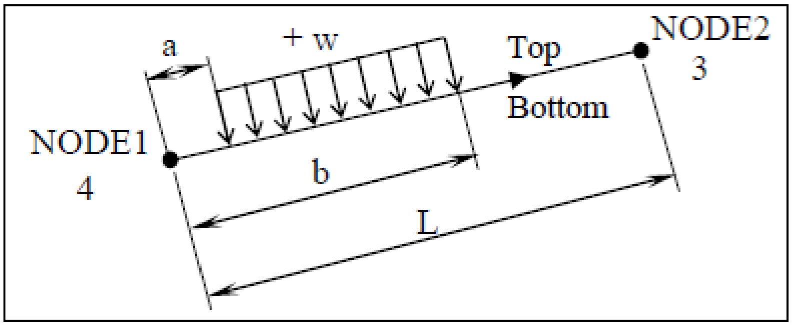


Figure 6-11 – "a" and "b" definitions for defining Uniformly Distributed Loads

6.5 A Simple Example in VecTor5

In this section, a simple example illustrating the FormWorks-Plus modeling procedure for Plane Frame Structures is presented. The objective of the example is to analyze a one-span two-story frame tested at the University of Toronto to investigate shear-related effects on frame deformation, load capacity and failure mechanism. The large-scale frame was designed with a center-to-center span of 3500 mm, a story height of 2000 mm, and an overall height of 4600 mm. Figure 6-12 shows the sectional properties of the frame.

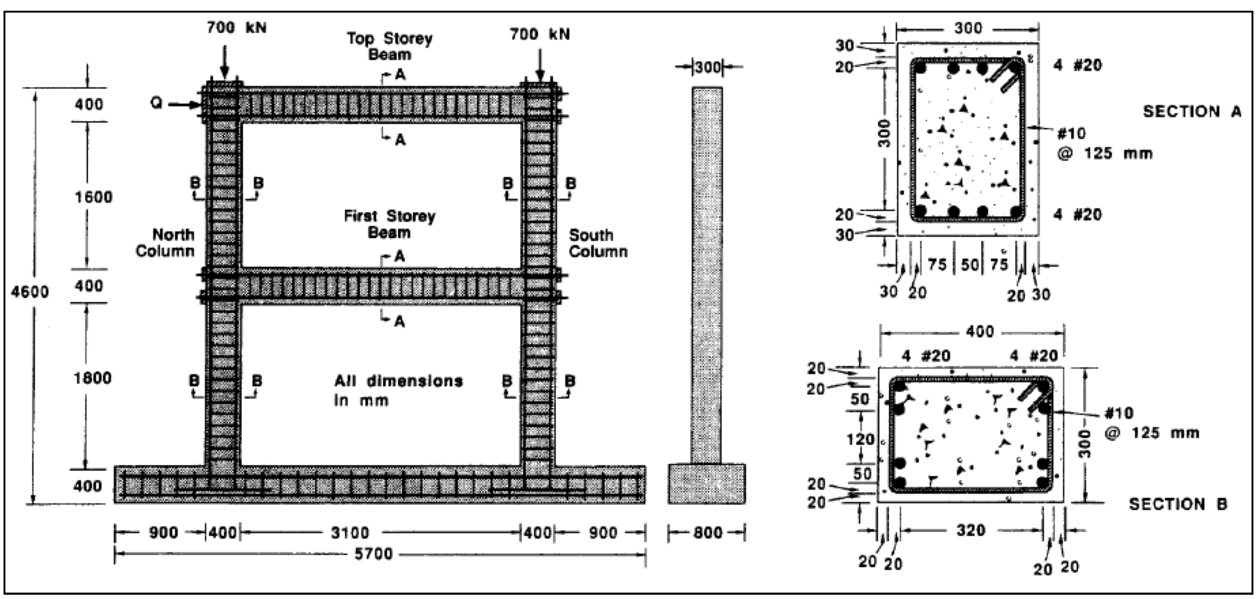


Figure 6-12 – Sectional properties of the frame (Vecchio, et al., 1992).

