

# USER BULLETIN 1

## BEAM MODELING WITH VECTOR5

### 1. INTRODUCTION

VecTor5 is a computer program for non-linear analysis of two dimensional frame structure made up of beams, columns and shear walls subjected to static and dynamic loading conditions as well as temperature loads. In this bulletin, modeling process for a simply-supported beam is discussed.

Completing this exercise will give you hands-on experience working with VecTor5. The example to be covered is a beam supported at both ends and subjected to a concentrated load at the midspan, which increases until the failure of the beam. Experimental details can be downloaded from [here](#).

The bulletin includes an explanation on how to create the structure and load data files, as well as determining the failure mode of the structure. The process of creating the global modeling is discussed first, followed by the sectional modeling. Information is then given about creating the load and other required files to run VecTor5, followed by the determination of failure mode.

The files required to perform an analysis with VecTor5 are: structure data file, load data file(s), job data file, and auxiliary data file. These files must be placed in the same folder together with VT5.exe. All input files can be modified using a standard text editor such as Microsoft Windows Notepad. The files required to determine failure modes are analysis results output files such as files with suffix .A5E. These files can be found at the same folder as structure data files and load data files.

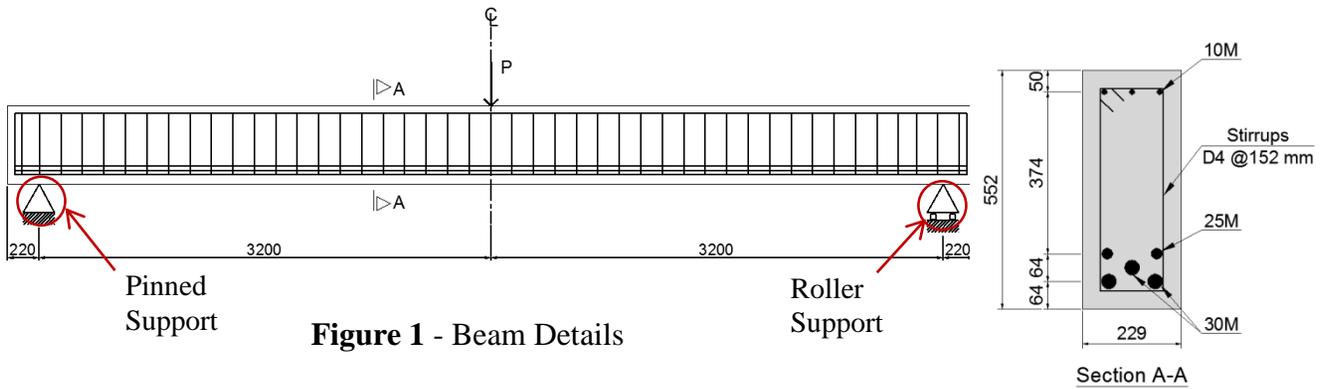
In order to get a better understanding of the program, it is recommended reading the User's Manual of Vector5, which can be downloaded from Dr. Guner's [website](#) (see User's Manuals tab).

---

This document is prepared by Harley Viana, and later edited by Agneya Loya, Chu Peng, and Kyle Blosser, as parts of projects supervised by Dr. Serhan Guner.

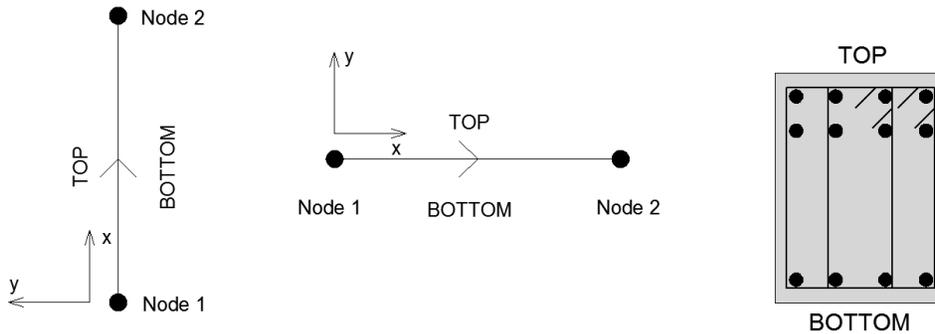
## 2. GLOBAL BEAM MODELING

This section will discuss the process of creating a VecTor5 model of the beam shown in **Figure 1**. First of all, it is necessary to sketch an analytical model of the structure.



