

# USER'S MANUAL OF VECTOR5

by

Serhan Guner

Frank J. Vecchio

September 2008

© Copyright by S. Guner and F. J. Vecchio (2008)

## ABSTRACT

---

VecTor5 is a computer program for nonlinear sectional analyses of two dimensional frame-related structures consisting of beams, columns and shear walls, subjected to temperature, static and dynamic loading conditions. Based on the Modified Compression Field Theory (MCFT) and the Disturbed Stress Field Model (DSFM), VecTor5 uses a smeared, rotating crack approach for reinforced concrete based on a total load, secant stiffness formulation.

The purpose of this report is to present the program documentation of VecTor5. This documentation discusses the analysis process of VecTor5 by describing the preparation of input text files and the interpretation of output text files. The input text files for two example analyses are also presented in the appendices.

This report is intended to be a comprehensive and practical manual for the analyst, describing the preparation of input files through numerical examples. In addition, when necessary, the appropriate use of the different formulations and options are discussed. The theoretical basis for the program can be found in Guner (2008).

This manual, the basic version of VecTor5, the PhD thesis of Guner, the contact information of the authors and other pertinent information can be found in the VecTor Analysis Group Website at '[www.civ.utoronto.ca/vector](http://www.civ.utoronto.ca/vector)'.

---

# TABLE OF CONTENTS

---

<b>ABSTRACT</b>	<b>ii</b>
<b>LIST OF FIGURES</b>	<b>iv</b>
<b>LIST OF TABLES</b>	<b>v</b>
<b>CHAPTER M1 - INTRODUCTION</b>	<b>1</b>
<b>CHAPTER M2 - ANALYSIS WITH VECTOR5</b>	<b>2</b>
M2.1 Input Data Files	2
M2.2 Expanded Data Files	3
M2.3 Performing an Analysis	4
M2.4 Output Files	5
M2.5 Analysis Process of VecTor5	6
<b>CHAPTER M3 - VECTOR5 INPUT FILES</b>	<b>7</b>
M3.1 Structure Data File (.S5R)	7
M3.2 Load Data Files (.S5L)	19
M3.3 Job Data File (VecTor.JOB)	36
M3.4 Auxiliary Data File (VT5.AUX)	47
<b>CHAPTER M4 - VECTOR5 OUTPUT FILES</b>	<b>51</b>
M4.1 Output Files (Analysis Results)	51
M4.2 Determination of Damage or Failure Modes	59
<b>REFERENCES</b>	<b>64</b>
<b>APPENDIX M1 - Example Static Analysis (Monotonic Load)</b>	<b>65</b>
<b>APPENDIX M2 - Example Dynamic Analysis (Impact Load)</b>	<b>77</b>

---

## LIST OF FIGURES

---

<b>Figure M.1</b> A Screen Shot during the VecTor5 Analysis	5
<b>Figure M.2</b> Schematic Representation of Analysis Process of VecTor5	6
<b>Figure M.3</b> Orientation of Frame Members: <b>(a)</b> Horizontal Member; <b>(b)</b> Vertical Member <b>(c)</b> Member Cross Section; <b>(d)</b> Global Coordinate System	10
<b>Figure M.4</b> Structure Data File: Member Reference Types	15
<b>Figure M.5</b> Beam VS-A1: <b>(a)</b> Cross Section Details; <b>(b)</b> Sectional Model	16
<b>Figure M.6</b> <b>(a)</b> A Member with Nodal Loads; <b>(b)</b> Global Coordinate system	20
<b>Figure M.7</b> Part of a Structural Model with Nodal Loads	21
<b>Figure M.8</b> Member M with End Actions in the Member-Oriented Coord. System	22
<b>Figure M.9</b> A Member with Concentrated Load in the Global Coordinate System	23
<b>Figure M.10</b> A Member with UDL in Member-Oriented Coordinate System	24
<b>Figure M.11</b> <b>(a)</b> A Member with Automatically Calculated Gravity Loads; <b>(b)</b> Degrees-of-Freedom for Gravity Loads	25
<b>Figure M.12</b> <b>(a)</b> Member M in Reference (Ambient) Temperature of $T$ °C; <b>(b)</b> Input Parameters for Temperature Loading	27
<b>Figure M.13</b> <b>(a)</b> A Member with Prescribed Nodal Displacements; <b>(b)</b> Degrees-of-Freedom for Prescribed Nodal Displacements	28
<b>Figure M.14</b> <b>(a)</b> A Member with Additional Lumped Masses, Initial Velocity and Const. Acceleration Loading; <b>(b)</b> DOF for Additional Lumped Masses	30
<b>Figure M.15</b> Multi-Linear Force-Time History	32
<b>Figure M.16</b> Multi-Linear Force-Time History with Automatically Added Branches	32
<b>Figure M.17</b> Suddenly Appearing Multi-Linear Force-Time History	33
<b>Figure M.18</b> Ground Acceleration-Time History Loading (Accelerogram)	34
<b>Figure M.19</b> VecTor.EQR Data File for Ground Acceleration Loads	35
<b>Figure M.20</b> Monotonic Loading Condition	38
<b>Figure M.21</b> Cyclic Loading Condition	38
<b>Figure M.22</b> Reversed-Cyclic Loading Condition	39
<b>Figure M.23</b> Concrete Post-Peak Response: Modified Park-Kent (1982)	62

---

## LIST OF TABLES

---

<b>Table M.1</b> Structure Data File: Input Field for General Parameters	7
<b>Table M.2</b> Structure Data File: Input Field for Nodal Coordinates (Regular Input)	8
<b>Table M.3</b> Structure Data File: Input Field for Nodal Coordinates (Compact Input)	8
<b>Table M.4</b> Structure Data File: Input Field for Nodal Coordinates (Mixed Input)	9
<b>Table M.5</b> Structure Data File: Input Field for Member Indices (Regular Input)	9
<b>Table M.6</b> Structure Data File: Input Field for Member Indices (Compact Input)	9
<b>Table M.7</b> Structure Data File: Input Field for Support Restraint List (Regular Input)	11
<b>Table M.8</b> Structure Data File: Input Field for Member Specifications	11
<b>Table M.9</b> Structure Data File: Input Field for Concrete Layers	15
<b>Table M.10</b> Structure Data File: Input Field for Steel Layers	17
<b>Table M.11</b> Structure Data File: Input Field for Detailed Member Output List (Regular Input)	18
<b>Table M.12</b> Structure Data File: Input Field for Detailed Member Output List (Mixed Input)	18
<b>Table M.13</b> Load Data File: Input Field for Load Case Parameters	19
<b>Table M.14</b> Load Data File: Input Field for Nodal Loads (Regular Input)	20
<b>Table M.15</b> Load Data File: Input Field for Nodal Loads (Compact Input, Examp. 1)	21
<b>Table M.16</b> Load Data File: Input Field for Nodal Loads (Compact Input, Examp. 2)	21
<b>Table M.17</b> Load Data File: Input Field for Nodal Loads (Mixed Input)	22
<b>Table M.18</b> Load Data File: Input Field for Member End Actions	22
<b>Table M.19</b> Load Data File: Input Field for Concentrated Loads	23
<b>Table M.20</b> Load Data File: Input Field for Uniformly Distributed Loads	24
<b>Table M.21</b> Load Data File: Input Field for Gravity Loads (Example 1)	25
<b>Table M.22</b> Load Data File: Input Field for Gravity Loads (Example 2)	25
<b>Table M.23</b> Load Data File: Input Field for Temperature Loads	26
<b>Table M.24</b> Load Data File: Input Field for Concrete Prestrains (Example 1)	27
<b>Table M.25</b> Load Data File: Input Field for Concrete Prestrains (Example 2)	28

<b>Table M.26</b> Load Data File: Input Field for Prescribed Nodal Displacements	28
<b>Table M.27</b> Load Data File: Additional Lumped Masses	30
<b>Table M.28</b> Load Data File: Input Field for Impulse, Blast and Impact Loads	31
<b>Table M.29</b> Load Data File: Input Field for Ground Accelerations	33
<b>Table M.30</b> Job Data File: Input Field for Structure Data	36
<b>Table M.31</b> Job Data File: Input Field for Loading Data (Monotonic Loads)	37
<b>Table M.32</b> Job Data File: Input Field for Loading Data (Cyclic Loads)	39
<b>Table M.33</b> Job Data File: Input Field for Loading Data (Reversed-Cyclic Loads)	39
<b>Table M.34</b> Example Temperature Loading	40
<b>Table M.35</b> Job Data File: Input Field for Loading Data (Dynamic Loads) (Output at Each Time Stage)	41
<b>Table M.36</b> Job Data File: Input Field for Loading Data (Dynamic Loads) (Output at Selected Time Stage Intervals)	41
<b>Table M.37</b> Job Data File: Input Field for Analysis Parameters	42
<b>Table M.38</b> Job Data File: Input Field for Material Behaviour Models	45
<b>Table M.39</b> Auxiliary Data File: Input Field for General Analysis Parameters	47
<b>Table M.40</b> Auxiliary Data File: Input Field for Dynamic Analysis Parameters	49
<b>Table M.41</b> Output File: General Analysis Parameters and Loading Condition	51
<b>Table M.42</b> Output File: Convergence Factors	52
<b>Table M.43</b> Output File: Compatibility Forces	53
<b>Table M.44</b> Output File: Member End Forces	53
<b>Table M.45</b> Output File: Support Reactions	54
<b>Table M.46</b> Output File: Nodal Displacements	54
<b>Table M.47</b> Output File: Member Deformations	55
<b>Table M.48</b> Output File: Member Section Average Values	55
<b>Table M.49</b> Output File: Member Section Crack Conditions	56
<b>Table M.50</b> Output File: Member Section Net Strains	56
<b>Table M.51</b> Output File: Member Section Concrete Stresses	57
<b>Table M.52</b> Output File: Member Section Long. Reinforcement Strains and Stresses	58

## LIST OF FIGURES

---

<b>Figure M.1</b>	A Screen Shot during the VecTor5 Analysis	5
<b>Figure M.2</b>	Schematic Representation of Analysis Process of VecTor5	6
<b>Figure M.3</b>	Orientation of Frame Members: <b>(a)</b> Horizontal Member; <b>(b)</b> Vertical Member <b>(c)</b> Member Cross Section; <b>(d)</b> Global Coordinate System	10
<b>Figure M.4</b>	Structure Data File: Member Reference Types	15
<b>Figure M.5</b>	Beam VS-A1: <b>(a)</b> Cross Section Details; <b>(b)</b> Sectional Model	16
<b>Figure M.6</b>	<b>(a)</b> A Member with Nodal Loads; <b>(b)</b> Global Coordinate system	20
<b>Figure M.7</b>	Part of a Structural Model with Nodal Loads	21
<b>Figure M.8</b>	Member M with End Actions in the Member-Oriented Coord. System	22
<b>Figure M.9</b>	A Member with Concentrated Load in the Global Coordinate System	23
<b>Figure M.10</b>	A Member with UDL in Member-Oriented Coordinate System	24
<b>Figure M.11</b>	<b>(a)</b> A Member with Automatically Calculated Gravity Loads; <b>(b)</b> Degrees-of-Freedom for Gravity Loads	25
<b>Figure M.12</b>	<b>(a)</b> Member M in Reference (Ambient) Temperature of $T$ °C; <b>(b)</b> Input Parameters for Temperature Loading	27
<b>Figure M.13</b>	<b>(a)</b> A Member with Prescribed Nodal Displacements; <b>(b)</b> Degrees-of-Freedom for Prescribed Nodal Displacements	28
<b>Figure M.14</b>	<b>(a)</b> A Member with Additional Lumped Masses, Initial Velocity and Const. Acceleration Loading; <b>(b)</b> DOF for Additional Lumped Masses	30
<b>Figure M.15</b>	Multi-Linear Force-Time History	32
<b>Figure M.16</b>	Multi-Linear Force-Time History with Automatically Added Branches	32
<b>Figure M.17</b>	Suddenly Appearing Multi-Linear Force-Time History	33
<b>Figure M.18</b>	Ground Acceleration-Time History Loading (Accelerogram)	34
<b>Figure M.19</b>	VecTor.EQR Data File for Ground Acceleration Loads	35
<b>Figure M.20</b>	Monotonic Loading Condition	38
<b>Figure M.21</b>	Cyclic Loading Condition	38
<b>Figure M.22</b>	Reversed-Cyclic Loading Condition	39
<b>Figure M.23</b>	Concrete Post-Peak Response: Modified Park-Kent (1982)	62

---

## CHAPTER M1

---

### INTRODUCTION

VecTor5 is a nonlinear sectional analysis program for two dimensional frame-related structures consisting of beams, columns and shear walls, subjected to temperature, static and dynamic loading conditions. 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 Modified Compression Field Theory (Vecchio and Collins, 1986) and the Disturbed Stress Field Model (Vecchio, 2000), 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. Using a direct stiffness method, VecTor5 performs a global frame analysis first. Rigorous sectional analyses of concrete member cross sections are then performed at various sections along the lengths of the members, using a distributed nonlinearity fibre model approach. The computed responses are enforced with the use of an unbalanced force approach where the unbalanced forces are reduced to zero iteratively.

VecTor5 is capable of considering such second order effects as 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. Required for the dynamic analyses, VecTor5 considers strain rate effects on the concrete and reinforcement, and damping effects on the structure.

The basic version of VecTor5 can be found in the software section of the VecTor Analysis Group website at '[www.civ.utoronto.ca/vector](http://www.civ.utoronto.ca/vector)'.

---



## CHAPTER M2

---

### ANALYSIS WITH VEC TOR5

This chapter describes the analysis process with VecTor5 by introducing the input and output data files and by outlining an example analysis with VecTor5. The chapter concludes with a flowchart depicting the analysis process of VecTor5.

#### M2.1 Input Data Files

To perform an analysis with VecTor5, at least four input text files are required; namely, the *Structure Data File*, *Job Data File*, *Load Data File(s)* and *Auxiliary Data File*. Provided together with the program VT5.exe, all input files have the fill-in-the-blanks format and can easily be modified using a standard text editor such as Microsoft Windows Notepad.

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.

Having the extension L5R, the *Load Data File* contains information regarding nodal loads, externally applied member end actions, concentrated loads, uniformly distributed loads, gravity loads, temperature loads, concrete prestains, prescribed nodal displacements, additional lumped masses, impulse forces and ground accelerations.

The *Job Data File* (*VecTor.JOB*) contains information regarding the loading data, analysis parameters, and material behaviour models. In the loading data, the total number of load or time stages, load, time, displacement and temperature increments, and the loading types such monotonic, cyclic or reversed-cyclic are specified. The type of the analysis (i.e., static or dynamic) is defined in the analysis parameters. There is no need to modify the analysis parameters and the material behaviour models to run the program in the default mode.

The *Auxiliary Data File (VT5.AUX)* contains two sets of information regarding the analysis parameters. The first set includes the general analysis parameters defining the sectional analysis mode, shear analysis mode, shear protection, dynamic averaging factor, concrete aggregate type and reference temperature. The second set specifies the dynamic analysis parameters consisting of the time integration method, the selection of two modes and the corresponding damping ratios in the case of using additional viscous damping, the ground acceleration factor in the x- and y-directions, and the mass factor due to self-weight. There is no need to modify this file to run the program in the default mode.

## **M2.2 Expanded Data Files**

At the beginning of the analysis, VecTor5 produces two expanded data files at a minimum. In these files, the input values are printed out as read by VecTor5; these values are used throughout the analysis. Moreover, additional values calculated by VecTor5 are printed out in the expanded data files. Therefore, it is recommended to check these files at the very beginning of the analysis for input errors and other inconsistencies.

Having the extension S5E, the *Expanded Structure Data File*, includes an organized list of values from the input *Structure Data File* (see Table M.66). In addition, it includes the concrete properties as calculated by VecTor5 for the selected concrete base curve. Furthermore, the longitudinal reinforcement ratios and the average crack spacings of the concrete layers in the element-oriented axes are listed. The members determined automatically for the shear protection (see Table M.39) are also indicated in the member indices section of the expanded data file with SP=1.

Having the extension L5E, the *Expanded Load Data Files* include an organized list of loading details as read from the related *Load Data File* (see Table M.67). Moreover, nodal gravity loads and nodal masses due to the self-weight of the structure as automatically calculated by VecTor5 are printed out in the cases when they are requested to be included in the analysis by the user.

Having the extension E5E, the *Expanded Eigen Analysis Results Data File* include an organized list of all modal damping ratios if present, all modal periods and the two mode shapes as selected by the user (see Table M.68). Mass and stiffness proportionality coefficients as calculated by VecTor5, are also printed out in this file when using the Rayleigh damping option. This file is only produced when performing a dynamic analysis. Particularly useful for the dynamic analyses, the modal periods found in this file are useful when selecting an appropriate time step length for the analysis as discussed in Section 7.9 of Guner (2008).

### **M2.3 Performing an Analysis**

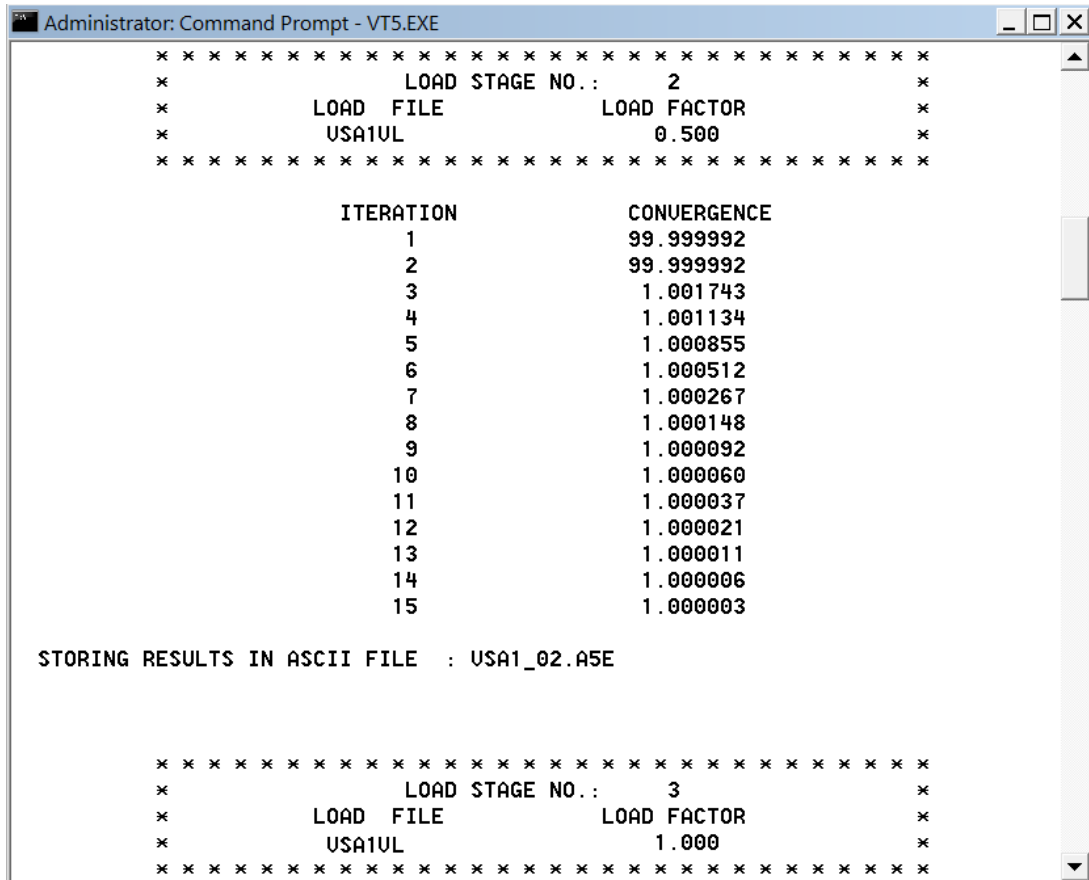
For illustrative purposes, consider the analysis of a simply supported beam VS-A1 subjected to one static load case. For this analysis, a *Structure Data File* named VSA1.S5R, a *Load Data File* named VSA1VL.L5R, a *Job Data File* named VecTor.JOB, an *Auxiliary Data File* named VT5.AUX and the executable program VT5.exe are needed. The names of the files with the S5R and L5R extensions can be defined as desired. The names of VecTor.JOB and VT5.AUX must not be changed. To resume a previous analysis, the related *Output Seed File*, with the extension of A5R, is also needed (see Table M.37). In the case of a dynamic time-history analysis, which is not defined in the *Load Data Files*, the *VecTor.EQR Data File* must also be provided as explained in Section M3.2 (see Figure M.19).

All of the files must be placed in the same folder of a personal computer. The analysis can be initiated by running the *VT5.exe* program. The program checks the input files before starting the analysis. In some cases, the input checking feature of the program might warn the user with a warning message or may terminate the analysis with an error message. There may be other input errors which are not detected by the program; therefore, it is recommended to check all input files carefully before running the analysis. The program produces at least two *Expanded Data Files* within the same folder. It is recommended to check these files, while the analysis is running, for input errors and for other inconsistencies.

## M2.4 Output Files

The program produces one *Output File* with the extension of A5E for each of the load or time stages being considered. In cases where the output of seed files are requested by the user, an *Output Seed File* with the extension of A5R is also produced. For example, for the analysis of Beam VS-A1, assume a load series ID of VSA1 is specified in the *Job Data File* as explained in Section M3.3 (see Table M.31). In this case, *Output Files* VSA1\_01.A5E, VSA1\_02.A5E and so on will be produced, where \_02 indicates the load or time stage, to which the *Output File* belongs.

During the analysis, a convergence factor is printed out on the computer screen at each global frame analysis iteration at each load stage as shown in Figure M.1. Convergence factors are useful to monitor the stability and the validity of the load or time stages. When the failure condition of the structure is reached, large convergence factors will cause the program to terminate, indicating the end of the analysis.



```
Administrator: Command Prompt - VT5.EXE
* * * * *
*          LOAD STAGE NO. :    2          *
*          LOAD FILE          LOAD FACTOR *
*          USA1VL             0.500      *
* * * * *

          ITERATION          CONVERGENCE
              1          99.999992
              2          99.999992
              3           1.001743
              4           1.001134
              5           1.000855
              6           1.000512
              7           1.000267
              8           1.000148
              9           1.000092
             10           1.000060
             11           1.000037
             12           1.000021
             13           1.000011
             14           1.000006
             15           1.000003

STORING RESULTS IN ASCII FILE : USA1_02.A5E

* * * * *
*          LOAD STAGE NO. :    3          *
*          LOAD FILE          LOAD FACTOR *
*          USA1VL             1.000      *
* * * * *
```

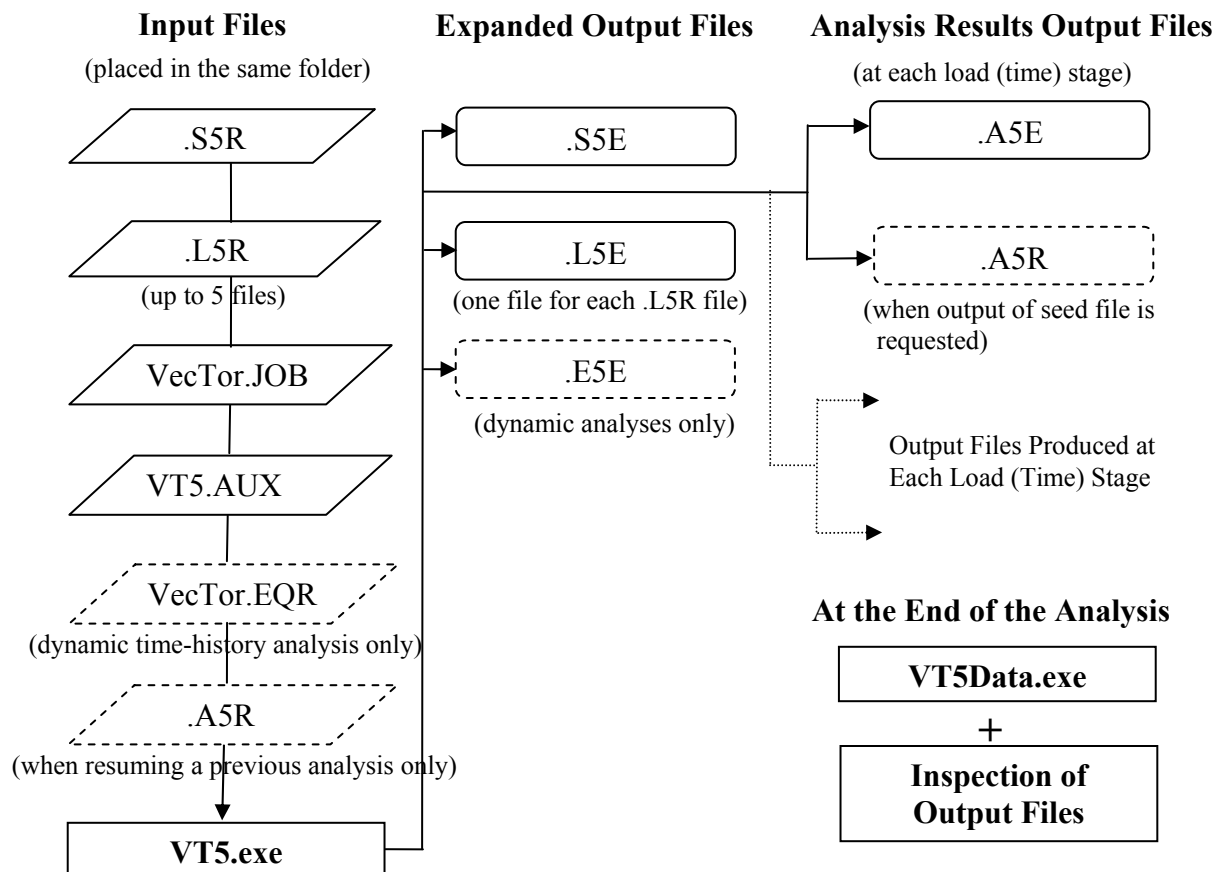
**Figure M.1** A Screen Shot during the VecTor5 Analysis

The load deflection data can be extracted by the provided post-processing program *VT5Data.exe*. It is required to enter the *Load Series ID* (VSA1 for the example analysis above), the *Reaction Node* and the *Displacement Node* when asked by the program. A data file *results.dat* will be added to the same folder, including the requested load-deflection data.