A total number of 91 nodes and 92 elements were used to create the structural model in FormWorks-Plus as shown in Figure 6-13. To apply loads, two load cases were defined, a lateral load case and a vertical load case. The lateral load was applied as a nodal displacement to the left side of the second story, increased monotonically from zero to failure in increments of 5 kN. A constant axial load of 700 kN was applied to each column as the vertical load case. The results of the analysis are shown in Figure 6-14.

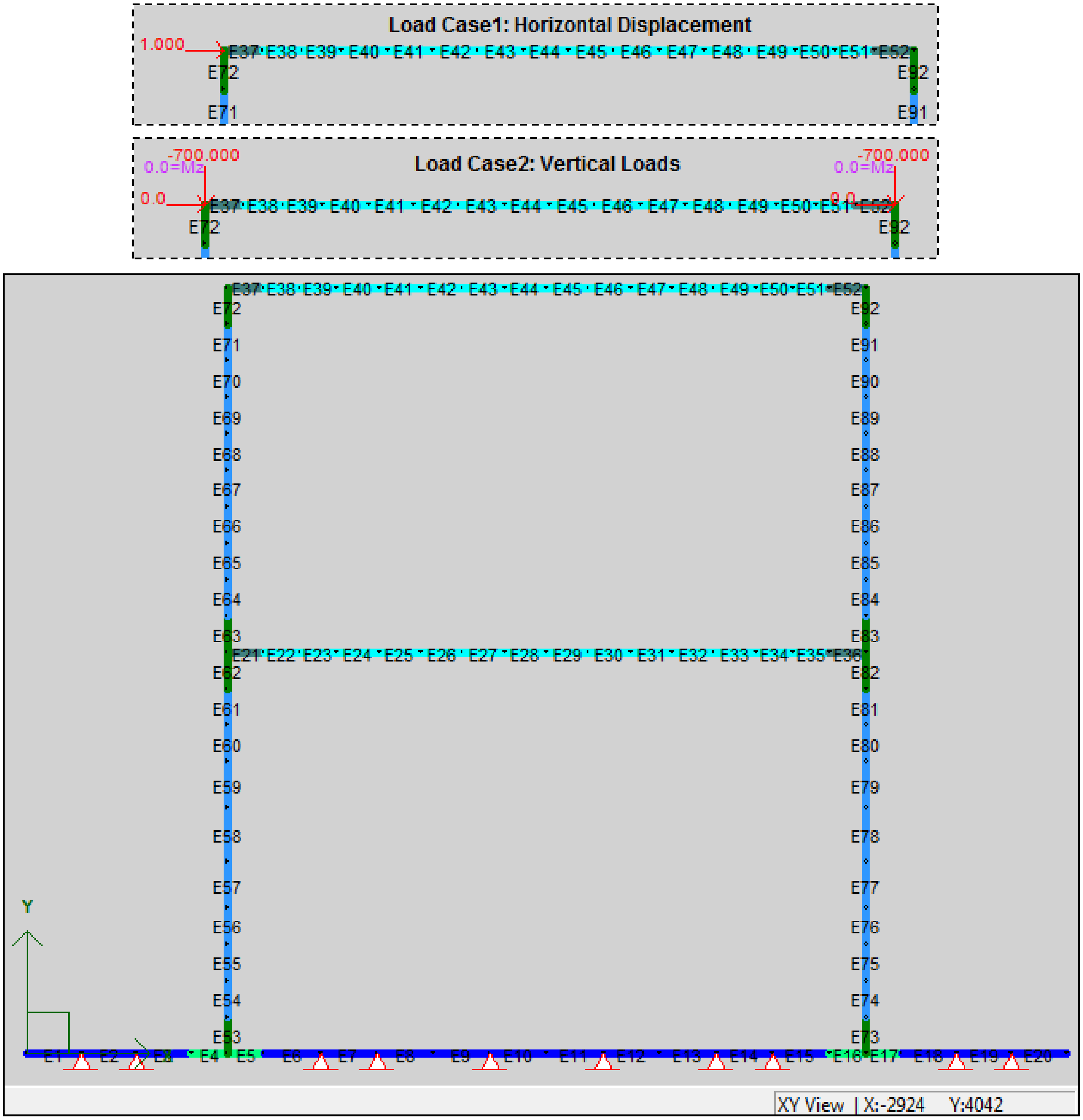


Figure 6-13 – Model created in FormWorks-Plus

Material Type	General Properties												
	Concrete Properties					Transverse Reinforcing Bars Properties							
	Colour	F'c	Ec	e0	Other Properties	St	Dbt	Fyz	Fyt	Fut	Est	eu	esht
		MPa	MPa	me		mm	mm	MPa	MPa	MPa	MPa	me	me
1		30	27386	2.025	default	125	11.3	454	454	640	200000	100	10
2		30	27386	2.025	default	125	11.3	454	454	640	200000	100	10
3		30	27386	2.025	default	200	11.3	454	454	640	200000	100	10
4		30	27386	2.025	default	125	11.3	454	454	640	200000	100	10
5		30	27386	2.025	default	125	11.3	454	454	640	200000	100	10
6		30	27386	2.025	default	200	11.3	454	454	640	200000	100	10

Table 6-1 (A) – Material properties defined in FormWorks-Plus – General properties

Concrete Layer Properties (from top of the section)															
Type	Colour	Comp #	Dc	Wc	Rho-t	Rho-z	Nx	Type	Colour	Comp #	Dc	Wc	Rho-t	Rho-z	Nx
			mm	mm	%	%					mm	mm	%	%	
		1	10	800	0	0.92	4			1	10	300	0	1.6	3
		2	10	800	0.25	0.92	2			2	10	300	1.066	1.6	2
		3	10	800	0.25	0.92	5			3	10	300	1.066	1.6	5