**Figure 1 - Beam Details**

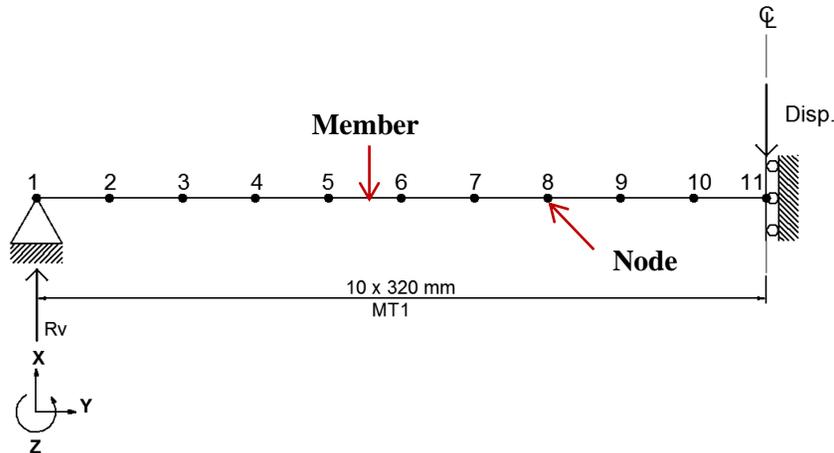
The top of the cross section is based on the element orientation. Therefore, it is important to pay special attention to this due to the fact that asymmetrical cross sections can have their position changed if the local coordinates are specified in a different way. The orientation of each member is defined as below:



**Figure 2 - Member Orientation and Local Coordinates**

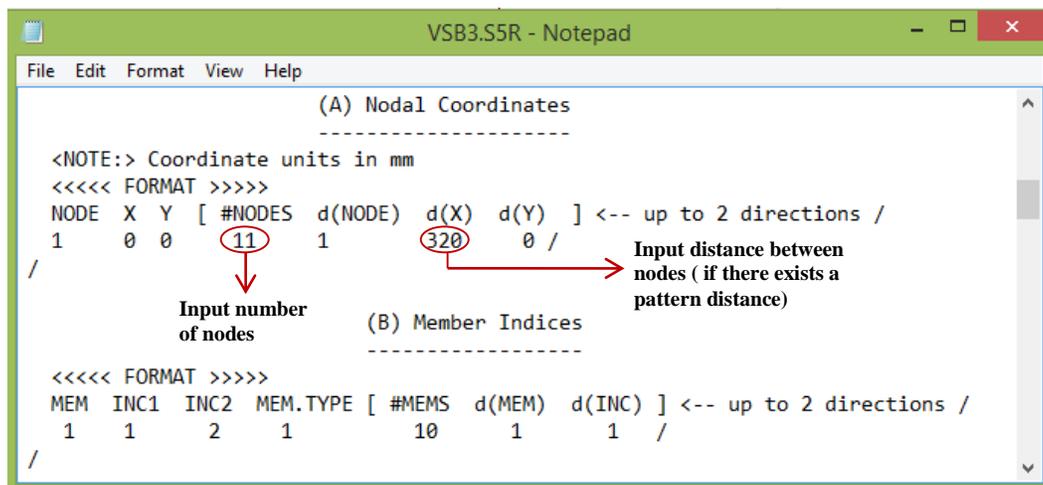
If the loads applied to the structure are symmetric relative to the plane of symmetry, then the full model can be replaced with half the model by applying appropriate boundary conditions. Remember that the mid-span load must be doubled. When modeling a beam, nodes are to be created to divide member lengths. It is common practice to space the nodes equally, although this is not necessary.

**Figure 3** illustrates the analytical model that is going to be used in this tutorial. Since the member length should be in the range of 50% of the section height, the half of the span was divided into 10 members of 320 mm, which corresponds to approximately 58% of the section height.