The *Output Files* of the load stages corresponding to the strength (peak load capacity) of the structure should be inspected to determine the damage or failure mode as explained in Section M4.2. The *Output Files* of the load stages corresponding to the serviceability limit state of the structure may also be inspected to determine such parameters as crack widths, deflections, reinforcement and concrete stresses for comparisons with the allowable limits.

### M2.5 Analysis Process of VecTor5

The analysis process with VecTor5 is schematically presented in Figure M.2.



**Figure M.2** Schematic Representation of Analysis Process of VecTor5

## CHAPTER M3

---

### VECTOR5 INPUT FILES

This chapter describes the preparation of input files for an analysis with VecTor5. The characters and numbers shown with regular fonts are already present in the input files; they should not be modified. The input fields which must be filled in by the user are indicated with bold numbers and characters. In several places in this chapter, advanced users are referred to Guner (2008) for more detailed information. For general applications by beginner- and intermediate-level users, this manual should suffice on its own.

#### M3.1 Structure Data File (.S5R)

For demonstrative purposes, the structural model of Beam VS-A1 is considered in the following section. The details of this beam are presented in Section 4.5 of Guner (2008). The *Structure Data File* starts with the input field for the *General Parameters* as shown in Table M.1.

**Table M.1** Structure Data File: Input Field for General Parameters

GENERAL PARAMETERS	
=====	
Structure Title (30 char. max.)	: <b>VSA1</b>
Structure File Name (8 char. max.)	: <b>VSA1</b>
No. of Members	: <b>6</b>
No. of Member Types	: <b>2</b>
No. of Nodes	: <b>7</b>
No. of Support Nodes	: <b>2</b>
No. of Support Restraints	: <b>3</b>

In Table M.1, the *Structure Title* is intended for descriptive purposes; the *Structure File name* must be the same as the name of the .S5R file saved in a particular folder of a personal computer. *No. of Members* and *No. of Nodes* are the total numbers used in the structural model. *No. of Member Types* is the total number of cross sections used in the model. The properties of each cross section (member type) are defined in following sections of the *Structure Data File*. *No. of support nodes* is the total number of nodes that

include at least one restrained degree-of-freedom; *No of Support Restraints* is the total number of restrained degrees-of-freedom.

The *Structure Data File* continues with the *Structure Parameters* input field beginning with the *Nodal Coordinates* input as shown in Table M.2, where *NODE* is the node number, *X* and *Y* are the coordinates of the node in the global coordinate system. Refer to Figure 4.7 of Guner (2008) for the graphical representation of this model.

**Table M.2** Structure Data File: Input Field for Nodal Coordinates (Regular Input)

STRUCTURE PARAMETERS					
=====					
(A) Nodal Coordinates					
-----					
NODE	X	Y	[ #NODES	d(NODE)	d(X) d(Y) ] ← up to 2 directions /
<b>1</b>	<b>0</b>	<b>0/</b>			
<b>2</b>	<b>305</b>	<b>0/</b>			
<b>3</b>	<b>610</b>	<b>0/</b>			
<b>4</b>	<b>915</b>	<b>0/</b>			
<b>5</b>	<b>1220</b>	<b>0/</b>			
<b>6</b>	<b>1525</b>	<b>0/</b>			
<b>7</b>	<b>1830</b>	<b>0/</b>			
/					

To define a number of nodes that follow a certain incremental pattern, the input field shown in brackets [ ] in Table M.2 can be filled in, where *#NODES* are the total number of nodes, *d(NODE)* is the increment in the node number, *d(X)* and *d(Y)* are the increments in the *X* and *Y* coordinates respectively, which can be positive or negative. Using this type of representation the same input shown above can be represented as given in Table M.3.

**Table M.3** Structure Data File: Input Field for Nodal Coordinates (Compact Input)

(A) Nodal Coordinates					
-----					
NODE	X	Y	[ #NODES	d(NODE)	d(X) d(Y) ] ← up to 2 directions /
<b>1</b>	<b>0</b>	<b>0</b>	<b>7</b>	<b>1</b>	<b>305 0/</b>
/					

In Table M.3, Node 1 is assumed to be located at (0, 0) mm and a total of seven nodes (*#NODES*) are spaced from Node 1 at 305 mm in the global x-direction (*d(X)*) and at 0 mm in the global y-direction (*d(Y)*), increasing in node number by 1 (*d(NODE)*).

It is also possible to use a combination of the two representations as shown in Table M.4.

**Table M.4** Structure Data File: Input Field for Nodal Coordinates (Mixed Input)

(A) Nodal Coordinates  
-----

NODE	X	Y	[ #NODES	d(NODE)	d(X)	d(Y)	] ← up to 2 directions /
<b>1</b>	<b>0</b>	<b>0</b>	<b>5</b>	<b>1</b>	<b>305</b>	<b>0</b>	/
<b>6</b>	<b>1525</b>	<b>0</b>					
<b>7</b>	<b>1830</b>	<b>0</b>					
/							

*Structure Parameters* continue with the *Member Indices*, which specify the two node numbers associated with each member as shown in Table M.5, where *MEM* is the member number, *NODE1* and *NODE2* are the two end node number of the member, *MEM. TYPE* is the member type (cross section type) of the member. Refer to Figure 4.7 of Guner (2008) for the graphical representation of this model.

**Table M.5** Structure Data File: Input Field for Member Indices (Regular Input)

(B) Member Indices  
-----

MEM	NODE1	NODE2	MEM.TYPE	[ #MEMS	d(MEM)	d(NODE)	] ← up to 2 dir./
<b>1</b>	<b>1</b>	<b>2</b>	<b>1</b>	/			
<b>2</b>	<b>2</b>	<b>3</b>	<b>1</b>	/			
<b>3</b>	<b>3</b>	<b>4</b>	<b>1</b>	/			
<b>4</b>	<b>4</b>	<b>5</b>	<b>1</b>	/			
<b>5</b>	<b>5</b>	<b>6</b>	<b>1</b>	/			
<b>6</b>	<b>6</b>	<b>7</b>	<b>1</b>	/			
/							

To define a number of members following a certain incremental pattern, the input field shown in brackets [ ] in Table M.5 can be filled in, where *#MEMS* are the total number of members, *d(MEM)* is the increment in the member number, and *d(NODE)* is the increment in the end node numbers of the members. Using this type of representation the same input shown above can be represented as given in Table M.6.

**Table M.6** Structure Data File: Input Field for Member Indices (Compact Input)

(B) Member Indices  
-----

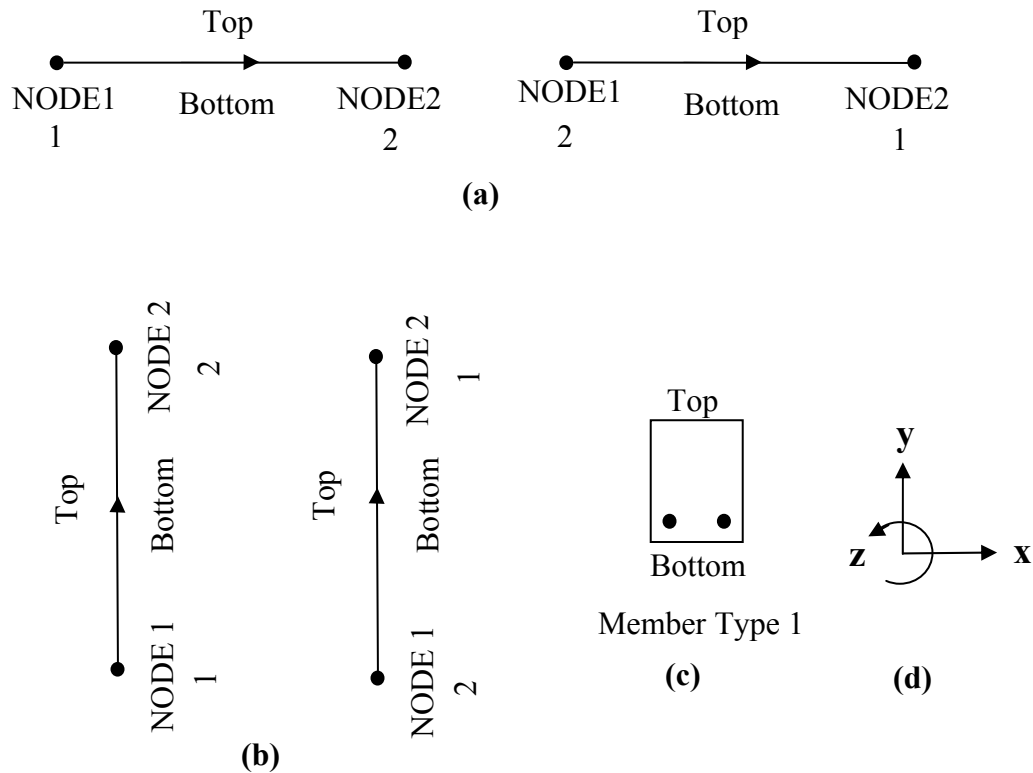
MEM	NODE1	NODE2	MEM.TYPE	[ #MEMS	d(MEM)	d(NODE)	] ← up to 2 dir./
<b>1</b>	<b>1</b>	<b>2</b>	<b>1</b>	<b>6</b>	<b>1</b>	<b>1</b>	/
/							



In Table M.6, it is specified that *Member 1* is between *Nodes 1* and *2* and has a *Member Type* (cross section type) of *1*. Starting from *Member 1*, a total number of six members are specified, increasing in member number by 1 and end node numbers by 1.

Similar to the representation in Table M.4, it is possible to use a combination of the two input methods together for the member indices. Note that *Member Type 2* is not used in the structural model of Beam VS-A1; that is, all members have a *Member Type* of 1. *Member Type 2* is defined for demonstrative purposes.

When using non-symmetrical cross sections, caution must be exercised to define the orientation of the members correctly, which is determined by the NODE1 and NODE2 numbers. For example, consider the orientation of two members in the global x- and y- directions as shown in Figure M.3(a) and Figure M.3(b). The top of the cross section is always determined assuming an element orientation from the first node number (NODE1) to the second node number (NODE2).



**Figure M.3** Orientation of Frame Members: **(a)** Horizontal Member; **(b)** Vertical Member; **(c)** Member Cross Section; **(d)** Global Coordinate System

*Structure Parameters* continue with the *Support Restraint List*, which specifies the restrained nodes as shown in Table M.7.

**Table M.7** Structure Data File: Input Field for Support Restraint List (Regular Input)

(C) Support Restraint List  
-----

NODE	X-RST	Y-RST	Z-RST [ #NODES d(NODE) ] <-- up to 2 directions /
<b>1</b>	<b>0</b>	<b>1</b>	<b>0 /</b>
<b>7</b>	<b>1</b>	<b>0</b>	<b>1 /</b>
<b>/</b>			

In Table M.7, *X-RST* and *Y-RST* correspond to the translational degrees-of-freedom; *Z-RST* corresponds to the rotational degree-of-freedom about the z-axis which is normal to the plane of the structure as shown in Figure M.3(d). Refer to Figure 4.7 of Guner (2008) for the graphical representation of this model.

*Structure Parameters* continue with the *Member Specifications*, which specify the cross section properties of each member type (i.e., cross section) used as shown in Table M.8.

**Table M.8** Structure Data File: Input Field for Member Specifications

(D) Member Specifications  
-----

MT	fc [ ft	Ec	e0	Mu	Cc	Kc	Agg	Dens]	[Smx	Smy]
	(MPa)	(MPa)	(me)		(/deg.C)	(mm <sup>2</sup> /hr)	(mm)	(kg/m <sup>3</sup> )	(mm)	(mm)
<b>1</b>	<b>22.60</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>
<b>2</b>	<b>33.20</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>
<b>/</b>										

MT	Nc	Ns	Fyz	St	Dbt	Fyt	Fut	Est	Esht	esht	[ Cs ]	Ref.
	(#)	(#)	(MPa)	(mm)	(mm)	(MPa)	(MPa)	(MPa)	(MPa)	(me)	(/deg.C)	Type
<b>1</b>	<b>34</b>	<b>3</b>	<b>600</b>	<b>210</b>	<b>6.4</b>	<b>600</b>	<b>649</b>	<b>200000</b>	<b>1530</b>	<b>3.0</b>	<b>0</b>	<b>1</b>
<b>2</b>	<b>34</b>	<b>2</b>	<b>500</b>	<b>110</b>	<b>6.4</b>	<b>500</b>	<b>600</b>	<b>200000</b>	<b>1000</b>	<b>8.0</b>	<b>0</b>	<b>1</b>
<b>/</b>												

For each member type used, the parameters in Table M.8 must be defined. *MT* is the member type or the cross section type, *f'c* is the concrete compressive strength, *f't* is the concrete tensile strength, *Ec* is the modulus of elasticity of concrete, *e0* is the strain corresponding to the peak stress of concrete, *Mu* is the Poisson's ratio, *Cc* is the

coefficient of thermal expansion of concrete,  $Kc$  is the thermal diffusivity of concrete,  $Agg$  is the maximum size of aggregate used in the concrete mixture,  $Dens$  is the density of concrete,  $Smx$  and  $Smy$  are the crack spacing in the element-oriented x- and y-directions respectively as shown in Figure 3.18 and defined in Section 3.7.9 of Guner (2008),  $Nc$  is the total number of concrete layers used,  $Ns$  is the total number of reinforcing and prestressing steel layers used,  $Fyz$  is the yield stress of the out-of-plane reinforcement,  $St$  is the spacing of the transverse reinforcement along the longitudinal direction of the member,  $Dbt$  is the diameter of the transverse reinforcement,  $Fyt$  and  $Fut$  are the yield and ultimate stresses of the transverse reinforcement respectively,  $Est$  is the modulus of elasticity of the transverse reinforcement,  $esht$  is the strain where strain hardening of the transverse reinforcement begins as defined in Section 3.3.3.4 of Guner (2008), and  $Cs$  is the coefficient of thermal expansion of both longitudinal and transverse reinforcement associated with this member type.

In Table M.8, the parameters in the brackets [ ] can be input as zero indicating that the default values are to be calculated by VecTor5 and assumed for the input. The default values are defined in Eq. M.1 to Eq. M.11.

$$f_t = 0.33 \times \sqrt{f'_c} \quad (M.1)$$

$$E_c = 5000 \times \sqrt{f'_c} \quad (M.2)$$

$$e_0 = (1.8 \times 0.0075 \times f'_c) \times 10^{-3} \quad (M.3)$$

It is important to input the appropriate concrete properties based on the selected concrete compression base curve. For concrete strengths up to 40 MPa (i.e.,  $f'_c \leq 40$  MPa), the use of the default Hognestad (parabola) model is recommended (see Table M.38). In this case, the program will use  $e_0$  value to calculate the modulus of elasticity of the concrete as follows:

$$E_c = 2 \times \frac{f'_c}{e_0} \quad (\text{M.4})$$

In other words, when using the Hognestad (parabola) option, the input  $E_c$  value is ignored; the  $E_c$  value is calculated through the use of Eq. M.4. In cases where the  $E_c$  value is desired to be input, a corresponding  $e_0$  must be calculated through Eq. M.4 and supplied to the program. It is always possible to check the calculated properties through the *Expanded Structure Data File*, which has the extension of S5E (see Table M.66).

In the cases where both  $E_c$  and  $e_0$  values are desired to be input (i.e., known), the Popovics – NSC (1973) option must be used for concrete strengths up to 50 MPa. In this option, the input  $f'_c$ , the  $E_c$  and  $e_0$  values are used to create the concrete compression base curve.

Similarly, for the high strength concrete with strengths greater than 50 MPa, two options are available. When only the  $e_0$  value is desired to be input (i.e., known), Popovics – HSC model should be used. In this case, the  $E_c$  value is calculated as follows:

$$n = 0.80 + \frac{f'_c}{17} \quad (\text{M.5})$$

$$E_c = \frac{f_c}{e_0} \times \frac{n}{n-1} \quad (\text{M.6})$$

In the cases where both the  $E_c$  and  $e_0$  values are desired to be input, the Hoshikuma – HSC (Hoshikuma et al., 1997) model should be used. In this model,  $f'_c$ ,  $E_c$  and  $e_0$  are used to create the concrete compression base curve.

Note that concrete strength  $f'_c$  must be input in all cases; there is no default value for  $f'_c$ . Default values for the other concrete properties are as follows:

$$\mu_u = 0.15 \quad (\text{M.7})$$

$$C_c = 10 \times 10^{-6} \quad 1 / ^\circ \text{C} \quad (\text{M.8})$$

$$K_c = 4320 \text{ mm}^2 / \text{hr} \quad (\text{M.9})$$

$$\text{Agg} = 10 \text{ mm} \quad (\text{M.10})$$

$$\text{Dens} = 2400 \text{ kg} / \text{m}^3 \quad (\text{M.11})$$

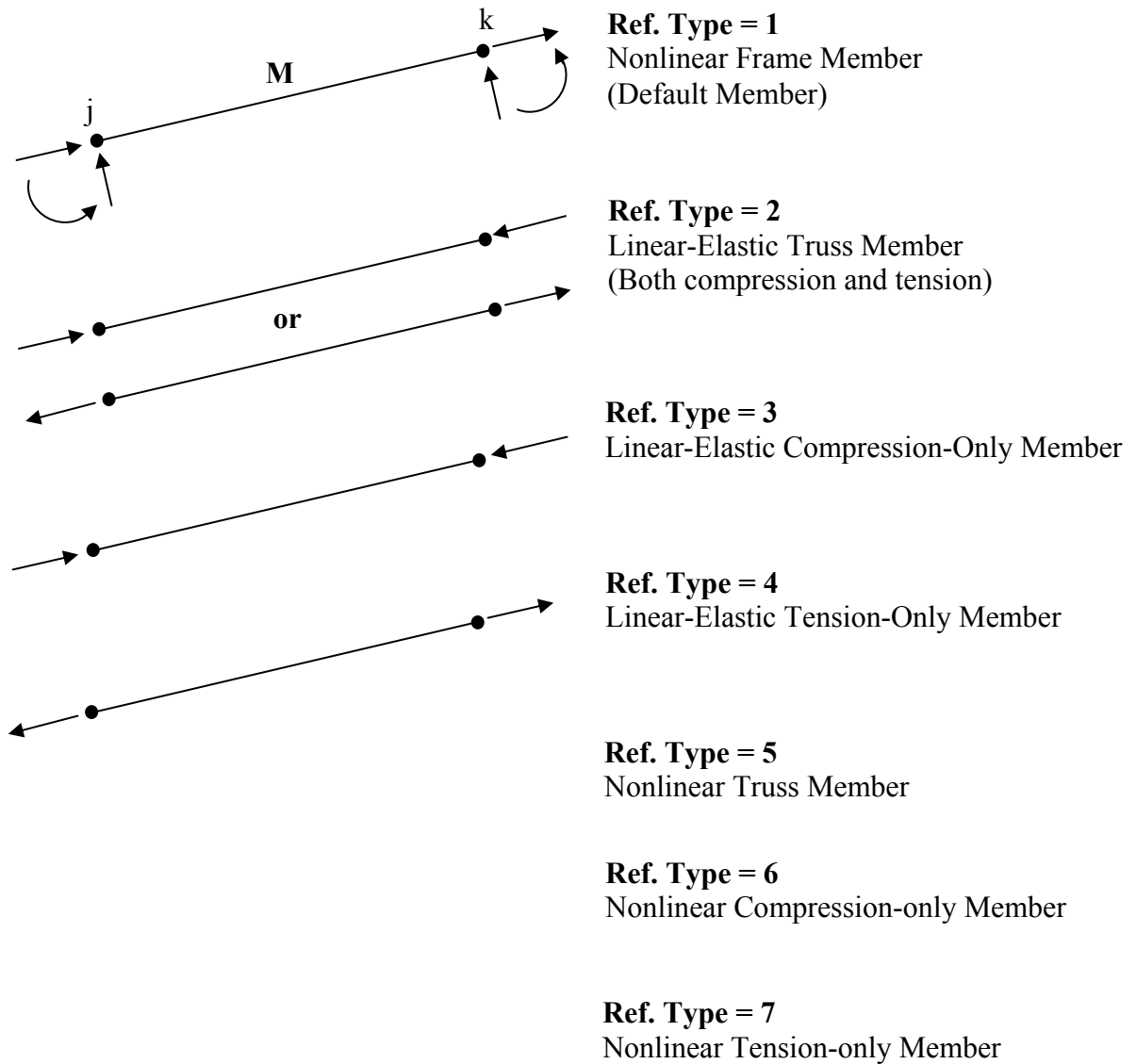
It is recommended to supply 0 and 0 for  $S_{mx}$  and  $S_{my}$  for the automatic calculation of the average crack spacing, as explained in Section 3.7.9 of Guner (2008). The automatically calculated crack spacing values for each concrete layer of each member type can be viewed in the *Expanded Structure Data File* (See Table M.66). It is also possible to input the desired average crack spacings if the values are known.

For the reinforcement, the only default value available is the coefficient of thermal expansion,  $C_s$ , which taken as  $11.5 \times 10^{-6} / ^\circ \text{C}$  if it is input as 0. All other values must be input explicitly by the user.

If an input value of 0 is desired to be used, a small value (e.g., 0.001) can be input to prevent the program from using the default value. For example, it is recommended by Lubell et al. (2004) to take the maximum aggregate size as zero for concrete strengths in excess of 70 MPa. In such a case,  $\text{Agg} = 0.001 \text{ mm}$  can be input.

*Ref. Type* is the member reference type, which specifies the member behaviour and nodal degrees-of-freedom. There are currently seven available member reference types as shown in Figure M.4, one of which should be selected.

*Structure Parameters* continue with the *concrete layers* input field, which specifies the geometry and the smeared transverse and s out-of-plane reinforcement ratios of each member type used as shown in Table M.9.



**Figure M.4** Structure Data File: Member Reference Types

**Table M.9** Structure Data File: Input Field for Concrete Layers

(E) Concrete Layers

---

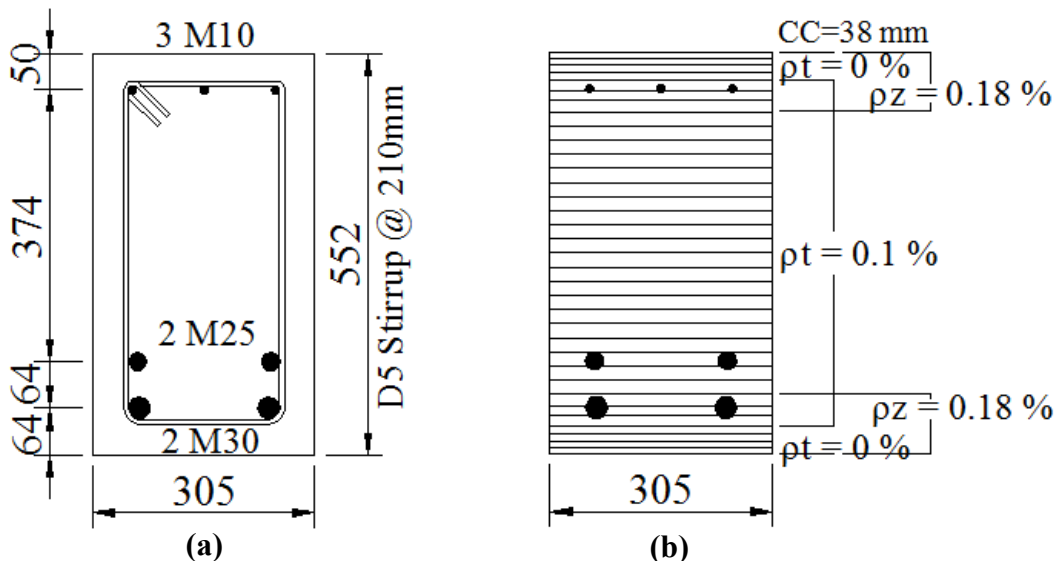
MT	Dc (mm)	Wc (mm)	Rho-t (%)	Rho-z (%)	Nx
1	8	305	0	0.18	1
1	10	305	0	0.18	3
1	14	305	0.1	0.18	2
1	16	305	0.1	0.18	1
1	19.4	305	0.1	0	20

**Table M.9** Structure Data File: Input Field for Concrete Layers (Continued)

1	16	305	0.1	0.18	1
1	14	305	0.1	0.18	2
1	10	305	0	0.18	3
1	8	305	0	0.18	1
2	8	305	0	0.22	1
2	10	305	0	0.22	3
2	14	305	0.2	0.22	2
2	16	305	0.2	0.22	1
2	19.4	305	0.2	0	20
2	16	305	0.2	0.22	1
2	14	305	0.2	0.22	2
2	10	305	0	0.22	3
2	8	305	0	0.22	1
/					

In Table M.9,  $MT$  is the member type or the cross section type,  $D_c$  is the thickness of the concrete layer,  $W_c$  is the width of the cross section,  $Rho-t$  and  $Rho-z$  are the ratios of the transverse and the out-of-plane reinforcement present in the corresponding concrete layer, and  $N_x$  is the number of concrete layers with identical details. Note that concrete layers input starts from the top of the cross section as defined in Figure M.3.