1		4	10	800	0.25	0	18	4		4	10	300	1.066	0	20
		5	10	800	0.25	0.92	5			5	10	300	1.066	1.6	5
		6	10	800	0.25	0.92	2			6	10	300	1.066	1.6	2
		7	10	800	0	0.92	4			7	10	300	0	1.6	3
		1	10	300	0	0.89	2			1	10	300	0	1.78	2
		2	10	300	0.533	0.89	2			2	10	300	1.066	1.78	2
		3	10	300	0.533	0.89	5			3	10	300	1.066	1.78	5
2		4	10	300	0.533	0	22	5		4	10	300	1.066	0	22
		5	10	300	0.533	0.89	5			5	10	300	1.066	1.78	5
		6	10	300	0.533	0.89	2			6	10	300	1.066	1.78	2
		7	10	300	0	0.89	2			7	10	300	0	1.78	2
		1	10	800	0	0.46	4			1	10	800	0	0.92	4
		2	10	800	0.125	0.46	2			2	10	800	0.25	0.92	2
		3	10	800	0.125	0.46	5			3	10	800	0.25	0.92	5
3		4	10	800	0.125	0	18	6		4	10	800	0.25	0	18
		5	10	800	0.125	0.46	5			5	10	800	0.25	0.92	5
		6	10	800	0.125	0.46	2			6	10	800	0.25	0.92	2
		7	10	800	0	0.46	4			7	10	800	0	0.92	4

Table 6-1 (B) – Material properties defined in FormWorks-Plus – Concrete layer properties

Longitudinal Reinforcing Bar Layer Properties (from top of the section)										
Type	Colour	Comp #	Ys	As	Db	Fy	Fu	Es	eu	esh
			mm	mm^2	mm	MPa	MPa	MPa	me	me
1		1	50	1200	19.5	418	596	200000	100	10
		2	350	1200	19.5	418	596	200000	100	10
2		1	40	1200	19.5	418	596	200000	100	10
		2	360	1200	19.5	418	596	200000	100	10
3		1	60	3000	19.5	418	596	200000	100	10
		2	340	3300	19.5	418	596	200000	100	10
4		1	50	2400	19.5	418	596	200000	100	10
		2	350	2400	19.5	418	596	200000	100	10
5		1	40	2400	19.5	418	596	200000	100	10
		2	360	2400	19.5	418	596	200000	100	10
6		1	60	6000	19.5	418	596	200000	100	10
		2	340	6600	19.5	418	596	200000	100	10

Table 6-1 (C) – Material properties defined in FormWorks-Plus – Reinforcement layer properties