**Figure 3 - Analytical Model**

Once the analytical model is done, users are able to fill Tables (A) and (B) of the *Structure Data File* as follows:



**Figure 4 - Inputting Nodal Coordinates and Member Indices**

Refer to the User manual for more details on the above method of defining the beam.

Summary of Section(A) and Section(B):

Node – refers to the beginning node number

X – the x-coordinate of the first node

Y – the y-coordinate of the first node

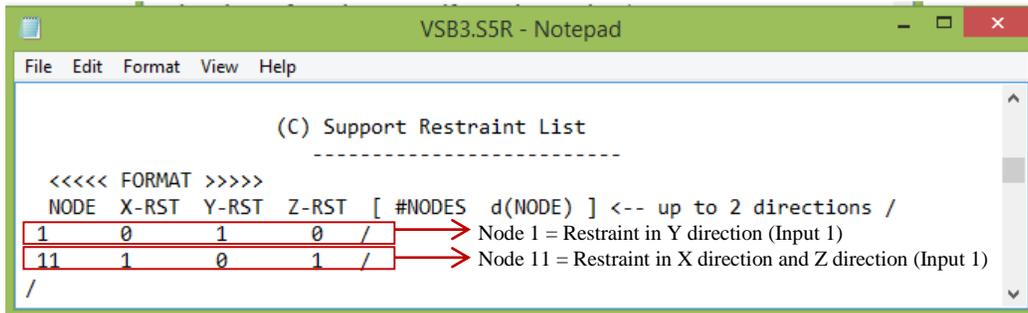
d(Node) – increment in node numbers

d(X), d(Y) – increment in X and Y coordinates - Depends on number of members the beam or half-beam is divided into. The criteria is to ensure member length is close to 50% of beam depth.

d(MEM) – increment in member numbers

INC 1, INC 2 – node numbers of initial member – this means the node increments increase by 1 at a time  
d(INC) – increment in node numbers

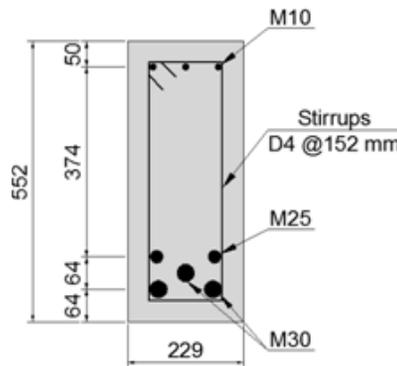
In **Figure 5**, X-RST and Y-RST correspond to the translational degrees-of-freedom; Z-RST corresponds to the rotational degree-of-freedom about the z-axis, which is normal to the plane of the structure as shown in **Figure 3**. You need to change the node numbers according to your support requirement. For this beam, the end nodes are defined:



**Figure 5 - Defining Boundary Condition**

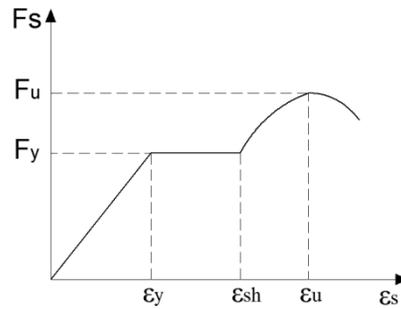
### 3. SECTIONAL MODELING

The sectional modeling consists of inputting information about the section features in the *Structure Data file (.S5R)*. It will be required to enter the information regarding steel reinforcement material properties used in the cross section.



**Figure 6 - Cross Section A-A**

The values of  $F_y$ ,  $F_u$ ,  $\epsilon_{sh}$  and  $\epsilon_u$  can be obtained from the stress-strain diagram of the reinforcing bars used. Remember that  $F_y$  and  $F_u$  are the yield and ultimate stresses of the transverse reinforcement respectively;  $\epsilon_{sh}$  is the strain where strain hardening of the reinforcement begins; and  $\epsilon_u$  is the strain corresponding to  $F_u$  stress (**Figure 7**).