The cross section and the sectional model of *Member Type 1* of Beam VS-A1 are presented in Figure M.5. The calculation of the transverse and out-of-plane reinforcement ratios and the selection of the concrete layer thicknesses are discussed in Section 4.5.2 of Guner (2008).



**Figure M.5** Beam VS-A1: (a) Cross Section Details; (b) Sectional Model

*Structure Parameters* continue with the specification of reinforcing and prestressing *Steel Layers* for each member type (cross section) used as shown in Table M.10.

**Table M.10** Structure Data File: Input Field for Steel Layers

(F) Steel Layers  
-----

MT	N	Ys (mm)	As (mm <sup>2</sup> )	Db (mm)	Fy (MPa)	Fu (MPa)	Es (MPa)	Esh (MPa)	esh (me)	Dep (me)
1	1	50.00	300.0	11.30	315.0	460.0	200000	730	7.70	0.000
1	2	424.00	1000.0	25.20	445.0	680.0	220000	1130	8.50	0.000
1	3	488.00	1400.0	29.90	436.0	700.0	200000	1610	11.40	0.000
2	1	50.00	300.0	11.30	315.0	460.0	200000	730	7.70	0.000
2	2	450.00	1400.0	29.90	436.0	700.0	200000	1610	11.40	0.000
/										

In Table M.10, *MT* is the member type, *N* is the longitudinal reinforcement (i.e., reinforcing or prestressing steel layer) component number starting from 1 and increasing in number by 1, *Ys* is the location of the longitudinal reinforcement layer from the top of the cross section, *As* is the total area of the longitudinal reinforcement layer, *Db* is the diameter of one bar, *Fy* and *Fu* are the yield and ultimate stresses of the longitudinal reinforcement layer respectively, *Es* is the modulus of elasticity of the longitudinal reinforcement, *esh* is the strain where the strain hardening of the longitudinal reinforcement begins as defined in Section 3.3.3.4 of Guner (2008), and *DEP* is the locked-in strain differential if the layer is a prestressed steel layer.

As discussed in Section 4.5.2 of Guner (2008), the location of the longitudinal reinforcement layers, *Ys*, can be defined independently from the concrete layer configuration.

*Structure Parameters* conclude with the *Detailed Member Output list* as shown in Table M.11. As defined in Section M4.2, detailed conditions of the concrete and reinforcement layers are printed out at each load (time) step for the members specified in the *Detailed Member Output List*.



Before running the analysis, a number of members which are expected to be critical (e.g., the ends of beams or columns in the case of lateral loads and the midspans in the case of static loads) may be specified for the detailed output. After the analysis, if any other member turns out to be critical, the *Detailed Member Output List* can be updated and the analysis can be repeated.

**Table M.11** Structure Data File: Input Field for Detailed Member Output List (Regular Input)

(G) Detailed Member Output List  
-----  
MEM [ #MEMS d(MEM) ] ← up to 2 directions /  
2/  
5/  
6/  
/

In Table M.11, *MEM* is the member number. To define a number of members following a certain incremental pattern, the input field shown in brackets [ ] can be filled in, where *#MEMS* are the total number of members, and *d(MEM)* is the increment in the member number. Using this type of representation, the same input can be created as given in Table M.12.

**Table M.12** Structure Data File: Input Field for Detailed Member Output List (Mixed Input)

(G) Detailed Member Output List  
-----  
MEM [ #MEMS d(MEM) ] ← up to 2 directions /  
2            2            3/  
6/  
/

*Detailed Member Output* list is useful when evaluating the damage or failure mode of the structure as discussed in Section M4.2. It can also be used, in the serviceability limit state, to determine if the crack widths, reinforcement stresses and concrete stresses are within the allowable limits.

### M3.2 Load Data Files (.S5L)

Similar to the *Structure Data File*, the *Load Data File* has fill-in-the-blanks format and can be modified by a standard text editor such as Microsoft Windows Notepad. The *Load Data File* starts with the *Load Case Parameters*, which specifies the loading conditions to be considered in this load case.

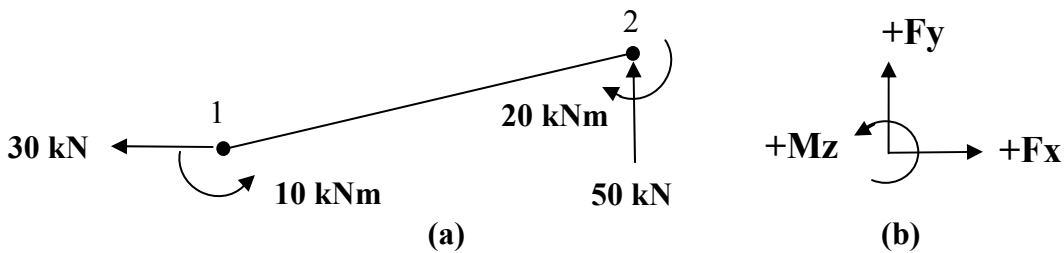
**Table M.13** Load Data File: Input Field for Load Case Parameters

LOAD CASE PARAMETERS		
*****		
Load Case ID	(15 char. max.)	: <b>VSA1 Vertical</b>
Load Case Data File	(8 char. max.)	: <b>VSA1VL</b>
Load Factored	(0 - 1)	: <b>1</b>
Time Factored	(0 - 1)	: <b>0</b>
No. of Loaded Nodes		: <b>0</b>
No. of Members w/ End Action Loads		: <b>0</b>
No. of Members w/ Concentrated Loads		: <b>0</b>
No. of Members w/ Distributed Loads		: <b>0</b>
No. of Members w/ Gravity Loads		: <b>0</b>
No. of Members w/ Temperature Loads		: <b>0</b>
No. of Members w/ Concrete Prestrain		: <b>0</b>
No. of Members w/ Support Displacements		: <b>1</b>
No. of Nodes w/ Lumped Mass Assignments		: <b>0</b>
No. of Nodes w/ Impulse Loads		: <b>0</b>
No. of Ground Acceleration Data		: <b>0</b>

In Table M.13, *Load Case ID* is for information purposes; that is, a name that describes the load case can be specified. The *Load Case Data File* is the name of the load case which must be referred to in the *Job Data File*, explained in Section M3.3 (see Table M.30). The *Load case data file* must have the same name as the name of the .L5R file saved in a particular folder of a personal computer. For analyses including mechanical and dynamic loads, the load must be factored (i.e., *Load Factored* : 1). The *Time Factored* option (i.e., *Time Factored* : 1) is only applicable when performing a thermal analysis, which is explained in Section M3.3 (see Table M.34). The following input fields in Table M.13 request the input of the total number of nodes with assigned nodal point

loads, the total number of members with assigned loading and the total number of ground acceleration data.

The *Load Case Data File* continues with the *Nodal Loads* input field. For any node subjected to mechanical forces, this input field of the *Load Data File* must be filled in the global x- and y-coordinate system. For illustrative purposes, assume that the external loads shown in Figure M.6 are acting on a particular part of a structural model, say *Node 1* and *Node 2*. The corresponding input to the nodal loads field must be as shown in Table M.14.



**Figure M.6 (a)** A Member with Nodal Loads; **(b)** Global Coordinate System

**Table M.14** Load Data File: Input Field for Nodal Loads (Regular Input)

```

NODAL LOADS
*****
<NOTE:> UNITS: kN, kN-m
NODE  Fx  Fy  Mz  [ #NODE  d(NODE)  d(Fx)  d(Fy)  d(Mz) ] ← (2)/
  1   -30  0   10/
  2     0  50  -20/
/

```

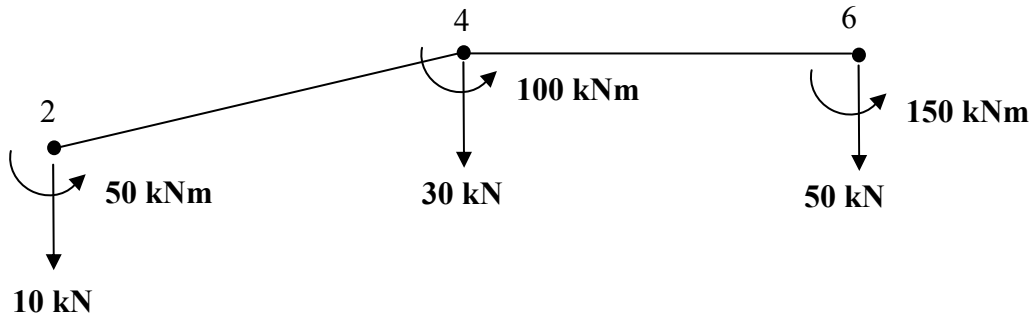
In Table M.14, the input field in the brackets [ ] may be used for specifying a number of nodal loads following a certain incremental pattern. *#NODE* is the total number of nodes on which the nodal loads are acting, *d(NODE)* is the increment in the node number, and *d(Fx)*, *d(Fy)* and *d(Mz)* are the increments in the x-, y- and z-component of the nodal load, respectively. For example, the nodal load on *Node 1* in Figure M.6 can be applied exactly on *Node 2* as shown in Table M.15.

**Table M.15** Load Data File: Input Field for Nodal Loads (Compact Input, Example 1)

```

NODAL LOADS
*****
<NOTE:> UNITS: kN,kN-m
NODE  Fx  Fy  Mz  [#NODE  d(NODE)  d(Fx)  d(Fy)  d(Mz) ] ← (2)/
  1   -30  0   10   2       1       0       0       0/
/
    
```

It is possible to change the magnitude of the components in such a representation. For example, for the loading in Figure M.7, the corresponding input becomes as shown in Table M.16.



**Figure M.7** Part of a Structural Model with Nodal Loads

**Table M.16** Load Data File: Input Field for Nodal Loads (Compact Input, Example 2)

```

NODAL LOADS
*****
<NOTE:> UNITS: kN,kN-m
NODE  Fx  Fy  Mz  [#NODE  d(NODE)  d(Fx)  d(Fy)  d(Mz) ] ← (2)/
  2    0  -10  50   3       2       0      -20   50/
/
    
```

It is also possible to use a combination of the two representations described above. For example, the loading in Figure M.7 can be input as shown in Table M.17.

**Table M.17** Load Data File: Input Field for Nodal Loads (Mixed Input)

```

NODAL LOADS
*****
<NOTE:> UNITS: kN,kN-m
NODE Fx  Fy  Mz  [#NODE  d(NODE)  d(Fx)  d(Fy)  d(Mz) ] ← (2)/
  2    0  -10  50    2      2      0    -20    50/
  6    0  -50  150/
/

```

The *Load Case Data File* continues with the externally applied *Member End Actions* input field as shown in Table M.18. For any member subjected to mechanical forces, this section of the *Load Data File* must be filled in. Contrary to the *Nodal Loads*, this type of load is defined relative to the member-oriented axes as shown in Figure M.8.

**Table M.18** Load Data File: Input Field for Member End Actions

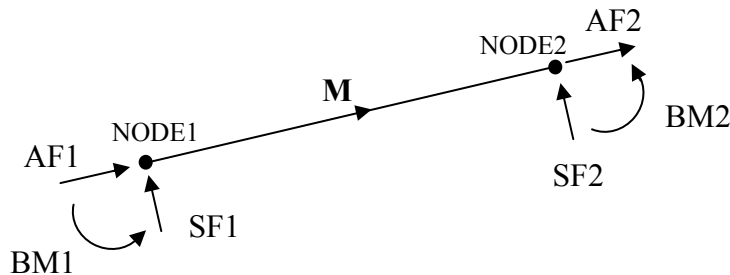
```

MEMBER END ACTIONS
*****
<NOTE:> UNITS: kN, kN-m
M AF1 SF1 BM1 AF2 SF2 BM2 [ #M  d(M) ] ← (2)/
/

```

In Table M.18,  $M$  is the member number,  $AF$  is the axial force,  $SF$  is the shear force and  $BM$  is the bending moment. Suffixes 1 and 2 refer to *NODE1* and *NODE2*, defined in Section M3.1 (see Figure M.3). Positive directions of the member end actions are presented in Figure M.8 for *Member M*.

In Table M.18, the input fields in the brackets [ ] may be used for specifying a number of members with the same end actions.  $\#M$  is the total number of nodes on which the same member end actions are acting, and  $d(M)$  is the increment in the member number.



**Figure M.8** Member M with End Actions in the Member-Oriented Coordinate System

The *Load Case Data File* continues with the *Concentrated Load* input field as shown in Table M.19. For any member subjected to mechanical forces acting within the two end nodes of the member, this section of the *Load Data File* must be filled in.

**Table M.19** Load Data File: Input Field for Concentrated Loads

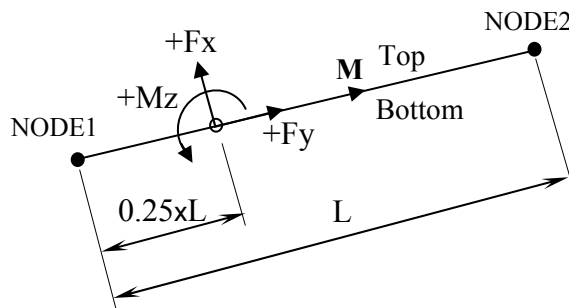
```

CONCENTRATED LOADS
*****
<NOTE:> UNITS: kN, kN-m,m
M Fx  Fy  Mz  x/L  [ #M  d(M)  d(Fx)  d(Fy)  d(Mz) ] ← (2)/
/

```

In Table M.19,  $M$  is the member number,  $x/L$  is the ratio of the distance from *NODE1* of the member to the concentrated load application point to the length of the members. For *Member 1* in Figure M.9,  $x/L$  must be entered as 0.25. The input field in the brackets [ ] may be used for specifying a number of members with the end actions following a certain incremental pattern.  $\#M$  is the total number of members on which the concentrated member end actions are acting,  $d(M)$  is the increment in the member number, and  $d(Fx)$ ,  $d(Fy)$  and  $d(Mz)$  are the increments in the concentrated loads which can be specified as either a positive or a negative quantity.

*Concentrated Loads* are defined relative to the member-oriented coordinate system. Positive  $Fx$  values act in a direction oriented from the bottom to the top of the member, positive  $Fy$  values act in the orientation of the member and positive moment values act in the counter-clockwise direction. As an example, *Member M* with concentrated loads acting in the positive directions is presented in Figure M.9.



**Figure M.9** A Member with Concentrated Load in the Global Coordinate System

The *Load Case Data File* continues with the *Uniformly Distributed Loads* input field as shown in Table M.20.

**Table M.20** Load Data File: Input Field for Uniformly Distributed Loads

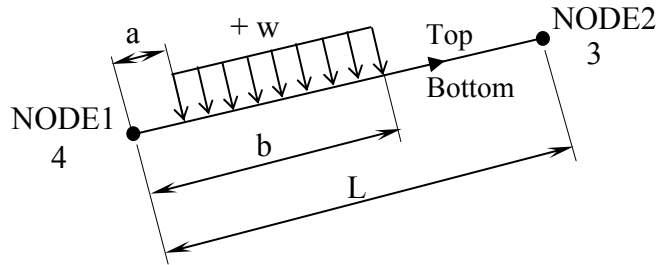
```

UNIFORMLY DISTRIBUTED LOADS
*****
<NOTE:> UNITS: kN/m,m
M w a/L b/L [ #M d(M) d(W) ] ← (2)
/

```

In Table M.20,  $M$  is the member on which the uniformly distributed load is acting, and  $w$  is the magnitude of the uniformly distributed load;  $a/L$  and  $b/L$  are depicted in Figure M.10, where  $a$  is the distance from *NODE1* of the member to the starting point of the uniformly distributed load and  $b$  is the distance from *NODE1* of the member to the ending point of the uniformly distributed load. For example, for a full length load,  $a/L=0.0$  and  $b/L=1.0$ ; for a half length load on the left-hand side  $a/L=0.0$  and  $b/L=0.5$ ; and for a half length load on the right hand-side  $a/L=0.5$  and  $b/L=1.0$  must be supplied.

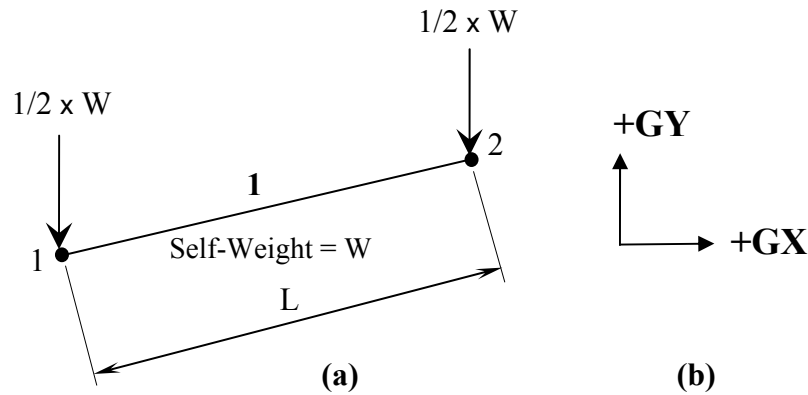
*Uniformly Distributed Loads* act in the member-oriented coordinate system with the positive values ( $+w$ ) acting in a direction oriented from the top to the bottom of the member, as defined in Figure M.3. For example, for the member shown in Figure M.10, the positive uniformly distributed load acts as shown in Figure M.10.



**Figure M.10** A Member with Uniformly Distributed Load in Member-Oriented Coordinate System

The *Load Case Data File* continues with the *Gravity Loads* input field. If the *Gravity Loads* are to be automatically included in the analysis, this part of the *Load Data File*

must be filled in. For the member in Figure M.11, the *Gravity Load* input must be as shown in Table M.21.



**Figure M.11** (a) A Member with Automatically Calculated Gravity Loads; (b) Degrees-of-Freedom for Gravity Loads

**Table M.21** Load Data File: Input Field for Gravity Loads (Example 1)

```

GRAVITY LOADS
*****
M  GX  GY [ #M  d(M) ] ← (2)/
1   0  -1/
/

```

In Table M.21, the input field in the brackets [ ] may be used for specifying a number of members whose self-weight is desired to be automatically considered. #M is the total number of members whose self-weight are to be included and d(M) is the increment in the member number. For example, for a structure model consisting of 100 members, the gravity load input must be as shown in Table M.22.

**Table M.22** Load Data File: Input Field for Gravity Loads (Example 2)

```

GRAVITY LOADS
*****
M  GX  GY [ #M  d(M) ] ← (2)/
1   0  -1  100  1/
/

```

Although *Gravity Loads* generally act in the negative Y direction as defined in Figure M.11(b), it is possible to consider other directions as well.



*Gravity Loads* are automatically calculated by VecTor5 through the use of Eq. M.12.

$$W = \rho \times A_g \times L \times g \quad (M.12)$$

where  $W$  is the total self-weight of the member in kN,  $\rho$  is the density of concrete,  $A_g$  is the gross cross sectional-area in mm<sup>2</sup>,  $h$  is the depth of the cross section in mm, and  $L$  is the length of the member in mm. Modifiable by the user, the default value of density for normal weight reinforced concrete was assumed to be 2400 kg/m<sup>3</sup>. Half of the total weight of the member is transferred to each end node.

The *Load Case Data File* continues with the *Temperature Loads* input field as shown in Table M.23. In the case of a thermal analysis, this part of the load case must be filled in.

**Table M.23** Load Data File: Input Field for Temperature Loads

```

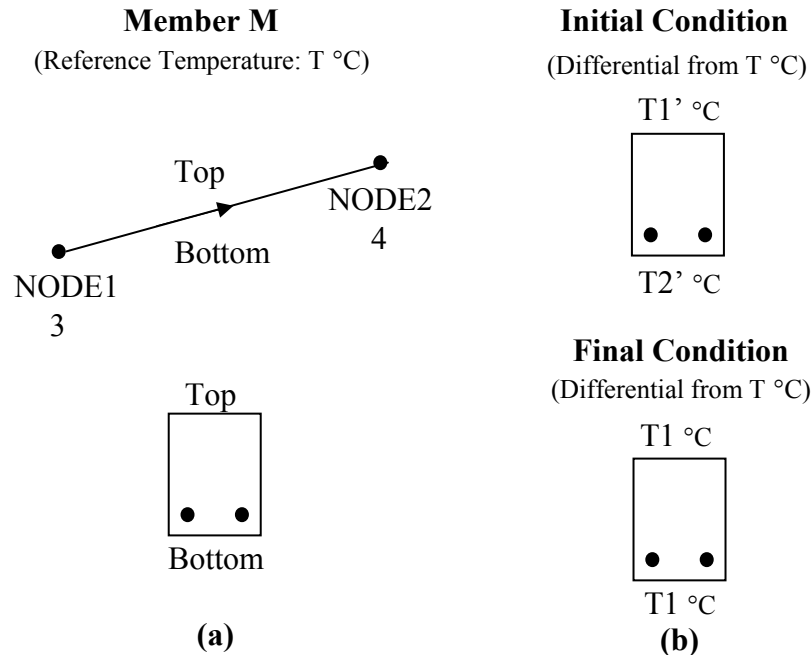
TEMPERATURE LOADS
*****
<NOTE:> UNITS: Deg.C,hrs
M T1' T2' T1 T2 TIME [ #M d(M) ] ← (2)/
/

```

In Table M.23,  $T1'$  is the initial temperature of the top of the cross section,  $T2'$  is the initial temperature of the bottom of the cross section,  $T1$  is the final temperature of the top of the cross section, and  $T2$  is the final temperature of the bottom of the cross section. These temperature values are differential values with respect to the reference temperature, which is defined in the *Auxiliary Data File* (see Table M.39). Assuming a reference temperature of +20 °C, for example,  $T1'$  must be +10 °C for an initial temperature of +30 °C at the top of the cross section.  $TIME$  is the time duration for which the thermal loading acts on the member.

The input field in the brackets [ ] may be used to specify a number of members subjected to the same thermal loading.  $\#M$  is the total number of members with the same thermal loading, and  $d(M)$  is the increment in the member number. Input values for a thermal

analysis are schematically presented in Figure M.12. More details regarding the thermal analysis are found in Section M3.3 (see Table M.34).



**Figure M.12** (a) Member M in Reference (Ambient) Temperature of T °C; (b) Top and Bottom Temperatures of the Member before and after the Thermal Loading

The *Load Case Data File* continues with the *Concrete Prestraints* input field. For any member subjected to *Concrete Prestraints*, this part of the *Load Data File* must be filled in. For example, for a member, say Member 1, under a shrinkage strain of  $-0.20 \times 10^{-3}$ , the *Concrete Prestraints* input must be as shown in Table M.24.

**Table M.24** Load Data File: Input Field for Concrete Prestraints (Example 1)

```

CONCRETE PRESTRAINS
*****
<NOTE:> UNITS: me
M STRAIN [ #M d(M) d(STRAIN) ] ← (2)/
1 -0.20/
/

```

In Table M.24, the input fields in brackets [ ] may be used for specifying a number of members with *Concrete Prestraints* following a certain pattern in magnitude. #M is the

total number of members to which the concrete prestraints is to be assigned,  $d(M)$  is the increment in the member number, and  $d(STRAIN)$  is the increment in the concrete prestrain. This increment can either be positive or negative. For example, for a structural model with 55 members, the same concrete prestrain can be applied to all members as shown in Table M.25.

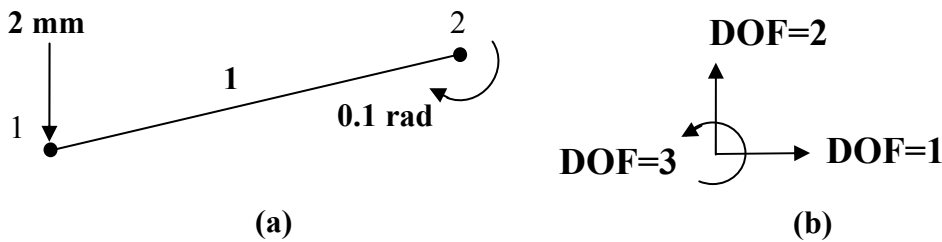
**Table M.25** Load Data File: Input Field for Concrete Prestraints (Example 2)

```

CONCRETE PRESTRAINS
*****
<NOTE:> UNITS: me
M STRAIN [#M d(M) d(STRAIN) ] ← (2)/
1 -0.20 55 1 0 /
/

```

The *Load Case Data File* continues with the *Prescribed Nodal Displacements* input field. For any node subjected to a translational or rotational displacement, this section of the *Load Data File* must be filled in. For example, for the part of a structural model shown in Figure M.13(a), the *Prescribed Nodal Displacements* input should be as shown in Table M.26.