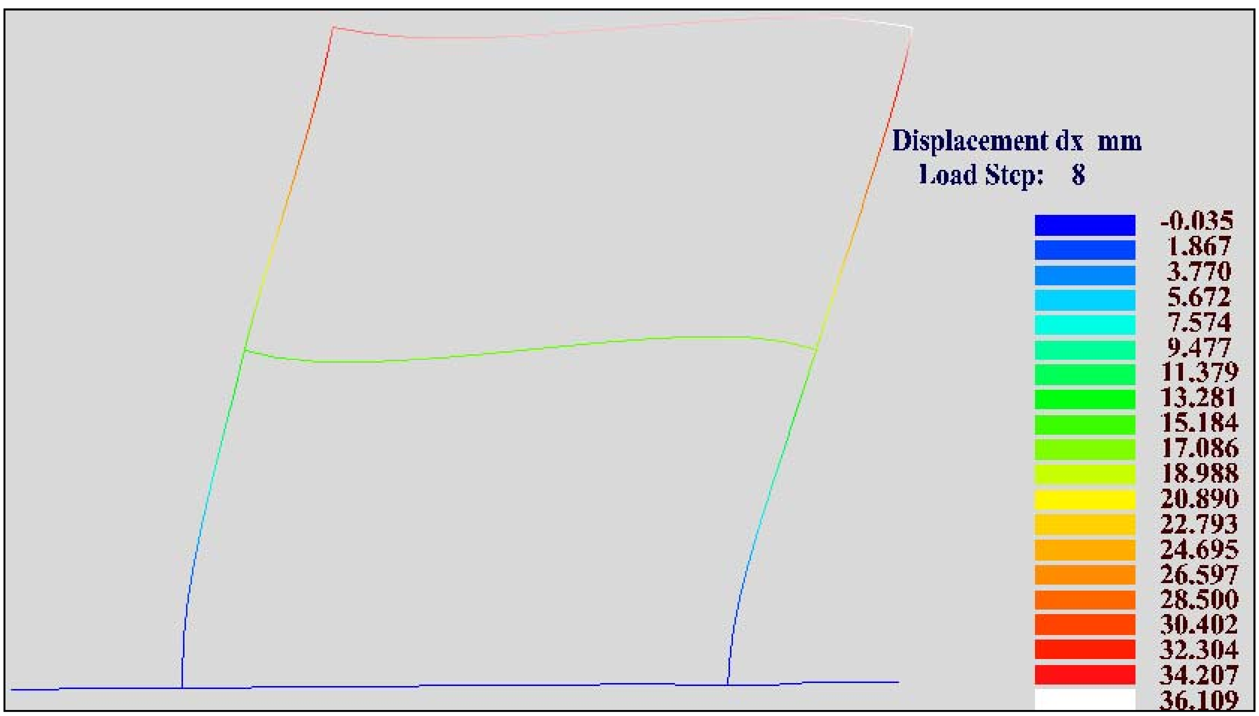


Figure 6-14– Deformed shape of structure

Chapter 7

Summary and Recommendations

7.1 Summary

The VecTor(c) suite of programs has been developed at the University of Toronto for nonlinear analysis of reinforced concrete structures. A graphics-based pre-processor, FormWorks, previously developed for use specifically with VecTor2, greatly facilitates the modeling process of 2D Membrane Structures. However, a user-friendly pre-processor is required for the entire suite of programs if they are to be of greater use by potential users.

The first objective of this thesis was to develop an updated version of FormWorks, FormWorks 3.5, which is more user-friendly and compatible with newer versions of VecTor2. The second objective was to create a new version of FormWorks, FormWorks-Plus, capable of modeling different types of structures including 2D Membrane Structures, 3D Solid Structures, Shell Structures, Plane Frame Structures, and Axisymmetric Solid Structures.

FormWorks 3.5 was mainly developed to model the different types of materials now available in VecTor2 including Reinforced Concrete, Structural Steel, Masonry, Wood, Concrete-Steel Laminate, Concrete-SFRC Laminate, Masonry-SFRC Laminate, Concrete-Ortho Laminate, Reinforcement, and Bond. In addition, the Job Page was updated with the most recent behavioural models and analysis parameters. The zooming feature of FormWorks was improved to display the geometry of the finite element model properly. A new load type, Nodal Thermal Load, was added to the program allowing the user to define thermal loads for nodes. The performance of the truss and rectangular elements was improved. Finally, updated dialog boxes for nodes, elements, restraints, and load types were added to facilitate the modifying process of finite element model.