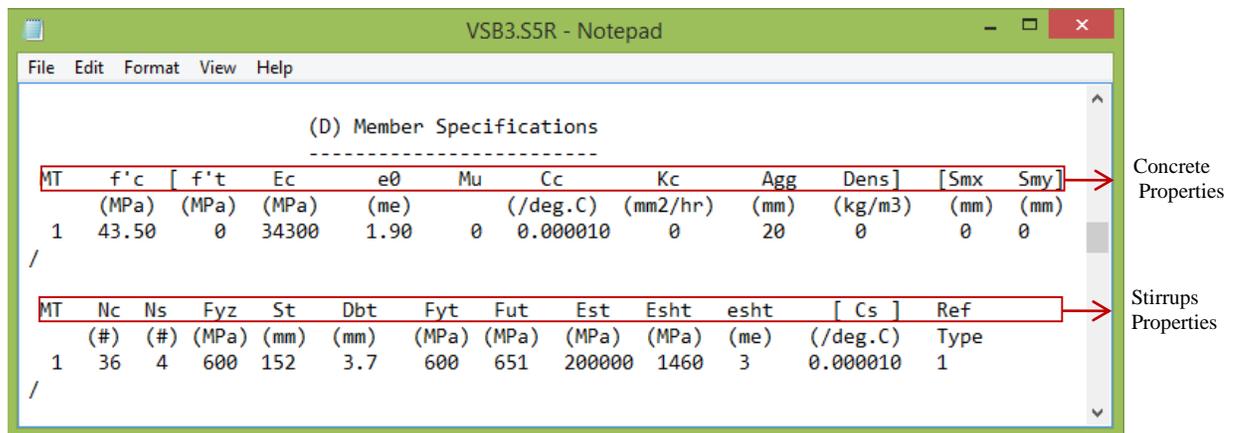


**Figure 7 - Stress-Strain Diagram**

**Table 1** shows the steel reinforcement material properties applied to the cross sections, which have to be input in the structure data file at *Member Specifications* and *Rebar Layers*. The properties of stirrups are inserted in *Member Specifications* at the second table; whereas the properties of longitudinal rebars are entered in *Rebar Layers* table (see **page 7**).

**Table 1 - Steel Reinforcement material properties**

Bar Size	Diameter (mm)	Cross Sectional Area (mm <sup>2</sup> )	$\epsilon_{sh} \times 10^{-3}$	Fy (MPa)	Fu (MPa)	Es (MPa)
10M	11.3	100	7.7	315	460	200000
25M	25.2	500	8.5	445	680	220000
30M	29.9	700	11.4	436	700	200000
D4	5.7	25.7	3.0	600	651	200000



**Figure 8 - Member Specifications**

For each member type used, the parameters shown in **Figure 8** must be defined, based on which concrete base curve is constructed. For more details, please see **page 10**.

MT – member type or cross-section type

f'c – concrete compressive strength

f't – concrete tensile strength (leave 0 to use the default values)

Ec – modulus of elasticity of concrete (leave 0 to use the default values)  
e0 – strain corresponding to peak stress of concrete (leave 0 to use the default values)  
Mu – poisson’s ratio (leave 0 to use the default values)  
Cc – coefficient of thermal expansion of concrete (leave 0 to use the default values)  
Kc – thermal diffusivity of concrete (leave 0 to use the default values)  
Agg – maximum size of aggregate used in the mixture (leave 0 to use the default values)  
Dens – density of concrete (leave 0 to use the default values)  
Smx, Smy – crack spacing in element oriented along x and y (leave 0 to use the default values)