**Figure M.13 (a)** A Member with Prescribed Nodal Displacements; **(b)** Degrees-of-Freedom for Prescribed Nodal Displacements

**Table M.26** Load Data File: Input Field for Prescribed Nodal Displacements

```

PRESCRIBED NODAL DISPLACEMENTS
*****
<NOTE:> UNITS: mm, rad
NODE DOF DISPL [#NODE d(NODE) ]/
1 2 -2/
2 3 -0.1/
/

```

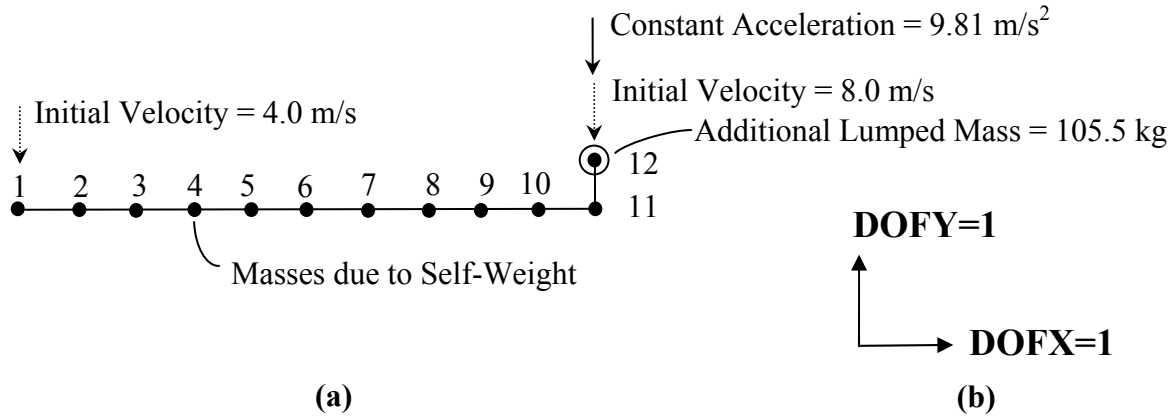
In Table M.26, *NODE* is the node number on which the nodal displacement is to be applied, *DOF* is the degree-of-freedom as shown in Figure M.13(b), *DISPL* is the magnitude of the applied nodal displacement or rotation. Note that *DOF* must always be entered as positive. For displacements acting in the negative directions, the *DISPL* value must be entered as negative.

The input field in the brackets [ ] may be used for specifying a number of nodes with the displacements having the same magnitude and direction. *#NODE* is the total number of nodes on which the same nodal displacements are acting, and *d(NODE)* is the increment in the node number.

The remaining input fields of the *Load Data File* are, only applicable when performing dynamic analyses which may involve the following loading conditions: ground acceleration loads (i.e., time-history analysis with earthquake accelerogram load data), impulse/impact/blast load - time history loads, and impact loads defined with an impacting mass and impact velocity.

The first input in the *Load Case Data File* for dynamic analyses is the *Additional Lumped Masses*. Through this input field, it is possible to assign additional nodal lumped masses to the nodes. Lumped masses due to the self-weight of the structure can automatically be considered, if desired, through the *Auxiliary Data File*, which is explained in Section M3.4 (see Table M.40). *Additional Lumped Masses* are automatically added to these self-masses. The resulting nodal masses are printed out in the related *Expanded Load Data File* (see Table M.67).

The purpose of this input field is to define *Additional Lumped Masses*, and is to assign *Initial Velocities* and *Constant Accelerations* to any mass which may be an additional or automatically calculated mass due to self-weight. As an example, for the model shown in Figure M.14(a), the *Additional Lumped Masses* must be input as shown in Table M.27.



**Figure M.14 (a)** A Member with Additional Lumped Masses, Initial Velocity and Constant Acceleration Loading; **(b)** Degrees-of-Freedom for Additional Lumped Masses

**Table M.27** Load Data File: Additional Lumped Masses

ADDITIONAL LUMPED MASSES  
\*\*\*\*\*

<NOTE:> UNITS: kg, m/s

NODE	DOF-X	DOF-Y	MASS	Vo-X	Vo-Y	Acc-X	Acc-Y	[ #NODE d(NODE) ]
1	0	1	0	0	-4.0	0	0/	
12	0	1	105.5	0	-8.0	0	-9.81/	
/								

In Table M.27, *NODE* is the node number to which the following assignments can be made. *DOF-X* and *DOF-Y* are the possible degrees-of-freedom for the assignments. A value of 1 must be entered to activate the degree-of-freedom and a value of 0 must be entered to ignore that degree-of-freedom. It is possible to consider both x- and y-degrees-of-freedom simultaneously. *Vo-X* and *Vo-Y* are the initial velocities in the x- and y-directions shown in Figure M.14(b) as *DOFX* and *DOFY*. *Acc-X* and *Acc-Y* are the constant accelerations in the x-and y-directions, respectively.

Note that the *Additional Lumped Masses* and the related assignments must be associated with the Load Case 1, as defined in Section M3.3 (see Tables M.35 and M.36), because all the additional lumped masses must be defined only once. Also note that neither *Additional Lumped Masses* nor the automatically calculated self-masses are converted to the static loads; they are only used in dynamic analyses. The automatic consideration of

the self-weight of the structure as static loads occurs through the *Load Data File*, as explained in Section M3.2 (see Tables M.21 and M.22).

*Initial velocities* are useful in performing impact analyses when only the impacting mass and the impact velocity are known. The technique of using a fictitious member (the member between *Nodes 11* and *12* in Figure M.14(a)) to simulate the load transfer from the impacting mass to the structure is discussed in Section 8.3 of Guner (2008) when modelling a number of beams for impact analyses.

*Constant accelerations* can be used, for example, to consider gravitational effects. They are used in Section 8.3 of Guner (2008) to simulate the gravity effects on the impacting mass to ensure that after the impact, the impacting mass will be under the influence of gravity. Note that the gravitational effects on the masses are not automatically considered; gravitational acceleration values must be assigned to all desired masses to consider such effects.

The *Load Case Data File* continues with the *Impulse, Blast and Impact Forces* input field as shown in Table M.28. Through the use of this input field, it is possible to define a multi-linear force-time history by four points as shown in Figure M.15.

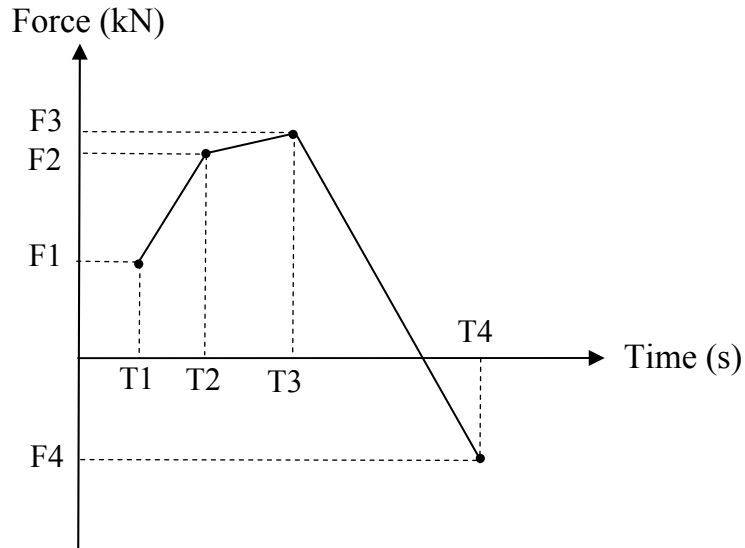
**Table M.28** Load Data File: Input Field for Impulse, Blast and Impact Loads

```

IMPULSE, BLAST AND IMPACT FORCES
*****
<NOTE:> UNITS: Sec, kN
NODE DOF T1 F1 T2 F2 T3 F3 T4 F4 [ #NODE d(NODE) ] /
4      1  0  500 0.001 2500 0.004 4000 0.007 -2500/
/

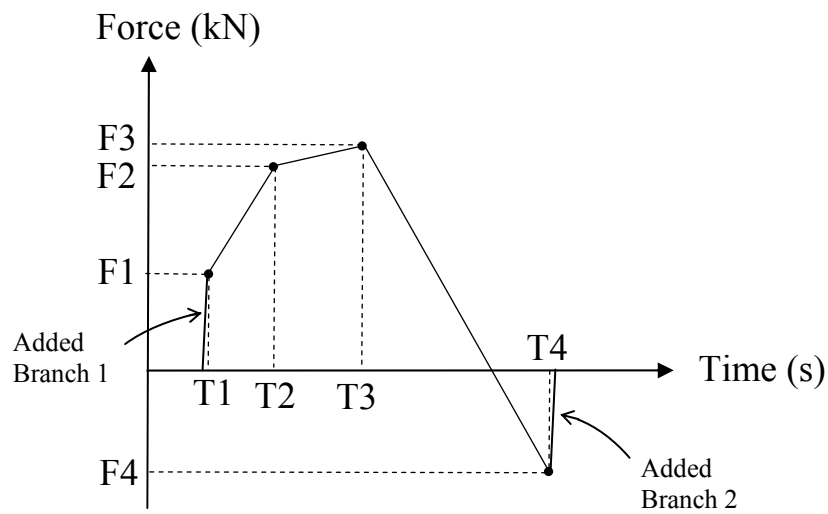
```

In Table M.28, NODE is the node number on which the *Impulse, Blast and Impact Forces* is to be applied, DOF is the degree-of-freedom as shown in Figure M.13(b). Note that DOF can only be positive 1 or 2; the rotational degree-of-freedom is not considered in this type of loading. T1, T2, T3, and T4 and F1, F2, F3 and F4 are the time and force values respectively, as shown in Figure M.15.



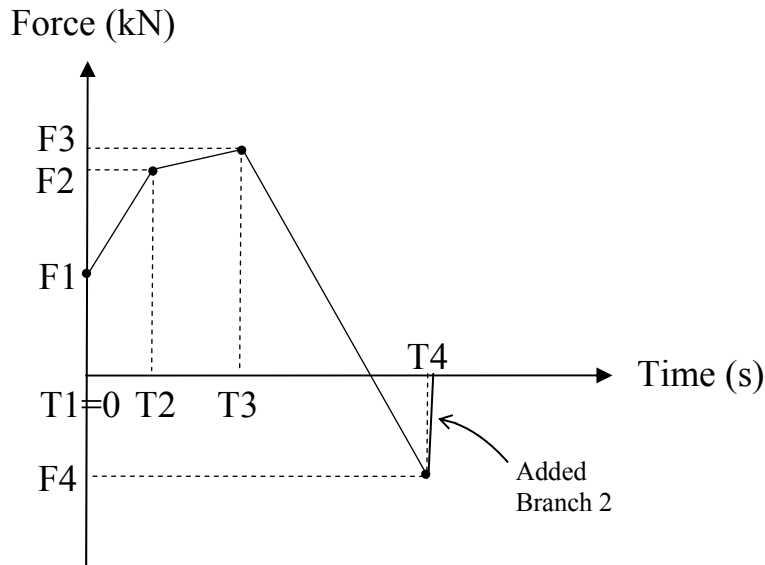
**Figure M.15** Multi-Linear Force-Time History

Since small variations in the force-time history may lead to significant differences in the computed responses, special attention must be exercised when defining the force-time history. If the first point of the loading corresponds to a nonzero time (i.e.,  $T1 > 0$ ), a preceding branch which originates from the previous time step will be automatically added by VecTor5 to the loading as shown in Figure M.16 (see the added branch 1). If the time step length is sufficiently small, the added branch may have an effect similar to a suddenly appearing force at  $T1 > 0$ . However, for a force truly appearing suddenly, a force-time history similar to the history shown in Figure M.17 must be used.



**Figure M.16** Multi-Linear Force-Time History with Automatically Added Branches

If the last point corresponds to a nonzero force (i.e.,  $F_4 > 0$ ), a branch is automatically added which goes to zero in the following time stage as shown in Figure M.16 and Figure M.17 (see the added branch 2).



**Figure M.17** Suddenly Appearing Multi-Linear Force-Time History

The *Load Case Data File* concludes with the *Ground Accelerations* input field. Other than using an external *VecTor.EQR Data File* explained below, *Ground Accelerations* can also be defined in the *Load Data File* by filling in the input field shown in Table M.29.

**Table M.29** Load Data File: Input Field for Ground Accelerations

```

GROUND ACCELERATIONS
*****
<NOTE:> UNITS: Sec, G
TIME  ACC-X  ACC-Y/
  0      0      0
  0.01  0.002984  0
  0.02 -0.00290  0
/

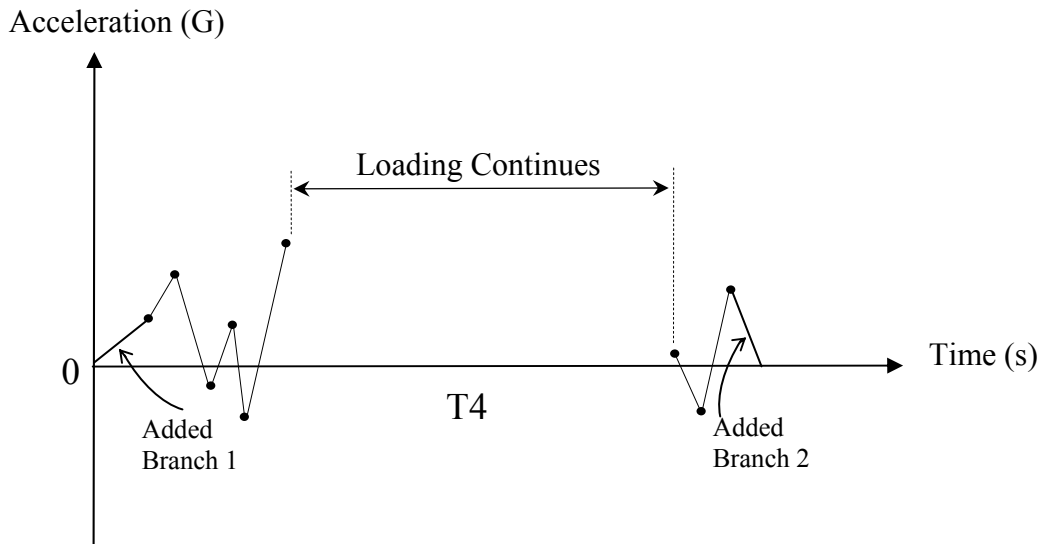
```

In Table M.29, *TIME* is the time corresponding to the acceleration value, and *ACC-X* and *ACC-Y* are the ground acceleration values acting in the global x- and y-directions. It is



possible to consider both acceleration components simultaneously acting on the structure. A scale factor can be applied to both or either component through the use of the *Auxiliary Data File* as explained in Section M3.4 (see Table M.40).

If the first point is defined with either a nonzero time or a nonzero acceleration value, a branch is automatically added by VecTor5 which goes to the origin regardless of the time step length used as shown in Figure M.18 (see the added branch 1). If the last point is defined with a nonzero acceleration value, a branch is automatically added which goes to a zero acceleration value at the next time stage, shown in Figure M.18 (see the added branch 2).



**Figure M.18** Ground Acceleration-Time History Loading (Accelerogram)

The total number of data points defined in this input field must be entered in the load case parameters located at the beginning of the *Load Data File* (see Table M.13, last line)

*Ground Accelerations* data can also be defined through the use of an external data file named *VecTor.EQR*. This data file must be created from nil as shown in Figure M.19 and placed in the same folder of a personal computer with the rest of the input data files.

```
1
NORTHTRIDGE EARTHQUAKE RECORDED BY SANTA MONICA CITY HALL
1000 0.02000 5 M
skip
skip
skip
0.060357202 0.060541042 0.061043608 0.058977426 0.060380844
0.058817228 0.067574517 0.035518577 -0.06813457 -0.028985117
```

**Figure M.19** VecTor.EQR Data File for Ground Acceleration Loads

In Figure M.19, the first line indicates the number of following lines to be skipped. The third line contains the following information consecutively: the total number of data points, time step length in seconds, total number of data points present at each line and units (*M* for  $m/s^2$ , *CM* for  $cm/s^2$ , *MM* for  $mm/s^2$  and *G* for *G*). The following three lines are set to be skipped by default. They may be used for storing additional information. Finally, the input of the acceleration data starts in a row-oriented order.

In order to consider a ground acceleration record as an external *VecTor.EQR Data File*, the analysis mode in the *Job Data File* must be set to 4 and the *VecTor.EQR Data File* must be placed under the same folder of a personal computer together with the other analysis input files. More details on the analysis modes are found in Section M3.3 (see Table M.37). The direction of the applied ground acceleration is specified in the *Auxiliary Data File*. It is also possible to consider a certain percentage of the same record acting in the global y-direction simultaneously with the main component acting in the global x-direction or vice versa. These operations are performed through the *Auxiliary Data File* as explained in Section M3.4 (see Table M.40).

When using the *VecTor.EQR Data File*, the first acceleration data point is automatically assumed to be (0, 0). The first value on this file must be the acceleration value corresponding to the end of the first time step length. For example, for an analysis with a time step length of 0.01 sec, the first acceleration value in the *VecTor.EQR Data File* must correspond to a time of 0.01 sec. If the last point is defined with a nonzero

acceleration value, a branch is automatically added which goes to zero acceleration value at the next time stage as shown in Figure M.18 (see the added branch 2).

The advantage of defining ground accelerations as a *VecTor.EQR Data File* rather than as a *Load Data File* is its convenience. As most earthquake records can be downloaded from the internet in a row-oriented order, they can be transformed to *VecTor.EQR Data File* conveniently. However, it is not possible to apply two different ground acceleration records acting simultaneously in both the global x- and y-directions when using a *VecTor.EQR Data File*. When defining the earthquake acceleration data through the *Load Data File*, it is possible to consider the different ground acceleration records acting simultaneously on the structure; both components can also be scaled through the use of the *Auxiliary Data File* (see Table M.40).

### M3.3 Job Data File (VecTor.JOB)

The *Job Data File* includes the input fields for the *Loading Data*, *Analysis Parameters* and *Material Behaviour Models*. The file starts with the information presented in Table M.30.

**Table M.30** Job Data File: Input Field for Structure Data

Job Title	(30 char max)	: <b>VSA1 Beam</b>
Job File Name	( 8 char max)	: <b>VS-A1</b>
Date	(30 char max)	: <b>June 2008</b>
STRUCTURE DATA		
-----		
Structure Type		: <b>5</b>
File Name	(8 char max)	: <b>VSA1</b>

In Table M.30, the *Job Tile*, *Job File Name* and *Date* can be used for information purposes; they are not required for the analysis operations. *Structure Type* must be 5, indicating that this is a VecTor5 analysis. The *File Name* should be the same as the *Structure File Name* as defined in Section M3.1 (see Table M.1).

The *Job Data File* continues with the *Loading Data* input field as shown in Table M.31.

**Table M.31** Job Data File: Input Field for Loading Data (Monotonic Loads)

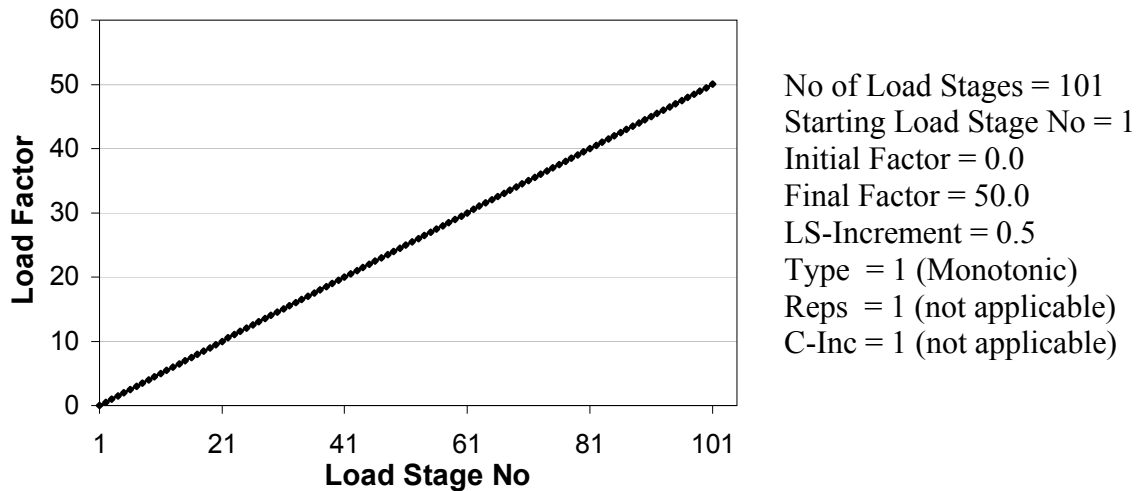
LOADING DATA							
-----							
No. of Load Stages	: 101						
Starting Load Stage No.	: 1						
Load Series ID	(5 char max) : VSA1						
Load Case	File Name	Initial	Final	LS-Inc	Type	Reps	C-Inc
1	VSA1VL	0.000	50.000	0.500000	1	1	0.000
2	NULL	0.000	0.000	0.000000	1	1	0.000
3	NULL	0.000	0.000	0.000000	1	1	0.000
4	NULL	0.000	0.000	0.000000	1	1	0.000
5	NULL	0.000	0.000	0.000000	1	1	0.000

In Table M.31, *Load Series ID* is used to name the *Output Files*. *Output Files* named VSA1\_01.A5E, VSA1\_02.A5E and so on will be produced at each load (time) stage for this particular example. The extension \_02 indicates to which load (time) stage the *Output File* belongs. If the output of seed files is requested in the *Analysis Parameters* defined below, *Output Files* named VSA1\_01.A5R, VSA1\_02.A5R and so on will also be produced.

A total number of five different load cases can be considered as temperature, static or dynamic loads. Temperature and static loads can be considered as monotonic, cyclic or reversed-cyclic. Dynamic loads are applied with their magnitudes as specified in the *Load Case Data File*. In other words, the magnitude of the dynamic loads is not changed through the *Job Data File*. Consider the three possible static analysis options below.

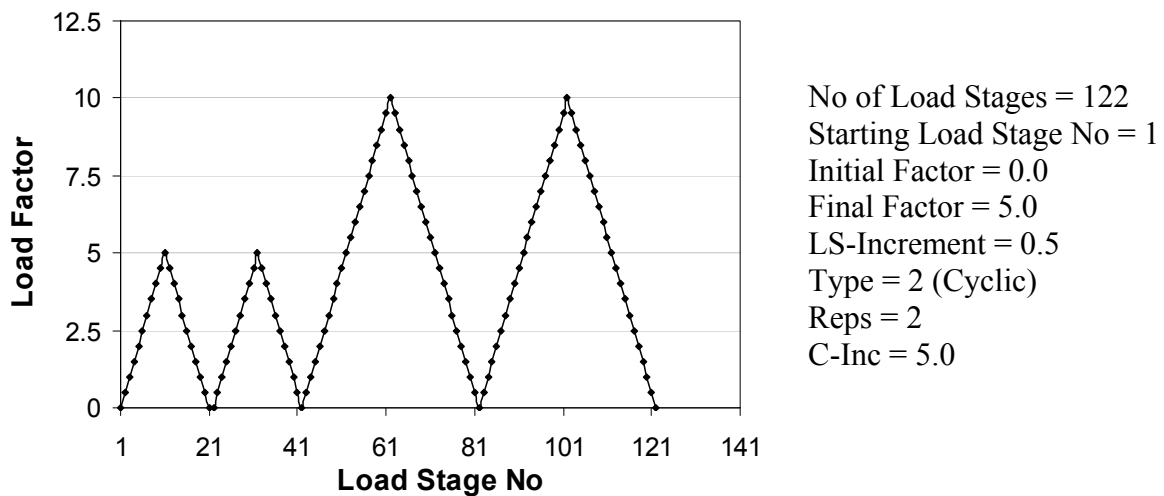
As an example of a ‘monotonic loading condition’, consider the analysis of Beam VS-A1. In the *Load Data File* VSA1VL, a vertical displacement of 1 mm was applied in the downwards direction at the midspan of the beam. It is now desired to increase this displacement monotonically until failure takes place. The loading pattern shown in Figure M.20 with 101 load stages and a load increment factor of 0.5 may be used for this purpose (see Table M.31 for the input). As the failure displacement of the beam is not

known before the analysis, it is wise to consider a large number of load stages, say 101 load stages with an increment of 0.5 mm, to avoid the possibility of the termination of the analysis before the failure of the beam occurs. Once the failure occurs, the program will terminate automatically; therefore, there is no disadvantage of specifying a large number of load stages at the beginning of the analysis.