FormWorks-Plus is a new version of FormWorks, compatible with entire suite of VecTor programs. In addition to the XY, XZ, and ZY plane views, sectional and projection views were added to the program, greatly contributing to its utility. FormWorks-Plus also uses a supporting program called 3D View which is written with the MFC library and OpenGL (Open Graphics Library), allowing the user to see a 3D view of the structural model. In addition, four different types of node coordinate definitions, five types of elements, several load types, and new material properties are added to the software, making it compatible with the remaining VecTor programs.

7.2 Recommendations

The following is a list of features and options that can be added to FormWorks-Plus to improve the program's performance and facilitate the modeling process:

- In the current version of FormWorks-Plus, nodes and elements must be defined manually using the increment feature which gives complete control over the mesh topology, but may be time consuming for complicated geometries. Having an automatic meshing feature or using predefined structures such as beams, columns, slabs, silos, tanks, and frames would allow easier redefinition or refinement of the mesh and save much time.

- Most graphical-based programs have Undo (reverses the last action performed) and Redo (undoes the last Undo action) features allowing the user to easily correct mistakes. Work needs to be done to add these useful features to the program.

- In order to find node coordinates for modeling complex structures, the user is required to draw the model in another program like AutoCAD and then add them in FormWorks-Plus manually. To avoid this time consuming process, an Insert feature can be added to the program to import node coordinates, created by a drawing software, from a text file.

- Defining sectional properties including concrete and reinforcing bar layers for Plane Frame Structures is complicated and requires an expert user with a good understanding of VecTor5. Another method that could be implemented to define sections only based on general properties and compute remaining material parameters automatically.

- The addition of a help option which provides a description of each part of the program would be beneficial.

- The current version of FormWorks-Plus is only capable of displaying the 3D view of a structural model. It would be useful to be able to modify the finite element model such that specifying loads, assigning material types, and redefining the mesh could be done in 3D view as well.

- Works need to be done to improve the displaying capabilities of the program for element loads. For instance, for uniformly distributed loads or pressure loads it would be more useful to display the load pattern with arrows or colour contours instead of just numbers on the elements.

- Different versions of FormWorks use different file extensions to save and open files. Previous versions of FormWorks stored data as a *.fws file. FormWorks 3.5 saves files with a *.fwk extension and finally FormWorks-Plus uses a *.fwp extension. Using different file extensions makes it easier for the user to recognize which version of FormWorks was used to create the file. It would be beneficial to make the program capable of opening all these file extensions.

References

Bazant, Z. P. (2002), “Concrete Fracture Models: Testing and Practice,” Engineering Fracture Mechanics, V. 69, pp 165-205.

Bonaldo, E., de Barros, J. A. O., and Lourenco, P. B. (2006), “Efficient Strengthening Technique for Reinforced Concrete Slabs,” Measuring, Monitoring and Modeling Concrete Properties, pp 125-131.

CEB-FIP (2010), “Model Code 2010,” Comité EURO-International du Béton, Paris, V.1, 311 pp.

Collins, M. P. and Mitchell, D. (1997), “Prestressed Concrete Structures,” Response Publications, 766 pp.

Fanella, D. A. and Naaman, A. (1985), “Stress-Strain Properties of Fiber Reinforced Mortar in Compression,” ACI Journal, 82 (4), pp 475-483.

Guner, S. (2008), “Performance Assessment of Shear-Critical Reinforced Concrete Plane Frames,” PhD. Thesis, Department of Civil Engineering, University of Toronto, 464 pp.

Gutkowski, R. M., Balogh, J., and To, L. G. (2010), “Finite-Element Modeling of Short-Term Field Response of Composite Wood-Concrete Floors/Decks,” Journal of Structural Engineering, pp 707-714.

Hasebe, K. and Usuki, S. (1987), “Application of Orthotropic Failure Criterion to Wood,” Journal of Engineering Mechanics, Vol. 115, No. 4, pp 867-872.