**Figure M.20** Monotonic Loading Condition

Consider now the analysis of the beam under the same midspan displacement of 1.0 mm applied as a ‘cyclic load’. The loading pattern shown in Figure M.21 may be used for this purpose (See Table M.32 for the input). In this analysis, a midspan displacement that will be changed in increments of 0.5 mm, and repeated for 2 cycles, will be applied before the load amplitude is increased by a factor of 5.0 (5 x 1.0 mm).



**Figure M.21** Cyclic Loading Condition

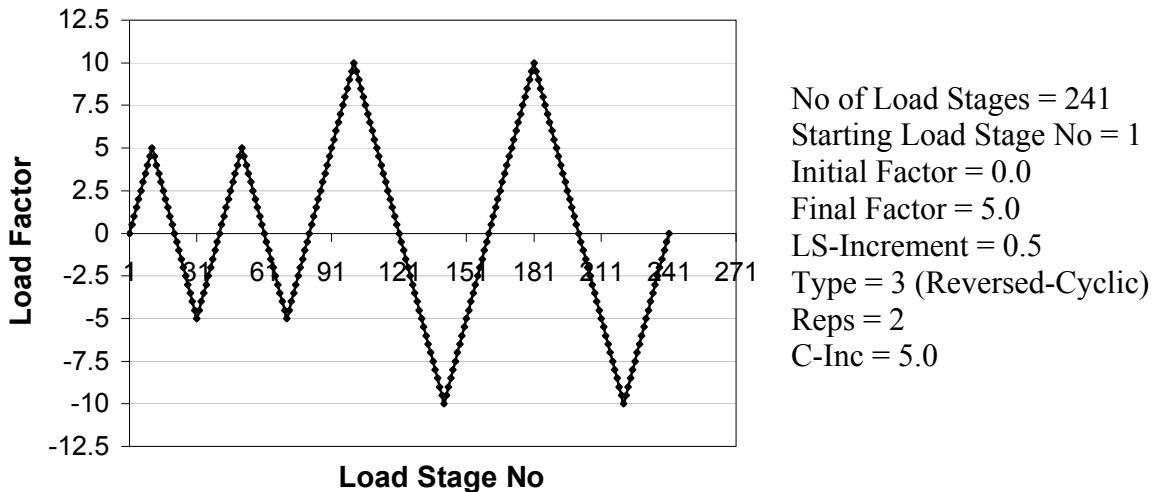
**Table M.32** Job Data File: Input Field for Loading Data (Cyclic Loads)

LOADING DATA

-----  
 No. of Load Stages : **122**  
 Starting Load Stage No. : **1**  
 Load Series ID (5 char max) : **VSA1**

Load Case	File Name (8 char max)	Initial	Final	LS-Inc	Type	Reps	C-Inc
1	<b>VSA1VL</b>	<b>0.000</b>	<b>5.000</b>	<b>0.500000</b>	<b>2</b>	<b>2</b>	<b>5.000</b>
2	NULL	0.000	0.000	0.000000	1	1	0.000
3	NULL	0.000	0.000	0.000000	1	1	0.000
4	NULL	0.000	0.000	0.000000	1	1	0.000
5	NULL	0.000	0.000	0.000000	1	1	0.000

Consider now the analysis of the beam under the same midspan displacement of 1.0 mm applied as a ‘reversed-cyclic load’. A loading protocol shown in Figure M.22 may be used for this purpose (see Table M.33 for the input). In this analysis, a midspan displacement that will be changed in increments of 0.5 mm, and a repeated for 2 cycles will be applied before the load amplitude is increased by 5.0 mm.



**Figure M.22** Reversed-Cyclic Loading Condition

**Table M.33** Job Data File: Input Field for Loading Data (Reversed-Cyclic Loads)

LOADING DATA

-----  
 No. of Load Stages : **241**  
 Starting Load Stage No. : **1**  
 Load Series ID (5 char max) : **VSA1**

**Table M.33** Job Data File: Input Field for Loading Data (Reversed-Cyclic) (Continued)

Load Case	File Name (8 char max)	Factors					
		Initial	Final	LS-Inc	Type	Reps	C-Inc
1	VSA1VL	0.000	5.000	0.500000	3	2	5.000
2	NULL	0.000	0.000	0.000000	1	1	0.000
3	NULL	0.000	0.000	0.000000	1	1	0.000
4	NULL	0.000	0.000	0.000000	1	1	0.000
5	NULL	0.000	0.000	0.000000	1	1	0.000

When performing ‘thermal analyses’, the three load application options defined above can be used. However, the user must define whether the load or the time is factored (see Table M.13). As an example, consider the temperature load shown in Table M.34, where Members 1 to 17 are exposed to a temperature increase of 50 °C at the top fibre of their cross sections for 1 hour.

**Table M.34** Example Temperature Loading

```

TEMPERATURE LOADS
*****
<NOTE:> UNITS: Deg.C,hrs
M  T1' T2' T1 T2 TIME [ #M  d(M) ] ← (2)/
1   0  0  50  0   1   17   1/
/

```

In order to determine the exposure time which will cause the structure to fail under the given thermal loading, a *Time Factored* analysis must be performed. In this case, the loading data in the *Job Data File* will control the time. For example, a monotonic analysis can be performed where the time is increased by 1 hour at each load (time) stage.

In order to determine the temperature gradient which will cause the structure to fail for the time specified (1 hour in Table M.34), a *Load Factored* analysis must be performed. In this case, the loading data in the *Job Data File* will control the temperature gradient. For example, it is possible to perform a monotonic analysis where the temperatures *T1'*, *T2'*, *T1* and *T2* are increased at each load (time) stage by 0, 0, 25, 0 °C respectively by providing a *LS-INC* of 0.5 in the *Job Data File*.

It should be noted that the specified temperature gradients are differential with respect to the *Reference Temperature* defined in the *Auxiliary Data File* (Section M3.4). For example, for a *Reference Temperature* of 20 °C, the real values of temperatures are  $T1'$ ,  $T2'$ ,  $T1$  and  $T2$  are 20, 20, 70, 20 °C, respectively

In dynamic analyses, the loading data is used to specify the time stages. As an example, consider the analysis of Beam VSA1 under an input ground acceleration record defined in the load case of VSA1ACC. Assume that the analysis will be performed for the first 5 seconds of the record with a time step length of 0.01 sec. The corresponding loading data is presented in Table M.35.

**Table M.35** Job Data File: Input Field for Loading Data for Dynamic Loads  
(Output at Each Time Stage)

LOADING DATA  
-----

No. of Load Stages : **501**  
 Starting Load Stage No. : **1**  
 Load Series ID (5 char max) : **VSA1**

Load Case	File Name (8 char max)	Initial	Factors Final	LS-Inc	Type	Reps	C-Inc
1	<b>VSA1ACC</b>	<b>0.000</b>	<b>5.000</b>	<b>0.010000</b>	<b>1</b>	<b>1</b>	<b>0.000</b>
2	NULL	0.000	0.000	0.000000	1	1	0.000
3	NULL	0.000	0.000	0.000000	1	1	0.000
4	NULL	0.000	0.000	0.000000	1	1	0.000
5	NULL	0.000	0.000	0.000000	1	1	0.000

In dynamic analyses, the input fields *Type* and *C-Inc* are not used; any value can be input in those fields. The input field *Reps* can be used to specify the desired output interval. For illustrative purposes, consider the loading data shown in Table M.36.

**Table M.36** Job Data File: Input Field for Loading Data for Dynamic Loads  
(Output at Selected Time Stage Intervals)

LOADING DATA  
-----

No. of Load Stages : **501**  
 Starting Load Stage No. : **1**  
 Load Series ID (5 char max) : **VSA1**



**Table M.36** Job Data File: Input Field for Loading Data for Dynamic Loads  
(Output at Selected Time Stage Intervals) (Continued)

Load Case	File Name (8 char max)	Factors					
		Initial	Final	LS-Inc	Type	Reps	C-Inc
1	<b>VSA1ACC</b>	<b>0.000</b>	<b>5.000</b>	<b>0.100000</b>	<b>1</b>	<b>10</b>	<b>0.000</b>
2	NULL	0.000	0.000	0.000000	1	1	0.000
3	NULL	0.000	0.000	0.000000	1	1	0.000
4	NULL	0.000	0.000	0.000000	1	1	0.000
5	NULL	0.000	0.000	0.000000	1	1	0.000

In the analysis above, a time step length of  $0.1 / 10 = 0.01$  will be used as was in the previous example. There will again be 501 load stages performed. However, the *Output Files* are produced at each tenth load stage; i.e., load stages 1, 11, 21, 31, ..., 501. This option is useful to limit the number of *Output Files* when performing a dynamic analysis with a large number of load (time) stages. The analysis will also take less time because there will be significantly fewer files saved to the hard drive during the analysis.

The selection of the time step length may have a significant impact on the computed responses of the structures as discussed in Section 7.9 of Guner (2008). Therefore a second analysis is recommended with the use of a smaller time step length to make sure that the computed response does not change significantly.

In the case of a dynamic analysis, *Load Cases 2 to 5* can be used to specify either a static load case as defined in Tables M.31, M.32 and M.33 or a dynamic load case. For these load cases, the factors (i.e., *Initial*, *Final*, *LS-Inc*, *Type*, *Reps* and *C-Inc*) can be used to scale the load case and to define a monotonic, cyclic or reversed-cyclic loading condition. In other words, only *Load Case 1* is used to specify the time stages in a dynamic analysis.

The *Job Data File* continues with the *Analysis Parameters* shown in Table M.37.

**Table M.37** Job Data File: Input Field for Analysis Parameters

ANALYSIS PARAMETERS

Analysis Mode	(1,3,4) : <b>3</b>
Seed File Name	(8 char max) : <b>NULL</b>
Convergence Limit	(>1.0) : <b>1.00001</b>
Averaging Factor	(<1.0) : <b>0.</b>

**Table M.37** Job Data File: Input Field for Analysis Parameters (Continued)

Maximum Iterations	: <b>100</b>
Convergence Criteria	(1-3) : <b>1</b>
Results Files	(1-4) : <b>1</b>
Output Format	(1) : <b>1</b>

Possible input values for the analysis parameters, shown in brackets in Table M.37, are listed below.

*Analysis Mode*

- 1 : All static and thermal analyses including monotonic, cyclic and reversed cyclic loading conditions,
- 3 : All dynamic analyses except time-history analyses defined with a *VecTor.EQR Data File*,
- 4 : Time-history analyses defined with a *VecTor.EQR Data File*.

Note that there is no option 2 currently available.

*Seed File Name*

Seed files are *Output Files* written in a binary format, which can be used to continue a previous analysis. They may be needed when a damaged structure is desired to be analyzed under a different loading condition. It can also be used to simply resume the previous analysis at a later time.

In order to resume an analysis, the *Output Seed File* name corresponding to the last load stage of the previous analysis must be specified. For example, assume that a cyclic analysis of Beam VSA1 with 50 load stages was performed and it is now desired to continue the same analysis with a reversed-cyclic loading. In such a case, the *Output Seed File* named VSA1\_50.A5R must be supplied in this area, and the new loading must be defined before starting the analysis.

*Convergence Limit*

The default value of 1.00001 is suggested for this purpose.

### *Averaging Factor*

The default value of 0 indicates that the dynamic averaging scheme will be used. Advanced users may refer to Section 3.10 of Guner (2008) for more detailed information on other averaging factors.

### *Maximum Number of Iterations*

The default value of 100 is suggested for the maximum number of iterations. Section 3.12 of Guner (2008) includes more information on the use of larger number of maximum number of iterations for advanced users.

### *Convergence Criteria*

- 1 : Unbalanced Forces,
- 2 : Weighted Displacements,
- 3 : Maximum Displacements.

The default option 2 is recommended as the convergence criteria. More detailed information on the convergence criteria formulations are found in Section 3 of Guner (2008) for advanced users.

### *Result Files*

- 1 : ASCII (regular text *Output Files*) and Binary (*Output Seed Files*) files,
- 2 : ASCII files only,
- 3 : Binary Files Only,
- 4 : ASCII and Binary Files, Last Load Stage Only.

The default option 2 is recommended for the result files. If there is a possibility of resuming the analysis with the use of *Output Seed Files*, an option including the output of a binary seed file must be selected (i.e., options 1, 3 or 4).

### *Output Format*

- 1: To computer.

There is only one output format currently available.

The *Job Data File* concludes with the *Material Behaviour Models* input field as shown in Table M.38. The material models which provide reasonable simulations in all analyses are defined as default options and listed in Table 4.1 of Guner (2008). The motivations for the availability of other options for advanced users are discussed in Section 4.2 of Guner (2008). In the default mode, the value of 1 must be entered for each of the available model.

Among the *Material Behaviour Models*, *Concrete Compression Base Curve* is recommended to be selected based on the concrete strength used, as explained in Section M3.1 (see Eq. M.4 to Eq. M.6). The available base curves are:

- 0 : Linear,
- 1 : Hognestad (Parabola) (Normal Strength Concrete: NSC),
- 2 : Popovics (NSC),
- 3 : Popovics (High Strength Concrete: HSC),
- 4 : Hoshikuma (HSC).

A detailed list and formulation for all available material behaviour models are found in Wong and Vecchio (2002), where the options are examined in the order which starts from the option number 0 and increases by 1. For example, the model of *Concrete Compression Post-Peak* is treated in Section 4.2 of Wong and Vecchio (2002). This section includes sub-sections 4.2.1 *Pre-Peak Base Curve*, 4.2.2 *Modified Park-Kent*, and so on. If a value of 0 is input in Table M.38 for *Concrete Compression Base Curve*, it will correspond to *Pre-Peak Base Curve*. Similarly, a value of 1 will correspond to the *Modified Park-Kent* which is the default value.

**Table M.38** Job Data File: Input Field for Material Behaviour Models

MATERIAL BEHAVIOUR MODELS

Concrete Compression Base Curve	(0-4) : 1
Concrete Compression Post-Peak	(0-3) : 1
Concrete Compression Softening	(0-8) : 1

**Table M.38** Job Data File: Input Field for Material Behaviour Models (Continued)

Concrete Tension Stiffening	(0-5) : 1
Concrete Tension Softening	(0-3) : 1
Concrete Tension Splitting	(1-2) : 1
Concrete Confined Strength	(0-2) : 1
Concrete Dilatation	(0-1) : 1
Concrete Cracking Criterion	(0-4) : 1
Concrete Crack Slip Check	(0-2) : 1
Concrete Bond or Adhesion	(0-4) : 1
Concrete Creep and Relaxation	(0-1) : 1
Concrete Hysteresis	(0-3) : 1
Reinforcement Hysteresis	(0-3) : 1
Reinforcement Dowel Action	(0-1) : 1
Reinforcement Buckling	(0-1) : 1
Element Strain Histories	(0-1) : 1
Element Slip Distortions	(0-4) : 1
Strain Rate Effects	(0-2) : 1
Structural Damping	(0-2) : 1
Geometric Nonlinearity	(0-1) : 1
Crack Allocation Process	(0-1) : 1

Two of these material behaviour models are applicable, when performing a dynamic analysis, as described below:

#### *Strain Rate Effects*

- 0 : Not considered,
- 1 : CEB-FIB (1990) model for the concrete and Malvar and Crawford model (1998) for the reinforcement,
- 2 : CEB-FIB (1990) model for the concrete and CEB-FIB (1988) model for the reinforcement.

The default option 1 is recommended for the consideration of strain rate effects. Details of these models can be found in Section 7.8 of Guner (2008); discussion of these options is presented in Section 8.8.5 of Guner (2008).

#### *Structural Damping*

- 0 : No additional viscous damping is considered,
- 1 : Rayleigh damping formulation is used with the additional viscous damping ratios specified in the *Auxiliary Data File*,
- 2: Alternative damping formulation is used with the additional viscous damping ratios specified in the *Auxiliary Data File*.

Specification of additional viscous damping ratios of 0.0% in the *Auxiliary Data File* will cause the analysis to be undamped regardless of the *Structural Damping* option selected here (see Table M.40). The default option 1 is suggested in the cases where additional viscous damping is desired to be used. Detailed discussion on the selection of these options is presented in Section 7.5.2 of Guner (2008). In addition, Sections 7.10 and 7.11 of Guner (2008) contain more advanced discussion on the use of additional viscous damping.

### M3.4 Auxiliary Data File (VT5.AUX)

The *Auxiliary Data File* consists of two sets of input fields: *General Analysis Parameters* and *Dynamic Analysis Parameters*. *General Analysis Parameters* are shown in Table M.39.

**Table M.39** Auxiliary Data File: Input Field for General Analysis Parameters

```

GENERAL ANALYSIS PARAMETERS
*****
Section Analysis Mode      (1-5) : 1
Shear Analysis Mode       (0-4) : 3
Shear Protection          (0-1) : 1

Concrete Aggregate Type   (1-2) : 1
Reference Temperature     (deg. C): 20.0
    
```

Possible input values for the *General Analysis Parameters*, shown in brackets in Table M.39, are listed below.

#### *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 option 1 is recommended as the section analysis mode.

### *Shear Analysis Modes*

- 0 : Shear not considered,
- 1 : Uniform Shear Flow (Multi-Layer),
- 2 : Uniform Shear Strain (Multi-Layer),
- 3 : Parabolic Shear Strain (Multi-Layer),
- 4 : Uniform Shear Strain (Single-Layer, approximate analysis).

The default option 3 is recommended as the shear analysis mode. More details of these analysis options are found in Section 3.7 of Guner (2008) for advanced users.

### *Shear Protection Algorithm*

Shear protection algorithm, as defined in Section 3.11 of Guner (2008), is considered by default for an option value of 1. To turn this feature off, a value of 0 must be supplied.

### *Concrete Aggregate Type*

- 1 : Carbonate aggregate,
- 2 : Silicious aggregate.

Concrete aggregate type is used when performing a thermal analysis. The effects of the use of different aggregate types are presented in Section 3.5 of Guner (2008).

### *Reference Temperature*

The reference temperature is the ambient temperature against which the top and bottom sectional temperatures are defined in a thermal analysis as defined in Section M3.2 (see Table M.23).

The *Auxiliary Data File* concludes with the *Dynamic Analysis Parameters* as shown in Table M.40.

**Table M.40** Auxiliary Data File: Input Field for Dynamic Analysis Parameters

```
DYNAMIC ANALYSIS PARAMETERS
*****
Time Integration Method (1-3)      : 3

Damping Assigned to 1st Mode      : 1
Damping Assigned to 2nd Mode      : 2
Damping Ratio for 1st Mode (%)     : 0
Damping Ratio for 2nd Mode (%)     : 0

Ground Accel. Factor in x-dir      : 1.0
Ground Accel. Factor in y-dir      : 0.0

Mass Factor due to Self-Weight     : 1.0
```

Possible input values for the *Dynamic Analysis Parameters* are listed below.

#### *Time Integration Method*

- 1 : Newmark's Average Acceleration,
- 2 : Newmark's Linear Acceleration,
- 3 : Wilson's Theta Method.

The default option 3 is recommended as the time integration method. As discussed in Section 7.10 of Guner (2008), the use of Newmark's Methods may require the use of additional viscous damping for stability reasons. In addition, Newmark's Linear Acceleration Method is a conditionally stable procedure as discussed in Section 7.6.4 of Guner (2008). More details on these time integration methods are found in Chapter 7 of Guner (2008).

#### *Additional Viscous Damping*

When using either the Rayleigh or the Alternative Damping as a way of providing additional viscous damping to the structure, the selection of two vibrational modes and the specification of the corresponding damping ratios are required. For the Rayleigh Damping, the damping ratios for the remaining modes are calculated automatically by VecTor5 as described in Section 7.5.2 of Guner (2008). For the Alternative Damping, as defined in Section 7.5.2 of Guner (2008), the remaining vibrational modes are assumed to



be undamped. It is also possible to assign zero damping for one of the two specified vibration modes when using the alternative damping option. In this case, only one vibrational mode of the structure will be damped and all remaining modes will be undamped. Such an option is not possible when using the Rayleigh Damping.

#### *Ground Acceleration Factors*

When performing a time-history analysis through the use of the *VecTor.EQR Data File*, two factors must be defined to specify the loading direction and to scale the input ground motion. The ground motion is applied to the structure as a combination of the global x- and y-directions based on the specified ground acceleration factors. A factor of 1.0 in the x-direction and a factor of 0 in the y-direction will cause the motion to be entirely applied in the x-direction with the magnitude defined in the *VecTor.EQR Data File*. It is also possible to apply a certain percentage of the same ground motion in the y-direction. For example, the input of 1.0 and 0.25 in x- and y-directions will cause input ground motion to be applied in the x-direction with a scale factor of 1.0 and cause the same ground motion to be applied in the y-direction after all acceleration values are multiplied by the scale factor of 0.25.

When performing a time-history analysis through the *Load Data File*, the scale factors are used to scale the input acceleration values in the x- and y-directions respectively. In this case, two different acceleration records can be defined in the *Load Data File* and applied with different scale factors simultaneously.

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

To consider the nodal lumped masses due to the self-weight of the structure, a nonzero factor must be supplied. A factor of 1.0 corresponds to the unfactored self-mass of the structure. Note that self-masses are only used in dynamic analyses; they are not converted to static forces. The self-weight of the structure can be automatically considered as static loads through the *Load Data File*, as explained in Section M3.2 (see Tables M.21 and M.22).

## CHAPTER M4

### VECTOR5 OUTPUT FILES

This chapter describes the analysis results *Output Files* produced by VecTor5 for each load or time stage. The chapter also discusses the determination of damage or failure modes by examining the analysis results *Output Files*.

#### M4.1 Output Files (Analysis Results)

As introduced in Section M2.4, VecTor5 produces one *Output File* with the extension of A5E for each of the load or time stages being considered. For demonstrative purposes, consider the analysis result of Beam VS-A1, introduced in Section M3.1, for *Load Stage 10* under the monotonic loading in Figure M.20.

The *Output File* starts with the *General Analysis Parameters* and *Loading Condition* as shown in Table M.41.

**Table M.41** Output File: General Analysis Parameters and Loading Condition

```
Job Title           : VSA1
Job File Name      : VSA1
Date               : Jan 2004
Structure File Name : VSA1
Load Series I.D.   : VSA1

Load Stage No.     : 10
Iteration No.      : 20
Convergence        : 1.000007
```

#### LOADING CONDITION

\*\*\*\*\*

LOAD CASE TITLE	FILE NAME	LOAD FACTOR
VSA1 Vertical	VSA1VL	4.500

The *Output File* continues with the *Convergence Factors* for the *Compatibility Forces* and the *Effective Stiffnesses* as shown in Table M.42.

**Table M.42** Output File: Convergence Factors

CONVERGENCE FACTORS						
*****						
M	COMPATIBILITY FORCES			EFFECTIVE STIFFNESSES		
	PU/PU''	SU/SU''	MU/MU''	AX/AX''	IZ/IZ''	IZ/IO
1	1.000	1.000	1.000	1.000	1.000	1.000
2	1.000	1.000	1.000	1.000	1.000	1.000
3	1.000	1.000	1.000	1.000	1.000	1.000
4	1.000	1.000	1.000	1.000	1.000	1.000
5	1.000	1.000	1.000	1.000	1.000	1.000
6	1.000	1.000	1.000	1.000	1.000	1.000

In Table M.42,  $M$  is the member number,  $PU$ ,  $SU$  and  $MU$  are the compatibility axial, shear and moment values at the last iteration (iteration number 20 for this example),  $PU''$ ,  $SU''$  and  $MU''$  are the compatibility axial, shear and moment values at the iteration one before the last iteration (iteration number 19 for this example). Advanced users may refer to Section 3.6 of Guner (2008) for detailed formulations. These ratios are useful when evaluating the convergence characteristics for each member. The *Convergence* factor of the complete structure in Table M.41, should also be considered in such an evaluation. *Convergence* factors in excess of 1.5 should raise concerns about the validity of the analysis result of that particular load stage indicating that the structure may be failing. Such a situation will most likely be visible on the load-deflection curve of the structure.

In Table M.42,  $AX$  is the transformed axial stiffness,  $IZ$  is the transformed bending stiffness, and  $IO$  is the gross bending stiffness of the members. The subscript “'' indicates that the values come from the iteration one before the last iteration. These ratios are useful when evaluating the convergence characteristics for each member in case of an *Effective Stiffness Analysis* as defined in Section M3.4.

The *Output File* continues with *Compatibility Forces* for the *Restoring Forces* and the *Unbalanced Forces* in the member-oriented coordinate system given in Figure M.8, as shown in Table M.43.

**Table M.43** Output File: Compatibility Forces

COMPATIBILITY FORCES						
*****						
M	RESTORING FORCES			UNBALANCED FORCES		
	PR (kN)	SR (kN)	MR (kN-m)	PU (kN)	SU (kN)	MU (kN-m)
1	19.5	-817.0	0.3	-0.0	0.0	-0.0
2	173.7	-2123.4	47.1	-0.0	0.0	-0.0
3	524.4	-2955.7	112.1	0.0	-0.0	0.0
4	865.8	-3635.7	176.4	0.1	-0.1	0.0
5	1183.6	-4107.0	241.0	0.0	-0.1	0.0
6	1431.5	-2012.4	305.8	-0.0	0.0	-0.0

In Table M.43,  $M$  is the member number,  $PR$ ,  $SR$  and  $MR$  are the compatibility restoring axial, shear and moment values as defined in Sections 3.4.3 and 3.6.8 of Guner (2008),  $PU$ ,  $SU$  and  $MU$  are the unbalanced axial, shear and moment values as defined in Section 3.6.8 of Guner (2008). Unbalanced forces should ideally be zero at all load stages. Significant unbalanced forces as compared to the total acting forces on the member (see Table M.44) may indicate that the member is failing. Such a situation is automatically detected by VecTor5 and the analysis is terminated, as described in Section 3.12 of Guner (2008). However, it is recommended inspecting the unbalanced forces for the load stages corresponding to the strength (peak load capacity) of the structure.

The *Output File* continues with the *Member End Forces* in the member-oriented coordinate system given in Figure M.8, as shown in Table M.44.

**Table M.44** Output File: Member End Forces

MEMBER END FORCES						
*****						
M	N1 (kN)	V1 (kN)	M1 (kN-m)	N2 (kN)	V2 (kN)	M2 (kN-m)
1	-0.42	123.30	-0.00	0.42	-123.30	37.55
2	-0.41	123.30	-37.55	0.41	-123.30	75.03
3	-0.38	123.30	-75.03	0.38	-123.30	112.44
4	-0.31	123.31	-112.44	0.31	-123.31	149.82
5	-0.22	123.31	-149.82	0.22	-123.31	187.17
6	-0.08	123.31	-187.17	0.08	-123.31	224.66

In Table M.44,  $M$  is the member number,  $N1$ ,  $V1$ , and  $M1$  are the axial force, shear force and bending moment values of  $NODE1$  of the member,  $N2$ ,  $V2$ , and  $M2$  are the axial force, shear force and bending moment values of  $NODE2$  of the member,

The *Output File* continues with the *Support Reactions* in the global coordinate system given in Figure M.6(b), as shown in Table M.45.

**Table M.45** Output File: Support Reactions

SUPPORT REACTIONS			
*****			
NODE	X-REAC (kN)	Y-REAC (kN)	Z-REAC (kN-m)
1	0.00	123.31	0.00
7	0.00	-123.31	224.66

The *Output File* continues with the *Nodal Displacements* in the global coordinate system given in Figure M.6(b), as shown in Table M.46.

**Table M.46** Output File: Nodal Displacements

NODAL DISPLACEMENTS			
*****			
NODE	X-DSPL (mm)	Y-DSPL (mm)	Z-ROT (Rad)
1	-0.228	0.000	-0.00336
2	-0.231	-1.035	-0.00332
3	-0.224	-2.052	-0.00310
4	-0.197	-2.981	-0.00265
5	-0.149	-3.752	-0.00199
6	-0.082	-4.296	-0.00111
7	0.000	-4.500	0.00000

The *Output File* continues with the *Member Deformations* as shown in Table M.47.

**Table M.47** Output File: Member Deformations

MEMBER DEFORMATIONS							
*****							
M	ECL	GXY	PHI	ESL-MAX	ESL-MIN	EST-MAX	WCR-MAX
	(me)	(me)	(me/m)	(me)	(me)	(me)	(mm)
1	-0.002	-0.045	0.135	0.027	-0.033	0.006	
2	0.027	-0.117	0.732	0.182	-0.138	0.025	0.05
3	0.094	-0.162	1.458	0.403	-0.235	0.042	0.12
4	0.160	-0.200	2.177	0.621	-0.332	0.057	0.18
5	0.221	-0.226	2.899	0.836	-0.434	0.072	0.24
6	0.269	-0.111	3.623	1.037	-0.550	0.088	0.30

In Table M.47, *M* is the member number, *ECL* is the axial concrete strain at the mid-depth of the cross section, *GXY* is the shear strain of the concrete at the mid-depth, *PHI* is the curvature. Advanced users may refer to Section 3.6.11 of Guner (2008) for the formulations. *ESL-MAX* is the maximum longitudinal reinforcement strain, *ESL-MIN* is the minimum longitudinal reinforcement strain, *EST-MAX* is the maximum transverse reinforcement strain and *WCR-MAX* is the maximum average crack width for the related member. All reinforcement strains are average values.

The *Output File* continues with the *Detailed Member Output* for the members defined in the *Detailed Member Output List* in Table M.11. For illustrative purposes, consider *Member 5* of VS-A1 beam. The *Detailed Member Output* starts with the average sectional values as shown in Table M.48.

**Table M.48** Output File: Member Section Average Values

MEMBER SECTION STRESSES AND STRAINS			
*****			
AVERAGE VALUES			
MEMBER : 5			
MOMENT	: 247.7 kN-m	CURVATURE	: 4.309 me/m
AXIAL LOAD:	0.6 kN	AXIAL STRAIN:	0.360 me
SHEAR	: -172.4 kN	SHEAR STRAIN:	-0.593 me

The *Output File* continues with the *Crack Conditions* as shown in Table M.49.

**Table M.49** Output File: Member Section Crack Conditions

CRACK CONDITIONS					
NC	WCR (mm)	SCR (mm)	VCI (MPa)	SLIP (mm)	STATE
1	0.00	0.0	0.00	0.00	0
2	0.00	0.0	0.00	0.00	0
3	0.00	0.0	0.00	0.00	0
4	0.00	0.0	0.00	0.00	0
5	0.00	0.0	0.00	0.00	0
.					
25	0.14	154.5	-0.01	-0.00	1
26	0.16	177.5	-0.01	-0.00	1
27	0.19	199.1	-0.00	-0.00	1
28	0.22	220.8	-0.00	-0.00	1
29	0.24	241.1	0.01	0.01	1

STATE 0: Layer uncracked  
 1: Tension Stiffening governs  
 2: Tension Softening governs  
 3: Reinf Reserve Capacity limited FC1  
 4: Crack Width Check limited FC2  
 5: VCI<sub>max</sub> limited FC1 (MCFT)

In Table M.49, *NC* is the concrete layer number from the top of the cross section, *WCR* is the average crack width, *SCR* is the average crack spacing, *VCI* is the interface shear stress on crack surface, *SLIP* is the crack shear slip of the DSFM (Vecchio, 2000) and *STATE* is the state of the concrete layer as defined in Table M.49.

The *Detailed Member Output* continues with the *Net Strains* as shown in Table M.50.

**Table M.50** Output File: Member Section Net Strains

NET STRAINS								
NC	TEMP (C)	ECT (me)	EX (me)	EY (me)	GXY (me)	E1 (me)	E2 (me)	THETA1 (Deg)
1	0.00	-0.567	-0.484	-0.000	-0.005	-0.000	-0.484	90.300
2	0.00	-0.541	-0.465	-0.000	-0.016	0.000	-0.466	91.002
3	0.00	-0.512	-0.444	-0.000	-0.028	0.000	-0.445	91.832

**Table M.50** Output File: Member Section Net Strains (Continued)

4	0.00	-0.483	-0.423	-0.000	-0.040	0.001	-0.424	92.723
5	0.00	-0.446	-0.394	-0.000	-0.055	0.001	-0.396	93.993
.								
.								
25	0.00	0.873	0.874	0.001	-0.044	0.875	-0.000	-1.442
26	0.00	0.912	0.913	0.000	-0.032	0.913	-0.000	-0.999
27	0.00	0.943	0.944	0.000	-0.022	0.944	-0.000	-0.666
28	0.00	0.975	0.975	0.000	-0.015	0.975	-0.000	-0.444
29	0.00	1.006	1.005	0.000	-0.035	1.006	-0.000	-0.999

In Table M.50, *NC* is the concrete layer number from the top of the cross section, *TEMP* is the temperature of the concrete layer with respect to the Reference Temperature (see Table M.39), *ECT* is the total strain, *EX*, *EY* and *GXY* are the net concrete strains, *E1* and *E2* are the net concrete principal strains, and *THETA1* is the inclination of the principal tensile stress field based on the MCFT (Vecchio and Collins, 1986) or the DSFM (Vecchio, 2000).

The *Detailed Member Output* continues with the *Concrete Stresses* as shown in Table M.51.

**Table M.51** Output File: Member Section Concrete Stresses

CONCRETE STRESSES							
NC	FCX (MPa)	FCY (MPa)	VC (MPa)	FC1 (MPa)	FC2 (MPa)	FC2/FP	BETA
1	-13.134	0.000	-0.069	-0.003	-13.134	0.580	1.00
2	-12.667	0.000	-0.222	0.001	-12.671	0.560	1.00
3	-12.119	0.000	-0.388	0.010	-12.131	0.536	1.00
4	-11.554	0.000	-0.551	0.024	-11.580	0.512	1.00
5	-10.798	-0.012	-0.757	0.041	-10.851	0.480	1.00
.							
.							
25	0.918	-0.000	-0.023	0.919	-0.010	0.002	0.20
26	0.909	0.000	-0.016	0.910	-0.010	0.002	0.20
27	0.903	0.000	-0.010	0.903	-0.010	0.002	0.20
28	0.897	0.000	-0.007	0.897	-0.010	0.002	0.20
29	0.891	0.000	-0.016	0.891	-0.010	0.002	0.20



In Table M.51, *NC* is the concrete layer number from the top of the cross section, *FCX* is the longitudinal axial stress of the concrete, *FCY* is the transverse stress of the concrete, *VCI* is the shear stress of the concrete, *FC1* and *FC2* are the principal stresses of the concrete, *FP* is the softened compressive strength of the concrete, *BETA* is the coefficient for the concrete compression softening.

The *Detailed Member Output* continues with the *Longitudinal Reinforcement Strains and Stresses* as shown in Table M.52.

**Table M.52** Output File: Member Section Longitudinal Reinforcement Strains and Stresses

LONGITUDINAL REINFORCEMENT STRAINS & STRESSES							
NS	DEPTH (mm)	TEMP (C)	AVERAGE		AT CRACK		FORCE (kN)
			STRAIN (me)	STRESS (MPa)	STRAIN (me)	STRESS (MPa)	
1	50.0	0.00	-0.434	-86.81	0.000	0.00	-26.04
2	424.0	0.00	0.650	143.05	0.674	140.47	143.05
3	488.0	0.00	0.836	167.15	0.894	186.34	234.01

In Table M.52, *NS* is the number of the steel layer from the top of the cross section, *DEPTH* is the location of the steel layer from the top of the cross section, *TEMP* is the temperature of the steel layer with respect to the *Reference Temperature* (see Table M.39).

The *Detailed Member Output* concluded with the *Transverse Reinforcement Strains and Stresses* as shown in Table M.53.

**Table M.53** Output File: Member Section Transverse Reinforcement Strains and Stresses

## TRANSVERSE REINFORCEMENT STRAINS &amp; STRESSES

NC	AVERAGE		AT CRACK	
	STRAIN (me)	STRESS (MPa)	STRAIN (me)	STRESS (MPa)
1	0.072	0.00	0.000	0.00
2	0.070	0.00	0.000	0.00
3	0.066	0.00	0.000	0.00
4	0.063	0.00	0.000	0.00
5	0.058	11.69	0.000	0.00
25	0.001	0.26	0.001	0.21
26	0.001	0.00	0.000	0.00
27	0.000	0.00	0.000	0.00
28	0.000	0.00	0.000	0.00
29	-0.000	0.00	0.000	0.00

**M4.2 Determination of Damage or Failure Modes**

As defined above, VecTor5 provides ample output including analysis results for nodes, members and concrete and steel layers. The output is especially useful when evaluating the dominant mechanism and the damage or failure mode of the structure. To illustrate the interpretation of the damage mode, consider the analysis result for Beam VS-A1, introduced in Section M3.1, at a midspan displacement of 25 mm where the beam load capacity dropped significantly (see Figure 4.10 of Guner (2008)). It is advisable to first inspect the member deformations to determine the critical members.

**Table M.54** Output for Member Deformations at Load Stage 51 (Beam VS-A1)

MEMBER DEFORMATIONS							
*****							
M	ECL (me)	GXY (me)	PHI (me/m)	ESL-MAX (me)	ESL-MIN (me)	EST-MAX (me)	WCR-MAX (mm)
1	-0.003	-0.07	0.212	0.042	-0.051	0.009	
2	0.21	-1.028	1.612	0.551	-0.155	0.951	0.49
3	0.299	-1.297	2.697	0.87	-0.311	1.304	0.65
4	0.369	-1.429	3.882	1.192	-0.508	1.561	0.77
<b>5</b>	0.446	<b>-2.366</b>	5.529	1.619	-0.803	3.672	1.62
<b>6</b>	0.789	-1.288	<b>35.628</b>	8.342	-7.262	2.446	2.55

Inspection of Table M.54 reveals that Member 5 is the most critical member in terms of shear deformations; Member 6 is the most critical member in terms of flexural deformations. To investigate the conditions of these members, the detailed member output should be utilized. First, consider some of the detailed member output calculated for Member 5 as presented in Table M.55.

**Table M.55** Detailed Output for Concrete Layers of Member 5 (Beam VS-A1)

NC	WCR (mm)	SLIP (mm)	FCX (MPa)	FCY (MPa)	VC (MPa)	FC1 (MPa)	FC2 (MPa)	FC2/FP (MPa)	BETA	E1 (x10 <sup>-3</sup> )	E2 (x10 <sup>-3</sup> )	THETA1 (Deg)
1	0	0.0	-15.05	0.00	-0.64	0.02	-15.07	0.7	1.0	0.001	-0.596	-87.6
2	0	0.0	-14.84	0.00	-2.09	0.29	-15.13	0.7	1.0	0.01	-0.586	-82.1
3	0.69	0.0	-12.29	0.00	-0.94	0.01	-12.36	0.7	0.8	1.484	-0.474	-85.6
4	1.48	-0.3	-9.41	0.00	-1.23	0.01	-9.57	0.7	0.6	3.599	-0.425	-82.6
5	0.4	0.0	-12.91	-0.22	-2.35	0.20	-13.34	0.7	0.8	1.099	-0.517	-79.8
6	0.6	0.0	-12.04	-0.32	-2.28	0.11	-12.47	0.7	0.8	1.624	-0.479	-79.4
7	1.09	0.0	-10.44	-0.24	-1.66	0.02	-10.70	0.7	0.7	2.693	-0.422	-81.0
8	1.16	0.0	-9.92	-0.27	-1.80	0.05	-10.25	0.7	0.6	2.842	-0.396	-79.8
9	1.19	0.0	-9.39	-0.31	-1.95	0.09	-9.79	0.7	0.6	2.91	-0.372	-78.4
10	1.51	0.0	-8.71	-0.25	-1.68	0.08	-9.04	0.7	0.5	3.536	-0.334	-79.2
11	1.57	0.0	-7.74	-0.48	-1.94	0.01	-8.23	0.7	0.5	3.456	-0.301	-75.9
12	1.62	0.0	-6.89	-0.48	-1.84	0.01	-7.38	0.7	0.5	3.596	-0.266	-75.1
13	1.62	0.0	-6.03	-0.49	-1.73	0.01	-6.52	0.7	0.4	3.636	-0.234	-74.0
14	1.58	0.0	-5.14	-0.49	-1.61	0.01	-5.65	0.6	0.4	3.584	-0.201	-72.6
15	1.5	0.0	-4.27	-0.50	-1.48	0.01	-4.79	0.6	0.4	3.452	-0.17	-70.9
16	1.25	-0.4	-2.24	-0.54	-1.48	0.31	-3.09	0.5	0.3	3.093	-0.11	-59.9
17	1.02	-0.3	-1.89	-0.47	-1.59	0.56	-2.92	0.4	0.3	2.822	-0.103	-57.1
18	0.9	-0.2	-1.44	-0.42	-1.44	0.60	-2.48	0.4	0.3	2.649	-0.087	-54.7
19	0.79	-0.2	-1.01	-0.37	-1.28	0.63	-2.02	0.4	0.3	2.488	-0.071	-52.1
20	0.68	-0.2	-0.64	-0.32	-1.12	0.65	-1.61	0.3	0.2	2.337	-0.057	-49.1
21	0.59	-0.1	-0.33	-0.27	-0.96	0.66	-1.27	0.3	0.2	2.202	-0.045	-45.9
22	0.49	-0.1	-0.08	-0.23	-0.81	0.66	-0.97	0.2	0.2	2.083	-0.035	-42.4
23	0.41	-0.1	0.12	-0.19	-0.68	0.66	-0.73	0.2	0.2	1.98	-0.026	-38.6
24	0.32	-0.1	0.27	-0.15	-0.56	0.65	-0.54	0.1	0.2	1.894	-0.019	-34.6
25	0.26	-0.1	0.38	-0.12	-0.45	0.64	-0.38	0.1	0.2	1.826	-0.014	-30.3
26	0.24	0.0	0.46	-0.09	-0.36	0.63	-0.27	0.1	0.2	1.779	-0.009	-26.1
27	0.28	0.0	0.51	-0.07	-0.28	0.62	-0.18	0.0	0.2	1.753	-0.006	-21.9
28	0.28	0.0	0.54	-0.05	-0.21	0.61	-0.11	0.0	0.2	1.745	-0.004	-17.9
29	0.27	0.0	0.54	-0.03	-0.15	0.58	-0.07	0.0	0.2	1.754	-0.002	-14.1
30	0.27	0.0	0.19	-0.02	-0.04	0.19	-0.03	0.0	0.2	1.782	-0.001	-11.5
31	0.31	0.0	0.01	0.00	0.00	0.01	-0.01	0.0	0.2	1.816	0.000	-9.5
32	0.36	0.0	0.01	0.00	0.00	0.01	-0.01	0.0	0.2	1.85	0.000	-6.9
33	0.41	0.0	0.01	0.00	0.00	0.01	-0.01	0.0	0.2	1.893	0.000	-4.1
34	0.47	0.0	0.01	0.00	0.00	0.01	-0.01	0.0	0.2	1.944	0.000	-1.3

In Table M.55, a crack width of 0.47 mm is calculated for the extreme tension layer (Layer 34) with almost zero degree angle from the vertical plane indicating that this is a flexural crack. While extending towards the top of the cross section (i.e., compression

zone), the crack widens significantly and takes a diagonal form reaching a maximum width of approximately 1.6 mm, indicating shear cracking. There is also an almost horizontal splitting crack occurring at layer 4. The  $FC2/FP$  values indicate that the concrete layers in the compression zone have reached 70% of their peak strengths, indicating that concrete compression crushing or failure has not yet occurred but may occur in the following load stages. Strain in the tension reinforcement is approximately  $1.6 \times 10^{-3}$  (not shown on Table M.55) indicating that the member is far from reaching the reinforcement rupture strain of  $175 \times 10^{-3}$ . As a result, a damage mode of significant diagonal cracking (i.e., shear distress) can be concluded for this member.

Consider now some of the detailed member output calculated for Member 6 as shown in Table M.56.

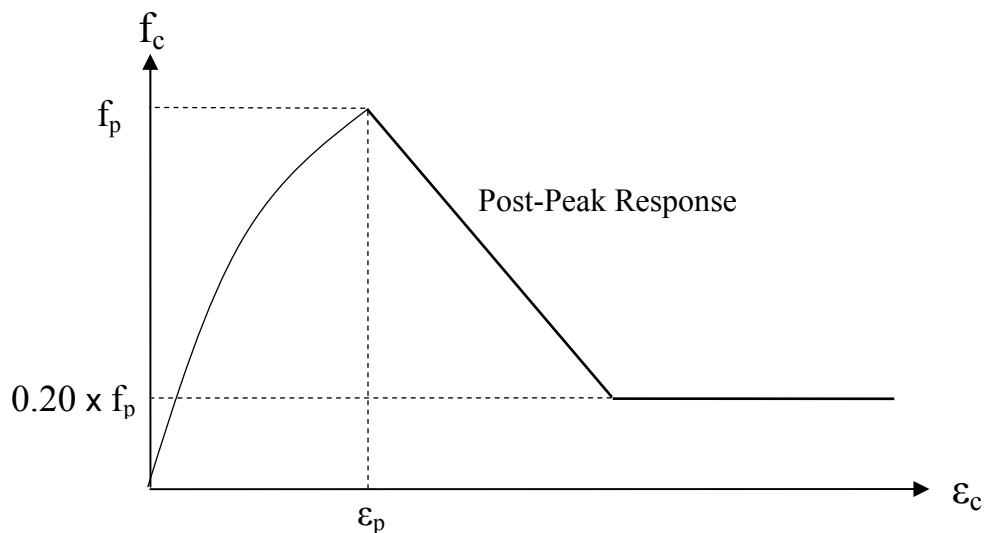
**Table M.56** Detailed Output for Concrete Layers of Member 6 (Beam VS-A1)

NC (mm)	WCR (mm)	SLIP	STATE	FCX (MPa)	FCY (MPa)	VC (MPa)	FC1 (MPa)	FC2 (MPa)	FC2/FP (MPa)	BETA	E1 ( $\times 10^{-3}$ )	E2 (Deg)	THETA1
1	0	0.0	0	-4.52	0.00	0.00	-0.04	-4.52	0.20	1.00	0.00	-1.77	-89.982
2	0	0.0	0	-4.52	0.00	-0.01	-0.05	-4.52	0.20	1.00	0.00	-1.77	-89.936
3	0	0.0	0	-4.52	0.00	-0.01	-0.07	-4.52	0.20	1.00	0.00	-1.77	-89.873
4	0	0.0	0	-4.52	0.00	-0.02	-0.07	-4.52	0.20	1.00	0.00	-1.76	-89.798
5	0	0.0	0	-7.95	-0.19	-0.05	-0.28	-7.95	0.34	1.00	-0.01	-1.88	-89.627
6	0	0.0	0	-10.15	-0.19	-0.10	-0.35	-10.15	0.42	1.00	-0.01	-1.90	-89.418
7	0	0.0	0	-11.61	-0.18	-0.17	-0.36	-11.61	0.49	1.00	-0.01	-1.89	-89.147
8	0.00	0.0	1	-7.71	-0.16	-0.16	-0.30	-7.71	0.34	1.00	-0.01	-1.62	-88.801
9	0.00	0.0	1	-9.74	-0.15	-0.34	-0.28	-9.75	0.43	1.00	-0.01	-1.58	-87.976
10	0.00	0.0	1	-11.82	-0.14	-0.67	-0.18	-11.86	0.52	1.00	0.00	-1.53	-86.719
11	0.00	0.0	1	-14.02	-0.14	-1.33	-0.02	-14.14	0.63	1.00	0.00	-1.48	-84.579
12	0.23	0.0	1	-17.20	-0.22	-2.00	0.01	-17.43	0.77	1.00	0.46	-1.45	-83.366
13	0.49	0.0	1	-19.48	-0.32	-2.58	0.03	-19.82	0.91	0.96	1.00	-1.34	-82.459
14	0.74	0.0	1	-19.54	-0.39	-2.96	0.06	-19.99	1.00	0.89	1.52	-1.14	-81.405
15	0.94	0.1	1	-16.92	-0.40	-3.07	0.15	-17.47	0.98	0.79	1.99	-0.84	-79.801
16	1.14	-0.1	3	-8.77	-0.49	-2.35	0.13	-9.39	0.73	0.57	2.52	-0.41	-75.183
17	0.47	-0.1	3	-0.54	-0.19	-0.78	0.43	-1.16	0.20	0.25	1.33	-0.04	-51.516
18	0.46	0.0	1	-0.02	-0.08	-0.06	0.02	-0.12	0.03	0.20	1.43	0.00	-30.77
19	0.6	0.0	1	0.01	-0.05	-0.02	0.02	-0.06	0.01	0.20	1.94	0.00	-19.86
20	0.73	0.0	1	0.01	-0.03	-0.01	0.01	-0.04	0.01	0.20	2.54	0.00	-14.20
21	0.83	0.0	1	0.01	-0.02	-0.01	0.01	-0.03	0.01	0.20	3.16	0.00	-10.78
22	0.88	0.0	1	0.01	-0.02	0.00	0.01	-0.02	0.00	0.20	3.80	0.00	-8.47
23	0.89	0.0	1	0.01	-0.01	0.00	0.01	-0.02	0.00	0.20	4.45	0.00	-6.79
24	0.85	0.0	1	0.01	-0.01	0.00	0.01	-0.01	0.00	0.20	5.10	0.00	-5.50
25	0.78	0.0	1	0.01	-0.01	0.00	0.01	-0.01	0.00	0.20	5.75	0.00	-4.46
26	0.87	0.0	1	0.01	-0.01	0.00	0.01	-0.01	0.00	0.20	6.41	0.00	-3.56
27	1.17	0.0	1	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	7.07	0.00	-2.71
28	1.25	0.0	1	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	7.68	0.00	-2.40
29	1.28	0.0	3	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	8.25	0.00	-1.91
30	1.37	0.0	3	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	8.81	0.00	-1.41
31	1.66	0.0	2	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	9.29	0.00	-1.01
32	1.94	0.0	2	0.01	0.00	0.00	0.01	-0.01	0.00	0.20	9.67	0.00	-0.71
33	2.23	0.0	2	0.01	0.00	0.00	0.01	0.00	0.00	0.20	10.05	0.00	-0.42
34	2.55	0.0	2	0.01	0.00	0.00	0.01	0.00	0.00	0.20	10.43	0.00	-0.14

In Table M.56, the maximum crack width of 2.55 mm is calculated for the extreme tension layer (Layer 34) with almost zero degree angle from the vertical plane indicating a flexural cracking. Diagonal shear cracking is calculated for Layer 16 with a crack width of 1.14 mm. Compared to the flexural cracking, the shear cracking is not significant.