Heiduschke, A. and Kasal, B. (2003), “Composite Cross Section with High Performance Fibre reinforced Concrete and Timbre,” Forest Products Journal, V. 53, No. 10, pp 74-78.

Horton, I. (2008), “Beginning Visual C++ 2008,” Indiana: Wiley Publishing Inc., 1356 pp.

Lourenco, P. J. B. (1996), “Computational Strategies for Masonry Structures,” PhD. Thesis, Department of Civil Engineering, University of Delft, 220 pp.

Mital, V. P., Masood, A., Ahmad, T., and Arif, M. (2008), “Failure of Overhead Water Tank in the State of Uttar Pradesh in India - A Case Study,” First International Conference on Construction in Developing Countries, Karachi, Pakistan.

Naaman, A. E. (2003), “Engineered Steel Fibers with Optimal Properties for Reinforcement of Cement Composites,” Journal of Advanced Concrete Technology, V. 1, No. 3, pp 241-252.

Onsongo, W. (1978), “The Diagonal Compression Field Theory for Reinforced Concrete Beams Subjected to Combined Torsion, Flexure and Axial Load,” PhD. Thesis, Department of Civil Engineering, University of Toronto.

Palermo, D., Abdulridha, A., and Foo, S. (2011), “Super elastic SMA Reinforced Concrete Elements: Applicability and Practicality,” fib Symposium, Session 2B-9, Prague.

Polak, M. A. (1992), “Nonlinear Analysis of Reinforced Concrete Shells,” PhD. Thesis, Department of Civil Engineering, University of Toronto, 195 pp.

Prosise, J. (1999), “Programming Windows with MFC,” Washington: Microsoft Press.

Schlaich, J., Schäfer, K., and Jennewein, M. (1987), “Toward a Consistent Design of Structural Concrete,” PCI Journal, Issue 32, pp 74-150.

Schlöglmann, K. H. (2004), “Structural Behaviour of Masonry Panels and their Rehabilitation Focussing on Lifeline Structures Subjected to Seismic Loads,” Diploma Thesis, Institute for Structural Concrete, Graz, 216 pp.

Segal, M. and Akeley, K. (2006), “The OpenGL Graphics System,” California: Silicon Graphics Inc., 333 pp.

Selby, R. G. (1990), “Nonlinear Finite Element Analysis of Reinforced Concrete Solids,” M.A.Sc. Thesis, Department of Civil Engineering, University of Toronto, 195 pp.

Seracino, R. (1995), “Towards Improving Nonlinear Analysis of Reinforced Concrete Shells,” M.A.Sc. Thesis, Department of Civil Engineering, University of Toronto, 194 pp.

Vecchio, F. J. (2000), “Disturbed Stress Field Model for Reinforced Concrete: Formulation,” Journal of Structural Engineering, V.126, No.9, pp 1070-1077.

Vecchio, F. J. and Collins, M. P. (1986), “The Modified Compression-Field Theory for Reinforced Concrete Elements Subjected to Shear,” ACI Journal, V.83, No.2, pp 219-231.

Vecchio, F. J. and Emara, M. B. (1992), “Shear Deformations in Reinforced Concrete Frames,” ACI Structural Journal, pp 46-56.

Vecchio, F. J. and McQuade, I. (2011), “Towards Improved Modelling of Steel-Concrete Composite Wall Elements,” Nuclear Engineering and Design, 241, pp 2629-2642.

Vecchio, F. J. and Selby, R. G. (1990), “Toward Compression-Field Analysis of Reinforced Concrete Solids,” Journal of Structural Engineering, V. 117, No. 6, pp 1740-1757.

Vecchio, F. J. and Zhou, C. E. (2004), “Nonlinear Finite Element Analysis of Reinforced Concrete Structures Subjected to Transient Thermal Loads,” Computers and Concrete, V. 2, No. 6, pp 455-479.

Wong, P. (2002), “User Facilities for 2D Nonlinear Finite Element Analysis of Reinforced Concrete,” M.A.Sc. Thesis, Department of Civil Engineering, University of Toronto, 213 pp.