The  $FC2/FP$  values indicate that concrete layers in the compression zone sustain 20% of their peak strengths.  $BETA$  values of 1.00 indicate that the sustained stress values correspond to the maximum strength of the concrete, thereby indicating compression crushing for those layers. In the default concrete post-peak base curve of Modified Park-Kent model (Park, Kent and Gill, 1982), there is  $0.20 \times f'_c$  residual stress present as shown in Figure M.23. Therefore, the stress values of  $0.20 \times f'_c$  ( $0.20 \times 22.6 = 4.52$  MPa for Beam VS-A1) indicate concrete crushing if they are calculated in the post-peak regime. More details on this model can be found in Wong and Vecchio (2002). The strain in the tension reinforcement is approximately  $8.3 \times 10^{-3}$  indicating that the member is far from reaching the reinforcement rupture strain of  $175 \times 10^{-3}$ .

Note that  $FC2/FP$  and  $BETA$  values reported for the tension layers in Table M.55 and Table M.56 do not have any significance in the interpretation of damage or failure modes.



**Figure M.23** Concrete Post-Peak Response: Modified Park-Kent (Park, Kent and Gill, 1982)

As a result, the damage mode of Beam VS-A1 can clearly be interpreted as shear-compression. If there were no crushing of the concrete in Member 6, the damage mode would be diagonal-tension. If there were no significant diagonal shear cracking in Member 5, the damage mode would be flexure-compression.

In addition, fracture of the transverse and longitudinal reinforcement is considered by VecTor5. If such a failure occurs, the load capacity of the structure suddenly drops noticeably in most cases; the ruptured reinforcement and the member it belongs to are written clearly in the *Output Files*.

## REFERENCES

---

Guner, S. (2008), "Performance Assessment of Shear-Critical Reinforced Concrete Plane Frames," PhD Thesis, Department of Civil Engineering, University of Toronto, 429 pp. (available in the Publications section of [www.civ.utoronto.ca/vector/](http://www.civ.utoronto.ca/vector/) under Theses)

Kent, D.C., and Park, R. (1971), "Flexural Members with Confined Concrete," ASCE Journal of the Structural Division, V. 97, No. ST7, Proc. Paper 8243, pp. 1341-1360.

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.

Wong, P. S. and Vecchio, F. J. (2002), "VecTor2 and FormWorks User's Manual," Technical Report, Department of Civil Engineering, University of Toronto, 217 p. (available in the Publications section of [www.civ.utoronto.ca/vector/](http://www.civ.utoronto.ca/vector/) under User Manuals)

## APPENDIX M1

### EXAMPLE STATIC ANALYSIS (MONOTONIC LOADING)

As an example application, the input data files of the Duong frame (Duong et al., 2007) are presented in Table M.57 to M.61. In addition to the *Structure*, *Job* and *Auxiliary Data Files*, two *Load Cases (Load Data Files)* are included in this analysis. Details of the structural model and loading are presented in Section 4.8 of Guner (2008).

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame

```

* * * * *
*
*           V e c T o r 5 D           *
*
*           S T R U C T U R E   D A T A           *
*
* * * * *

GENERAL PARAMETERS
=====

Structure Title           (30 char. max.) : KVD
Structure File Name      ( 8 char. max.) : KVD
No. of Members           (1200 max.) : 76
No. of Member Types      ( 50 max.) : 12
No. of Nodes             : 75
No. of Support Nodes     : 6
No. of Support Restraints : 12

STRUCTURE PARAMETERS
=====

(A) Nodal Coordinates
-----
<NOTE:> Coordinate units in mm
<<<<< FORMAT >>>>>
NODE  X      Y  [ #NODES  d(NODE)  d(X)  d(Y)  ] <-- up to 2 dir. /
1     0      0   /
2     350    0   /
3     650    0   /
4     900    0   /
5     1100   0   /
6     1300   0   /
7     1500   0   /
8     1750   0   /
9     2050   0   /
10    2350   0   /
11    2600   0   /

```



**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

12	2800	0	/
13	3000	0	/
14	3200	0	/
15	3450	0	/
16	3750	0	/
17	4100	0	/
18	1100	200	/
19	1100	450	/
20	1100	700	/
21	1100	950	/
22	1100	1210	/
23	1100	1440	/
24	1100	1670	/
25	1100	1900	/
26	1100	2100	/
27	1100	2300	/
28	1100	2500	/
29	1100	2750	/
30	1100	3000	/
31	1100	3250	/
32	1100	3500	/
33	1100	3750	/
34	1100	4000	/
35	1100	4200	/
36	3000	200	/
37	3000	450	/
38	3000	700	/
39	3000	950	/
40	3000	1210	/
41	3000	1440	/
42	3000	1670	/
43	3000	1900	/
44	3000	2100	/
45	3000	2300	/
46	3000	2500	/
47	3000	2750	/
48	3000	3000	/
49	3000	3250	/
50	3000	3500	/
51	3000	3750	/
52	3000	4000	/
53	3000	4200	/
54	1300	2100	/
55	1450	2100	/
56	1600	2100	/
57	1750	2100	/
58	1900	2100	/
59	2050	2100	/
60	2200	2100	/
61	2350	2100	/
62	2500	2100	/
63	2650	2100	/
64	2800	2100	/
65	1300	4200	/
66	1450	4200	/
67	1600	4200	/

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

```

68    1750  4200  /
69    1900  4200  /
70    2050  4200  /
71    2200  4200  /
72    2350  4200  /
73    2500  4200  /
74    2650  4200  /
75    2800  4200  /
/

```

(B) Member Indices

```

-----
<<<<< FORMAT >>>>>
MEM  INC1  INC2  MEM.TYPE  [ #MEMS  d(MEM)  d(INC)  ]<--up to 2 dir./
1     1     2     6           3       1       1       /
4     4     5     12          2       1       1       /
6     6     7     6           6       1       1       /
12    12    13    12          2       1       1       /
14    14    15    6           3       1       1       /
17    5     18    10          1       1       1       /
18    18    19    4           4       1       1       /
22    22    23    2           3       1       1       /
25    25    26    8           2       1       1       /
27    27    28    2           3       1       1       /
30    30    31    5           4       1       1       /
34    34    35    11          1       1       1       /
35    13    36    10          1       1       1       /
36    36    37    4           4       1       1       /
40    40    41    2           3       1       1       /
43    43    44    8           2       1       1       /
45    45    46    2           3       1       1       /
48    48    49    3           4       1       1       /
52    52    53    9           1       1       1       /
53    26    54    7           1       1       1       /
54    54    55    1          10      1       1       /
64    64    44    7           1       1       1       /
65    35    65    7           1       1       1       /
66    65    66    1          10      1       1       /
76    75    53    7           1       1       1       /
/

```

(C) Support Restraint List

```

-----
<<<<< FORMAT >>>>>
NODE  X-RST  Y-RST  Z-RST  [ #NODES  d(NODE)  ] <-- up to 2 dir. /
1      1      1      0/
2      1      1      0/
7      1      1      0/
11     1      1      0/
16     1      1      0/
17     1      1      0/
/

```

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

(D) Member Specifications

MT	f'c [f't (MPa) (MPa)	Ec (MPa)	e0 (me)	Mu	Cc (/deg.C)	Kc (mm2/hr)	Agg (mm)	Dens] (kg/m3)	[Smx (mm)	Smy] (mm)
1	42.9 0	30058	2.31	0	0	4320	0	0	0	0
2	42.9 0	30058	2.31	0	0	4320	0	0	0	0
3	42.9 0	30058	2.31	0	0	4320	0	0	0	0
4	42.9 0	30058	2.31	0	0	4320	0	0	0	0
5	42.9 0	30058	2.31	0	0	4320	0	0	0	0
6	42.9 0	30058	2.31	0	0	4320	0	0	0	0
7	42.9 0	30058	2.31	0	0	4320	0	0	0	0
8	42.9 0	30058	2.31	0	0	4320	0	0	0	0
9	42.9 0	30058	2.31	0	0	4320	0	0	0	0
10	42.9 0	30058	2.31	0	0	4320	0	0	0	0
11	42.9 0	30058	2.31	0	0	4320	0	0	0	0
12	42.9 0	30058	2.31	0	0	4320	0	0	0	0
/										

MT	Nc (#)	Ns (#)	Fyz (MPa)	St (mm)	Dbt (mm)	Fyt (MPa)	Fut (MPa)	Est (MPa)	Esht (MPa)	esht (me)	[ Cs ] (/deg.C)	Ref. Typ
1	30	2	506	300	9.5	506	615	210000	1025	28.3	0.00001	1
2	30	2	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
3	30	3	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
4	30	4	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
5	30	3	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
6	30	2	455	175	11.3	455	583	192400	1195	22.8	0.00001	1
7	30	2	506	300	9.5	506	615	210000	1025	28.3	0.00001	1
8	30	2	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
9	30	3	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
10	30	4	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
11	30	3	455	130	11.3	455	583	192400	1195	22.8	0.00001	1
12	30	2	455	175	11.3	455	583	192400	1195	22.8	0.00001	1
/												

(E) Concrete Layers

MT	Dc (mm)	Wc (mm)	Rho-t (%)	Rho-z (%)	Nx
1	10	300	0.0	0.237	3
1	10	300	0.158	0.237	2
1	10	300	0.158	0.237	5
1	20	300	0.158	0	10
1	10	300	0.158	0.237	5
1	10	300	0.158	0.237	2
1	10	300	0.0	0.237	3
2	10	300	0.0	1.154	2
2	10	300	1.026	1.154	2
2	10	300	1.026	1.154	6
2	20	300	1.026	0	10
2	10	300	1.026	1.154	6
2	10	300	1.026	1.154	2
2	10	300	0.0	1.154	2
3	10	300	0.0	1.154	2

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

3	10	300	1.026	1.154	2
3	10	300	1.026	1.154	6
3	20	300	1.026	0	10
3	10	300	1.026	1.154	6
3	10	300	1.026	1.154	2
3	10	300	0.0	1.154	2
4	10	300	0.0	1.154	2
4	10	300	1.026	1.154	2
4	10	300	1.026	1.154	6
4	20	300	1.026	0	10
4	10	300	1.026	1.154	6
4	10	300	1.026	1.154	2
4	10	300	0.0	1.154	2
5	10	300	0.0	1.154	2
5	10	300	1.026	1.154	2
5	10	300	1.026	1.154	6
5	20	300	1.026	0	10
5	10	300	1.026	1.154	6
5	10	300	1.026	1.154	2
5	10	300	0.0	1.154	2
6	10	800	0.0	1.039	3
6	10	800	0.429	1.039	2
6	10	800	0.429	1.039	6
6	22.5	800	0.429	0	8
6	10	800	0.429	1.039	6
6	10	800	0.429	1.039	2
6	10	800	0.0	1.039	3
7	10	300	0.0	0.474	3
7	10	300	0.316	0.474	2
7	10	300	0.316	0.474	5
7	20	300	0.316	0	10
7	10	300	0.316	0.474	5
7	10	300	0.316	0.474	2
7	10	300	0.0	0.474	3
8	10	300	0.0	2.308	2
8	10	300	2.052	2.308	2
8	10	300	2.052	2.308	6
8	20	300	2.052	0	10
8	10	300	2.052	2.308	6
8	10	300	2.052	2.308	2
8	10	300	0.0	2.308	2
9	10	300	0.0	2.308	2
9	10	300	2.052	2.308	2
9	10	300	2.052	2.308	6
9	20	300	2.052	0	10
9	10	300	2.052	2.308	6
9	10	300	2.052	2.308	2
9	10	300	0.0	2.308	2
10	10	300	0.0	2.308	2
10	10	300	2.052	2.308	2
10	10	300	2.052	2.308	6
10	20	300	2.052	0	10
10	10	300	2.052	2.308	6
10	10	300	2.052	2.308	2
10	10	300	0.0	2.308	2
11	10	300	0.0	2.308	2

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

11	10	300	2.052	2.308	2
11	10	300	2.052	2.308	6
11	20	300	2.052	0	10
11	10	300	2.052	2.308	6
11	10	300	2.052	2.308	2
11	10	300	0.0	2.308	2
12	10	800	0.0	2.078	3
12	10	800	0.858	2.078	2
12	10	800	0.858	2.078	6
12	22.5	800	0.858	0	8
12	10	800	0.858	2.078	6
12	10	800	0.858	2.078	2
12	10	800	0.0	2.078	3
/					

(F) Rebar Layers

MT	N	Ys	As	Db	Fy	Fu	Es	Esh	esh	Dep
		(mm)	(mm <sup>2</sup> )	(mm)	(MPa)	(MPa)	(MPa)	(MPa)	(me)	(me)
1	1	50	1200	19.5	447	603	198400	1372	17.1	0
1	2	350	1200	19.5	447	603	198400	1372	17.1	0
2	1	40	1200	19.5	447	603	198400	1372	17.1	0
2	2	360	1200	19.5	447	603	198400	1372	17.1	0
3	1	40	600	19.5	447	603	198400	1372	17.1	0
3	2	90	600	19.5	447	603	198400	1372	17.1	0
3	3	360	1200	19.5	447	603	198400	1372	17.1	0
4	1	40	600	19.5	447	603	198400	1372	17.1	0
4	2	90	600	19.5	447	603	198400	1372	17.1	0
4	3	310	600	19.5	447	603	198400	1372	17.1	0
4	4	360	600	19.5	447	603	198400	1372	17.1	0
5	1	40	1200	19.5	447	603	198400	1372	17.1	0
5	2	310	600	19.5	447	603	198400	1372	17.1	0
5	3	360	600	19.5	447	603	198400	1372	17.1	0
6	1	50	2400	19.5	447	603	198400	1372	17.1	0
6	2	350	2400	19.5	447	603	198400	1372	17.1	0
7	1	50	2400	19.5	447	603	198400	1372	17.1	0
7	2	350	2400	19.5	447	603	198400	1372	17.1	0
8	1	40	2400	19.5	447	603	198400	1372	17.1	0
8	2	360	2400	19.5	447	603	198400	1372	17.1	0
9	1	40	1200	19.5	447	603	198400	1372	17.1	0
9	2	90	1200	19.5	447	603	198400	1372	17.1	0
9	3	360	2400	19.5	447	603	198400	1372	17.1	0
10	1	40	1200	19.5	447	603	198400	1372	17.1	0
10	2	90	1200	19.5	447	603	198400	1372	17.1	0
10	3	310	1200	19.5	447	603	198400	1372	17.1	0
10	4	360	1200	19.5	447	603	198400	1372	17.1	0
11	1	40	2400	19.5	447	603	198400	1372	17.1	0
11	2	310	1200	19.5	447	603	198400	1372	17.1	0
11	3	360	1200	19.5	447	603	198400	1372	17.1	0
12	1	50	4800	19.5	447	603	198400	1372	17.1	0
12	2	350	4800	19.5	447	603	198400	1372	17.1	0
/										

**Table M.57** Structure Data File for VecTor5 Analysis of Duong Frame (Continued)

```

(G) Detailed Member Output List
=====
<<<<< FORMAT >>>>>
MEM [ #MEMS d(MEM) ] <-- up to 2 directions /
54/
55/
62/
63/
/

```

**Table M.58** Load Data File for VecTor5 Analysis of Duong Frame (Case: HL)

```

* * * * *
*   V e c T o r 5 D   *
*   L O A D   D A T A   *
* * * * *

LOAD CASE PARAMETERS
*****

Load Case ID          (15 char. max.)      : HL
Load Case Data File   (8 char. max.)       : HL
Load Factored         (0 - 1)              : 1
Time Factored         (0 - 1)              : 0

No. of Loaded Nodes                                     : 0
No. of Members w/ End Action Loads                   : 0
No. of Members w/ Concentrated Loads                 : 0
No. of Members w/ Distributed Loads                  : 0
No. of Members w/ Gravity Loads                      : 0
No. of Members w/ Temperature Loads                 : 0
No. of Members w/ Concrete Prestrain                : 0
No. of Members w/ Support Displacements              : 1
No. of Nodes w/ Lumped Mass Assignments             : 0
No. of Nodes w/ Impulse Loads                       : 0
No. of Ground Acceleration Data                     : 0

NODAL LOADS
*****

<NOTE:> UNITS: kN,kN-m
<<<<< FORMAT >>>>>
NODE Fx Fy Mz [ #NODE d(NODE) d(Fx) d(Fy) d(Mz) ]<-- (2)/
/

MEMBER END ACTIONS
*****

<NOTE:> UNITS: kN,kN-m
<<<<< FORMAT >>>>>
M AF1 SF1 BM1 AF2 SF2 BM2 [ #M d(M) ]<-- (2)/
/

```

**Table M.58** Load Data File for VecTor5 Analysis of Duong Frame (Case: HL) (Continued)

CONCENTRATED LOADS  
\*\*\*\*\*

<NOTE:> UNITS: kN, kN-m, m  
<<<<< FORMAT >>>>>  
M Fx Fy Mz x/L [ #M d(M) d(Fx) d(Fy) d(Mz) ]<-- (2)/  
/

UNIFORMLY DISTRIBUTED LOADS  
\*\*\*\*\*

<NOTE:> UNITS: kN/m, m  
<<<<< FORMAT >>>>>  
M W a/L b/L [ #M d(M) d(W) ]<-- (2)/  
/

GRAVITY LOADS  
\*\*\*\*\*

<NOTE:>  
<<<<< FORMAT >>>>>  
M GX GY [ #M d(M) ]<-- (2)/  
/

TEMPERATURE LOADS  
\*\*\*\*\*

<NOTE:> UNITS: Deg.C, hrs  
<<<<< FORMAT >>>>>  
M T1' T2' T1 T2 TIME [ #M d(M) ]<-- (2)/  
/

CONCRETE PRESTRAINS  
\*\*\*\*\*

<NOTE:> UNITS: me  
<<<<< FORMAT >>>>>  
M STRAIN [ #ELMT d(ELMT) d(STRAIN) ]<-- (2)/  
/

PRESCRIBED NODAL DISPLACEMENTS  
\*\*\*\*\*

<NOTE:> UNITS: mm, rad  
<<<<< FORMAT >>>>>  
Jnt DOF DISPL [ #Jnt d(Jnt) ]/  
35 1 1.0/  
/

ADDITIONAL LUMPED MASSES  
\*\*\*\*\*

<NOTE:> UNITS: kg, m/s  
<<<<< FORMAT >>>>>  
NODE DOF-X DOF-Y MASS Vo-X Vo-Y Acc-X Acc-Y [ #NODE d(NODE) ] /  
/

IMPULSE, BLAST AND IMPACT FORCES  
\*\*\*\*\*

<NOTE:> UNITS: Sec, kN  
<<<<< FORMAT >>>>>  
Jnt DOF T1 F1 T2 F2 T3 F3 T4 F4 [ #NODE d(NODE) ] /  
/

**Table M.58** Load Data File for VecTor5 Analysis of Duong Frame (Case: HL) (Continued)

```

GROUND ACCELERATION
*****
<NOTE:> UNITS:  Sec, G
<<<<< FORMAT >>>>>
TIME ACC-X ACC-Y/
/

```

**Table M.59** Load Data File for VecTor5 Analysis of Duong Frame (Case: VL)

```

* * * * *
*   V e c T o r 5 D   *
*   L O A D   D A T A   *
* * * * *

LOAD CASE PARAMETERS
*****

Load Case ID          (15 char. max.)      : VL
Load Case Data File   (8 char. max.)       : VL
Load Factored         (0 - 1)              : 1
Time Factored         (0 - 1)              : 0

No. of Loaded Nodes          : 2
No. of Members w/ End Action Loads      : 0
No. of Members w/ Concentrated Loads    : 0
No. of Members w/ Distributed Loads     : 0
No. of Members w/ Gravity Loads        : 0
No. of Members w/ Temperature Loads    : 0
No. of Members w/ Concrete Prestrain   : 0
No. of Members w/ Support Displacements : 0
No. of Nodes w/ Lumped Mass Assignments : 0
No. of Nodes w/ Impulse Loads          : 0
No. of Ground Acceleration Data        : 0

```

```

NODAL LOADS
*****

<NOTE:> UNITS:  kN,kN-m
<<<<< FORMAT >>>>>
NODE Fx   Fy   Mz [ #NODE d(NODE) d(Fx) d(Fy) d(Mz) ]<-- (2)/
 35  0  -420.0  0/
 53  0  -420.0  0/
/

```

```

MEMBER END ACTIONS
*****

<NOTE:> UNITS:  kN,kN-m
<<<<< FORMAT >>>>>
M AF1 SF1 BM1 AF2 SF2 BM2 [ #M d(M) ]<-- (2)/
/

```

```

CONCENTRATED LOADS
*****

<NOTE:> UNITS:  kN,kN-m,m
<<<<< FORMAT >>>>>
M Fx Fy Mz x/L [ #M d(M) d(Fx) d(Fy) d(Mz) ]<-- (2)/
/

```



**Table M.59** Load Data File for VecTor5 Analysis of Duong Frame (Case: VL) (Continued)

UNIFORMLY DISTRIBUTED LOADS  
\*\*\*\*\*

<NOTE:> UNITS: kN/m,m  
<<<<< FORMAT >>>>>  
M W a/L b/L [ #M d(M) d(W) ]<-- (2)/  
/

GRAVITY LOADS  
\*\*\*\*\*

<NOTE:>  
<<<<< FORMAT >>>>>  
M GX GY [ #M d(M) ]<-- (2)/  
/

TEMPERATURE LOADS  
\*\*\*\*\*

<NOTE:> UNITS: Deg.C,hrs  
<<<<< FORMAT >>>>>  
M T1' T2' T1 T2 TIME [ #M d(M) ]<-- (2)/  
/

CONCRETE PRESTRAINS  
\*\*\*\*\*

<NOTE:> UNITS: me  
<<<<< FORMAT >>>>>  
M STRAIN [ #ELMT d(ELMT) d(STRAIN) ]<-- (2)/  
/

PRESCRIBED NODAL DISPLACEMENTS  
\*\*\*\*\*

<NOTE:> UNITS: mm,rad  
<<<<< FORMAT >>>>>  
Jnt DOF DISPL [ #Jnt d(Jnt) ]/  
/

ADDITIONAL LUMPED MASSES  
\*\*\*\*\*

<NOTE:> UNITS: kg, m/s  
<<<<< FORMAT >>>>>  
NODE DOF-X DOF-Y MASS Vo-X Vo-Y Acc-X Acc-Y [ #NODE d(NODE) ] /  
/

IMPULSE, BLAST AND IMPACT FORCES  
\*\*\*\*\*

<NOTE:> UNITS: Sec, kN  
<<<<< FORMAT >>>>>  
Jnt DOF T1 F1 T2 F2 T3 F3 T4 F4 [ #NODE d(NODE) ] /  
/

GROUND ACCELERATION  
\*\*\*\*\*

<NOTE:> UNITS: Sec, G  
<<<<< FORMAT >>>>>  
TIME ACC-X ACC-Y/  
/

**Table M.60** Job Data File for VecTor5 Analysis of Duong Frame

```

* * * * *
*   V e c T o r   *
*   J O B   D A T A   *
* * * * *

```

```

Job Title           (30 char max) : KVD
Job File Name      ( 8 char max) : KVD
Date               (30 char max) : Feb 2006

```

STRUCTURE DATA

-----

```

Structure Type           : 5
File Name                (8 char max) : KVD

```

LOADING DATA

-----

```

No. of Load Stages           : 500
Starting Load Stage No.     : 1
Load Series ID              (5 char max) : KVD

```

Load Case	File Name (8 char max)	Initial	Final	LS-Inc	Factors Type	Reps	C-Inc
1	HL	0.000	1000.000	2.000	1	1	0.000
2	VL	1.000	1.000	0.000	1	1	0.000
3	NULL	0.000	0.000	0.000	1	1	0.000
4	NULL	0.000	0.000	0.000	1	1	0.000
5	NULL	0.000	0.000	0.000	1	1	0.000

ANALYSIS PARAMETERS

-----

```

Analysis Mode           (1-2) : 1
Seed File Name         (8 char max) : NULL
Convergence Limit      (>1.0) : 1.00001
Averaging Factor       (<1.0) : 0.
Maximum Iterations     : 100
Convergence Criteria   : 2
Results Files          : 2
Output Format           : 1

```

MATERIAL BEHAVIOUR MODELS

-----

```

Concrete Compression Base Curve (0-3) : 2
Concrete Compression Post-Peak (0-3) : 1
Concrete Compression Softening (0-8) : 1
Concrete Tension Stiffening (0-5) : 1
Concrete Tension Softening (0-3) : 1
Concrete Tension Splitting (1-2) : 1
Concrete Confined Strength (0-2) : 1
Concrete Dilatation (0-1) : 1
Concrete Cracking Criterion (0-4) : 1
Concrete Crack Slip Check (0-2) : 1
Concrete Crack Width Check (0-2) : 1
Concrete Bond or Adhesion (0-4) : 1
Concrete Creep and Relaxation (0-1) : 1
Concrete Hysteresis (0-3) : 2

```

**Table M.60** Job Data File for VecTor5 Analysis of Duong Frame (Continued)

Reinforcement Hysteresis	(0-3) : 1
Reinforcement Dowel Action	(0-1) : 1
Reinforcement Buckling	(0-1) : 1
Element Strain Histories	(0-1) : 1
Element Slip Distortions	(0-4) : 1
Strain Rate Effects	(0-1) : 1
Structural Damping	(0-1) : 1
Geometric Nonlinearity	(0-1) : 1
Crack Allocation Process	(0-1) : 1

**Table M.61** Auxiliary Data File for VecTor5 Analysis of Duong Frame

```

* * * * *
*       V e c T o r 5 D       *
*   A u x i l i a r y D a t a   *
* * * * *

```

GENERAL ANALYSIS PARAMETERS

\*\*\*\*\*

Section Analysis Mode	(1-5) : 1
Shear Analysis Mode	(0-4) : 3
Shear Protection	(0-1) : 1

Concrete Aggregate Type	(1-2) : 1
Reference Temperature (deg. C)	: 20.0

DYNAMIC ANALYSIS PARAMETERS

\*\*\*\*\*

Time Integration Method	(1-3) : 1
-------------------------	-----------

Mode #1 for Rayleigh Damping	: 1
Mode #2 for Rayleigh Damping	: 2
Damping Ratio for #1 (%)	: 0
Damping Ratio for #2 (%)	: 0

Ground Accel. Factor in x-dir	: 5.0
Ground Accel. Factor in y-dir	: 0.0

Mass Factor due to Self-Weight	: 0.0
--------------------------------	-------

## APPENDIX M2

### EXAMPLE DYNAMIC ANALYSIS (IMPACT LOADING)

As an example application, the input data files of the SS3a-1 beam (Saatci, 2007) are presented in Table M.62 to Table M.65. In addition to the *Structure*, *Job* and *Auxiliary Data Files*, one *Load Case (Load Data File)* is included in this analysis. Details of the structural model and loading are presented in Section 8.3 of Guner (2008). The *Expanded Data Files* as produced by VecTor5 are also presented in Table M.66 to Table M.68.

**Table M.62** Structure Data File for VecTor5 Analysis of Beam SS3a-1

```

* * * * *
*
*           V e c T o r 5 D
*
*           S T R U C T U R E   D A T A
*
* * * * *

GENERAL PARAMETERS
=====

Structure Title           (30 char. max.) : SS3
Structure File Name      (8 char. max.)  : SS3
No. of Members           (50 max.)         : 11
No. of Member Types     (10 max.)         : 2
No. of Nodes             (45 max.)         : 12
No. of Support Nodes    : 3
No. of Support Restraints : 5

STRUCTURE PARAMETERS
=====

(A) Nodal Coordinates
-----
<NOTE:> Coordinate units in mm
<<<<< FORMAT >>>>>
NODE  X      Y [ #NODES d(NODE)  d(X)  d(Y) ] <-- up to 2 dir. /
1     0      0  4      1      235   0/
5    940     0  7      1      250   0/
12 2440    100/
/

(B) Member Indices
-----
<<<<< FORMAT >>>>>
MEM  INC1  INC2  MEM.TYPE [ #MEMS  d(MEM)  d(INC) ] <-- up to 2 dir./
1     1     2     1      10      1      1   /

```

**Table M.62** Structure Data File for VecTor5 Analysis of Beam SS3a-1 (Continued)

11 11 12 2/  
/

(C) Support Restraint List

```

<<<<< FORMAT >>>>>
-----
NODE X-RST Y-RST Z-RST [ #NODES d(NODE) ] <-- up to 2 dir. /
5 0 1 0 /
11 1 0 1 /
12 1 0 1 /
/

```

(D) Member Specifications

```

-----
MT f'c [ f't Ec e0 Mu Cc Kc Agg Dens] [Smx Smy]
(MPa) (MPa) (MPa) (me) (/deg.C) (mm2/hr) (mm) (kg/m3) (mm) (mm)
1 46.7 0 27000 2.51 0 0 0 10 0 0 0
2 400. 400. 1000000 4.00 0 0 0 10 0.0001 0 0
/

```

```

MT Nc Ns Fyz St Dbt Fyt Fut Est Esht esht [Cs ] Ref.
(#) (#) (MPa) (mm) (mm) (MPa) (MPa) (MPa) (MPa) (me) (/deg.C) Type
1 32 2 605 100 7.0 605 652 190250 2794 3.19 0 1
2 32 0 605 100 7.0 605 652 190250 2794 3.19 0 3
/

```

(E) Concrete Layers

```

-----
MT Dc Wc Rho-t Rho-z Nx
(mm) (mm) (%) (%)
1 7.75 250 0 0.57 4
1 11 250 0.3 0.57 2
1 15.2 250 0.3 0.57 1
1 15.2 250 0.3 0 18
1 15.2 250 0.3 0.57 1
1 11 250 0.3 0.57 2
1 7.75 250 0 0.57 4
2 7.75 250 0 0. 4
2 11 250 0. 0. 2
2 15.2 250 0. 0. 1
2 15.2 250 0. 0 18
2 15.2 250 0. 0. 1
2 11 250 0. 0. 2
2 7.75 250 0 0. 4
/

```

(F) Rebar Layers

```

-----
MT N Ys As Dbt Fy Fu Es Esh esh Dep
(mm) (mm2) (mm) (MPa) (MPa) (MPa) (MPa) (me) (me)
1 1 53 1400 29.9 464 630 195000 1088 12.5 0
1 2 357 1400 29.9 464 630 195000 1088 12.5 0
/

```

**Table M.62** Structure Data File for VecTor5 Analysis of Beam SS3a-1 (Continued)

(G) Detailed Member Output List

```

=====
<<<<< FORMAT >>>>>
MEM  [ #MEMS  d(MEM) ] <-- up to 2 directions /
3/
4/
5/
6/
8/
9/
10/
/

```

**Table M.63** Load Data File for VecTor5 Analysis of Beam SS3a-1

```

* * * * *
*   V e c T o r 5 D   *
*   L O A D   D A T A   *
* * * * *

```

LOAD CASE PARAMETERS  
\*\*\*\*\*

```

Load Case ID          (15 char. max.)      : SS3
Load Case Data File   (8 char. max.)       : SS3
Load Factored         (0 - 1)              : 1
Time Factored        (0 - 1)              : 0

No. of Loaded Nodes          : 0
No. of Members w/ End Action Loads : 0
No. of Members w/ Concentrated Loads : 0
No. of Members w/ Distributed Loads  : 0
No. of Members w/ Gravity Loads      : 0
No. of Members w/ Temperature Loads  : 0
No. of Members w/ Concrete Prestrain : 0
No. of Members w/ Support Displacements : 0
No. of Nodes w/ Lumped Mass Assignments : 1
No. of Nodes w/ Impulse Loads        : 0
No. of Ground Acceleration Data      : 0

```

NODAL LOADS  
\*\*\*\*\*

```

<NOTE:> UNITS: kN,kN-m
<<<<< FORMAT >>>>>
NODE Fx Fy Mz [ #NODE d(NODE) d(Fx) d(Fy) d(Mz) ]<-- (2)/
/

```

MEMBER END ACTIONS  
\*\*\*\*\*

```

<NOTE:> UNITS: kN,kN-m
<<<<< FORMAT >>>>>
M AF1 SF1 BM1 AF2 SF2 BM2 [ #M d(M) ]<-- (2)/
/

```

**Table M.63** Load Data File for VecTor5 Analysis of Beam SS3a-1 (Continued)

CONCENTRATED LOADS  
\*\*\*\*\*

<NOTE:> UNITS: kN, kN-m, m  
<<<<< FORMAT >>>>>  
M Fx Fy Mz x/L [ #M d(M) d(Fx) d(Fy) d(Mz) ]<-- (2)/  
/

UNIFORMLY DISTRIBUTED LOADS  
\*\*\*\*\*

<NOTE:> UNITS: kN/m, m  
<<<<< FORMAT >>>>>  
M W a/L b/L [ #M d(M) d(W) ]<-- (2)/  
/

GRAVITY LOADS  
\*\*\*\*\*

<NOTE:>  
<<<<< FORMAT >>>>>  
M GX GY [ #M d(M) ]<-- (2)/  
/

TEMPERATURE LOADS  
\*\*\*\*\*

<NOTE:> UNITS: Deg.C, hrs  
<<<<< FORMAT >>>>>  
M T1' T2' T1 T2 TIME [ #M d(M) ]<-- (2)/  
/

CONCRETE PRESTRAINS  
\*\*\*\*\*

<NOTE:> UNITS: me  
<<<<< FORMAT >>>>>  
M STRAIN [ #ELMT d(ELMT) d(STRAIN) ]<-- (2)/  
/

PRESCRIBED NODAL DISPLACEMENTS  
\*\*\*\*\*

<NOTE:> UNITS: mm, rad  
<<<<< FORMAT >>>>>  
Jnt DOF DISPL [ #Jnt d(Jnt) ]/  
/

ADDITIONAL LUMPED MASSES  
\*\*\*\*\*

<NOTE:> UNITS: kg, m/s  
<<<<< FORMAT >>>>>  
NODE DOF-X DOF-Y MASS Vo-X Vo-Y Acc-X Acc-Y [ #NODE d(NODE) ] /  
12 1 1 105.5 0 -8.0 0 -9.81/  
/

IMPULSE, BLAST AND IMPACT FORCES  
\*\*\*\*\*

<NOTE:> UNITS: Sec, kN  
<<<<< FORMAT >>>>>  
Jnt DOF T1 F1 T2 F2 T3 F3 T4 F4 [ #NODE d(NODE) ] /  
/

**Table M.63** Load Data File for VecTor5 Analysis of Beam SS3a-1 (Continued)

```

GROUND ACCELERATION
*****
<NOTE:> UNITS:  Sec, G
<<<<< FORMAT >>>>>
TIME  ACC-X  ACC-Y/
/

```

**Table M.64** Job Data File for VecTor5 Analysis of Beam SS3a-1

```

* * * * *
*   V e c T o r   *
*   J O B   D A T A   *
* * * * *

Job Title                (30 char max) : SS3
Job File Name           ( 8 char max) : SS3
Date                    (30 char max) : Nov 2007

STRUCTURE DATA
-----
Structure Type          : 5
File Name               (8 char max) : SS3

LOADING DATA
-----
No. of Load Stages     : 25400
Starting Load Stage No. : 1
Load Series ID         (5 char max) : SS3

Load   File Name          Initial   Final     Factors
Case  (8 char max)       Initial   Final     LS-Inc   Type Reps   C-Inc
  1   SS3                 0.000   400.000   0.000500  1   50   0.000
  2   NULL                0.000   0.000    0.000000  1   1   0.000
  3   NULL                0.000   0.000    0.000000  1   1   0.000
  4   NULL                0.000   0.000    0.000000  1   1   0.000
  5   NULL                0.000   0.000    0.000000  1   1   0.000

ANALYSIS PARAMETERS
-----
Analysis Mode           (1-2) : 3
Seed File Name         (8 char max) : NULL
Convergence Limit      (>1.0) : 1.00001
Averaging Factor       (<1.0) : 0.5
Maximum Iterations     : 100
Convergence Criteria   : 1
Results Files          : 1
Output Format           : 1

MATERIAL BEHAVIOUR MODELS
-----
Concrete Compression Base Curve      (0-3) : 2
Concrete Compression Post-Peak       (0-3) : 1
Concrete Compression Softening       (0-8) : 1
Concrete Tension Stiffening          (0-5) : 1
Concrete Tension Softening           (0-3) : 1

```



**Table M.64** Job Data File for VecTor5 Analysis of Beam SS3a-1 (Continued)

Concrete Tension Splitting	(1-2) : 1
Concrete Confined Strength	(0-2) : 1
Concrete Dilatation	(0-1) : 1
Concrete Cracking Criterion	(0-4) : 1
Concrete Crack Slip Check	(0-2) : 1
Concrete Crack Width Check	(0-2) : 1
Concrete Bond or Adhesion	(0-4) : 1
Concrete Creep and Relaxation	(0-1) : 1
Concrete Hysteresis	(0-3) : 2
Reinforcement Hysteresis	(0-3) : 2
Reinforcement Dowel Action	(0-1) : 1
Reinforcement Buckling	(0-1) : 1
Element Strain Histories	(0-1) : 1
Element Slip Distortions	(0-4) : 1
Strain Rate Effects	(0-1) : 1
Structural Damping	(0-1) : 1
Geometric Nonlinearity	(0-1) : 1
Crack Allocation Process	(0-1) : 1

**Table M.65** Auxiliary Data File for VecTor5 Analysis of Beam SS3a-1

```

* * * * *
*           V e c T o r 5 D           *
*   A u x i l i a r y   D a t a   *
* * * * *

```

GENERAL ANALYSIS PARAMETERS

\*\*\*\*\*

Section Analysis Mode	(1-5) : 1
Shear Analysis Mode	(0-4) : 3
Shear Protection	(0-1) : 1
Concrete Aggregate Type	(1-2) : 1
Reference Temperature (deg. C)	: 20.0

DYNAMIC ANALYSIS PARAMETERS

\*\*\*\*\*

Time Integration Method	(1-3) : 3
Mode #1 for Rayleigh Damping	: 1
Mode #2 for Rayleigh Damping	: 2
Damping Ratio for #1 (%)	: 0.
Damping Ratio for #2 (%)	: 0.
Ground Accel. Factor in x-dir	: 0.0
Ground Accel. Factor in y-dir	: 0.0
Mass Factor due to Self-Weight	: 1.0

**Table M.66** Expanded Structure Data File Created by VecTor5 for Beam SS3a-1

```

* * * * *
*
*           V e c T o r 5 D
*
*           E X P A N D E D
*           S T R U C T U R E   D A T A
*
* * * * *

```

```

Structure File Name      : SS3
Structure Title         : SS3
Date                   : Nov 2007
No. of Members         : 11
No. of Member Types    : 2
No. of Nodes          : 12
No. of Support Nodes   : 3
No. of Support Restraints : 5

```

NODAL COORDINATES

\*\*\*\*\*

(mm)

Node	X-Coord	Y-Coord
----	-----	-----
1	0.0	0.0
2	235.0	0.0
3	470.0	0.0
4	705.0	0.0
5	940.0	0.0
6	1190.0	0.0
7	1440.0	0.0
8	1690.0	0.0
9	1940.0	0.0
10	2190.0	0.0
11	2440.0	0.0
12	2440.0	100.0

MEMBER INDICES

\*\*\*\*\*

Member	Inc1	Inc2	Member Type	SP
-----	----	----	-----	--
1	1	2	1	0
2	2	3	1	0
3	3	4	1	0
4	4	5	1	1
5	5	6	1	1
6	6	7	1	0
7	7	8	1	0
8	8	9	1	0
9	9	10	1	0
10	10	11	1	1
11	11	12	2	1

**Table M.66** Expanded Structure Data File Created by VecTor5 for Beam SS3a-1 (Continued)

SP=0: Shear Protection is NOT Active  
 1: Shear Protection is Active

SUPPORT RESTRAINT LIST

\*\*\*\*\*

(0 = Free ; 1 = Fixed)

Node	X-Rst	Y-Rst	Z-Rst
5	0	1	0
11	1	0	1
12	1	0	1

CONCRETE MATERIAL SPECIFICATIONS

\*\*\*\*\*

Mem Typ	fc (MPa)	ft (MPa)	e0 (me)	Ec (MPa)	MU	Cc (/C)	Kc (mm2/hr)	Dens (kg/m3)	Agg (mm)	Nc
1	46.7	2.26	2.51	27000.0	0.15	1.00E-05	4320.	2400.	10.0	32
2	400.0	400.00	4.00	1000000.0	0.15	1.00E-05	4320.	0.	10.0	32

REINFORCEMENT COMPONENT PROPERTIES

\*\*\*\*\*

Mem Typ	Fyz (MPa)	St (mm)	Db (mm)	Fyt (MPa)	Fut (MPa)	Est (MPa)	Esht (MPa)	esht (me)	Cs (/C)	Ns	Ref Typ
1	605.0	100.	7.0	605.0	652.0	190250.0	2794.0	3.2	1.15E-05	2	1
2	605.0	100.	7.0	605.0	652.0	190250.0	2794.0	3.2	1.15E-05	0	3

CONCRETE LAYER DATA

\*\*\*\*\*

Mem Typ	Layer	Dc (mm)	Wc (mm)	RhoT (%)	RhoZ (%)	RhoL (%)	Smx (mm)	Smy (mm)
1	1	7.75	250.0	0.000	0.570	1.993	247.4	410.0
1	2	7.75	250.0	0.000	0.570	1.993	231.9	410.0
1	3	7.75	250.0	0.000	0.570	1.993	216.4	410.0
1	4	7.75	250.0	0.000	0.570	1.993	200.9	410.0
1	5	11.00	250.0	0.300	0.570	1.993	182.1	410.0
1	6	11.00	250.0	0.300	0.570	1.993	179.0	410.0
1	7	15.20	250.0	0.300	0.570	1.993	179.0	410.0
1	8	15.20	250.0	0.300	0.000	1.993	194.7	410.0
1	9	15.20	250.0	0.300	0.000	1.993	225.1	410.0
1	10	15.20	250.0	0.300	0.000	1.993	255.5	410.0
1	11	15.20	250.0	0.300	0.000	1.993	285.9	410.0
1	12	15.20	250.0	0.300	0.000	3.986	241.3	410.0
1	13	15.20	250.0	0.300	0.000	3.986	271.7	410.0
1	14	15.20	250.0	0.300	0.000	3.986	302.1	410.0
1	15	15.20	250.0	0.300	0.000	3.986	332.5	410.0
1	16	15.20	250.0	0.300	0.000	3.986	362.9	410.0
1	17	15.20	250.0	0.300	0.000	3.986	362.9	410.0
1	18	15.20	250.0	0.300	0.000	3.986	332.5	410.0

**Table M.66** Expanded Structure Data File Created by VecTor5 for Beam SS3a-1 (Continued)

1	19	15.20	250.0	0.300	0.000	3.986	302.1	410.0
1	20	15.20	250.0	0.300	0.000	3.986	271.7	410.0
1	21	15.20	250.0	0.300	0.000	3.986	241.3	410.0
1	22	15.20	250.0	0.300	0.000	1.993	285.9	410.0
1	23	15.20	250.0	0.300	0.000	1.993	255.5	410.0
1	24	15.20	250.0	0.300	0.000	1.993	225.1	410.0
1	25	15.20	250.0	0.300	0.000	1.993	194.7	410.0
1	26	15.20	250.0	0.300	0.570	1.993	179.0	410.0
1	27	11.00	250.0	0.300	0.570	1.993	179.0	410.0
1	28	11.00	250.0	0.300	0.570	1.993	182.1	410.0
1	29	7.75	250.0	0.000	0.570	1.993	200.9	410.0
1	30	7.75	250.0	0.000	0.570	1.993	216.4	410.0
1	31	7.75	250.0	0.000	0.570	1.993	231.9	410.0
1	32	7.75	250.0	0.000	0.570	1.993	247.4	410.0
2	1	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	2	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	3	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	4	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	5	11.00	250.0	0.000	0.000	0.000	410.0	410.0
2	6	11.00	250.0	0.000	0.000	0.000	410.0	410.0
2	7	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	8	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	9	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	10	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	11	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	12	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	13	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	14	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	15	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	16	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	17	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	18	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	19	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	20	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	21	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	22	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	23	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	24	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	25	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	26	15.20	250.0	0.000	0.000	0.000	410.0	410.0
2	27	11.00	250.0	0.000	0.000	0.000	410.0	410.0
2	28	11.00	250.0	0.000	0.000	0.000	410.0	410.0
2	29	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	30	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	31	7.75	250.0	0.000	0.000	0.000	410.0	410.0
2	32	7.75	250.0	0.000	0.000	0.000	410.0	410.0

REINFORCEMENT LAYER DATA

\*\*\*\*\*

Mem	N	Ys	Dbs	As	Fy	Fu	Es	Esh	esh	Dep
Typ		(mm)	(mm)	(mm <sup>2</sup> )	(MPa)	(MPa)	(MPa)	(MPa)	(me)	(me)
1	1	53.0	29.9	1400.0	464.0	630.0	195000.	1088.	12.5	0.000
1	2	357.0	29.9	1400.0	464.0	630.0	195000.	1088.	12.5	0.000

**Table M.67** Expanded Load Data File Created by VecTor5 for Beam SS3a-1

```

* * * * *
*
*           V e c T o r 5 D
*
*   E X P A N D E D   L O A D   D A T A
*
* * * * *

```

```

Load Case File Name      : SS3
Load Case I.D.          : SS3
Load / Time Factored    : Load

```

```

No. of Loaded Nodes      : 0
No. of Members w/ End-Action Loads : 0
No. of Members w/ Concentrated Loads : 0
No. of Members w/ Distributed Loads : 0
No. of Members w/ Gravity Loads : 0
No. of Members w/ Temperature Loads : 0
No. of Members w/ Concrete Prestrain : 0
No. of Members w/ Displaced Supports : 0
No. of Nodes w/ Lumped Mass Assign : 1
No. of Nodes w/ Impulse Forces : 0
No. of Ground Acceleration Data : 0

```

```

Reference Temperature    : 20.0

```

LUMPED MASSES  
\*\*\*\*\*

Node	Mass-x self (kg)	Mass-x add. (kg)	Vel-x init. (m/s)	Acc-x const. (m/s <sup>2</sup> )	Mass-y self (kg)	Mass-y add. (kg)	Vel-y init. (m/s)	Acc-y const. (m/s <sup>2</sup> )
1	28.9	0.0	0.00	0.00	28.9	0.0	0.00	0.00
2	57.8	0.0	0.00	0.00	57.8	0.0	0.00	0.00
3	57.8	0.0	0.00	0.00	57.8	0.0	0.00	0.00
4	57.8	0.0	0.00	0.00	57.8	0.0	0.00	0.00
5	59.7	0.0	0.00	0.00	0.0	0.0	0.00	0.00
6	61.5	0.0	0.00	0.00	61.5	0.0	0.00	0.00
7	61.5	0.0	0.00	0.00	61.5	0.0	0.00	0.00
8	61.5	0.0	0.00	0.00	61.5	0.0	0.00	0.00
9	61.5	0.0	0.00	0.00	61.5	0.0	0.00	0.00
10	61.5	0.0	0.00	0.00	61.5	0.0	0.00	0.00
11	0.0	0.0	0.00	0.00	30.8	0.0	0.00	0.00
12	0.0	0.0	0.00	0.00	0.0	105.5	-8.00	-9.81

**Table M.68** Eigen Analysis Results Data File Created by VecTor5 for Beam SS3a-1

```

* * * * *
*
*           V e c T o r 5 D
*
*   E I G E N   A N A L Y S I S   R E S U L T S
*
* * * * *

```

```

Structure File Name           : SS3
Number of Modes Considered    : 21
Mode #1 for Rayleigh Damping  : 1
Mode #2 for Rayleigh Damping  : 2
Damping Ratio (%) for Mode #1 : 0.00
Damping Ratio (%) for Mode #2 : 0.00

Rayleigh Damp. Coeff. for Mass (1/s) : 0.000E+00
Rayleigh Damp. Coeff. for Stiff. (s) : 0.000E+00

```

MODAL DAMPING RATIOS  
\*\*\*\*\*

MODE	DAMPING (%)	MODE	DAMPING (%)
1	0.00	12	0.00
2	0.00	13	0.00
3	0.00	14	0.00
4	0.00	15	0.00
5	0.00	16	0.00
6	0.00	17	0.00
7	0.00	18	0.00
8	0.00	19	0.00
9	0.00	20	0.00
10	0.00	21	0.00
11	0.00		

MODAL PERIODS  
\*\*\*\*\*

MODE	PERIOD (s)	MODE	PERIOD (s)
1	0.178E-01	12	0.275E-03
2	0.487E-02	13	0.245E-03
3	0.266E-02	14	0.239E-03
4	0.140E-02	15	0.226E-03
5	0.895E-03	16	0.217E-03
6	0.729E-03	17	0.203E-03
7	0.546E-03	18	0.173E-03
8	0.499E-03	19	0.138E-03
9	0.400E-03	20	0.132E-03
10	0.321E-03	21	0.301E-04
11	0.284E-03		

**Table M.68** Eigen Analysis Results Data File Created by VecTor5 for Beam SS3a-1 (Continued)

MODE SHAPES  
\*\*\*\*\*

Mode 1			Mode 2		
Node	Disp-x	Disp-y	Node	Disp-x	Disp-y
1	0.000	1.000	1	0.000	1.000
2	0.000	0.746	2	0.000	0.697
3	0.000	0.493	3	0.000	0.409
4	0.000	0.243	4	0.000	0.164
5	0.000	0.000	5	-0.000	0.000
6	0.000	-0.247	6	-0.000	-0.054
7	0.000	-0.470	7	-0.000	-0.008
8	0.000	-0.659	8	-0.000	0.094
9	0.000	-0.804	9	-0.000	0.211
10	0.000	-0.896	10	-0.000	0.304
11	0.000	-0.928	11	0.000	0.341
12	0.000	-0.928	12	0.000	0.341

---