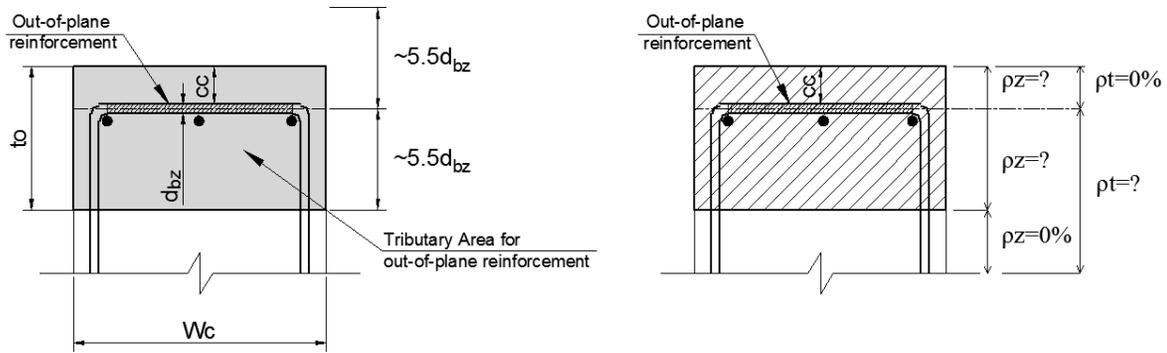
If an input of 0 is input for the variable in [ ], the program will use the default values. After performing an analysis, these default values can be verified from the .S5E data. Values of  $f'_c$ , Ec, e0 depend on the grade of concrete used and can be obtained from an appropriate design code.

Nc – total number of concrete layers  
Ns – total number of reinforcing and prestressing steel layers (must match number in the Rebar layers)  
Fyz – yield stress of out-of-plane reinforcement (z-direction)  
St – spacing of transverse reinforcement along the longitudinal direction  
Dbt – bar diameter of transverse reinforcement  
Fyt, Fut – yield and ultimate stresses of transverse reinforcement  
Est – modulus of elasticity of transverse reinforcement  
Cs – coefficient of thermal expansion of transverse and longitudinal reinforcement  
esht – strain value at which strain hardening begins  
eut – strain value at ultimate stress  
Ref – reference type, specifies member behaviour and nodal degrees of freedom (the default value is 1, more details are given in the User Manual)

Nc depends on the sectional modeling (see below). Ns can be easily determined by inspection of the cross-section. St, Fyz, Dbt, Fyt, Fut, Est, esht, and eut are for the stirrups used and will be obtained from a material standard code or steel coupon tests.

### 3.1 Defining Concrete Layers

The calculation of Rho-z and Rho-t, ratios of the out-of-plane and transverse reinforcement present in the corresponding concrete layer respectively, involves the concept of tributary area, as shown in **Figure 9**.

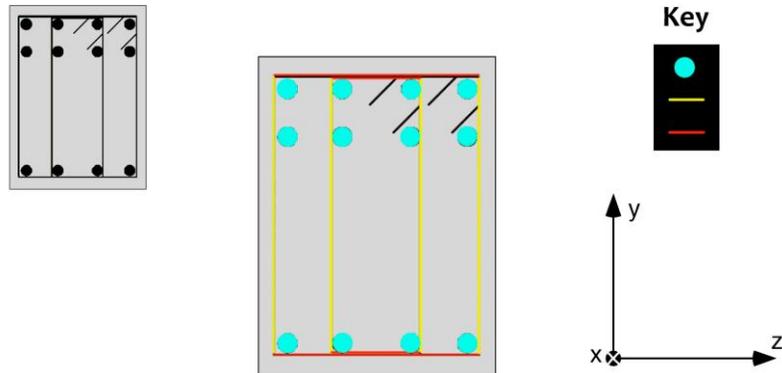


**Figure 9** - Determination of the out-of-plane reinforcement tributary area

The tributary area is the area over which the out-of-plane reinforcement is smeared or spread. This area is defined as the diameter of the out-of-plane reinforcement times a factor of 5.5 and then adding the concrete cover (CC). This is the  $t_o$  variable in Equations 1 and 2 below. Note 1 from **Table 2** further describes this determination.

Refer to **Figure 10** for the directions and meaning of transverse, longitudinal and out of plane reinforcement:

1. Cyan along x-axis is the longitudinal reinforcement
2. Yellow along y-axis is the transverse reinforcement
3. Red along z-axis is the out-of-plane reinforcement



**Figure 10** – Reinforcement notation

$$\rho_t = \frac{2 \times A_b}{S_t \times W_c} \quad (\text{Eqn. 1})$$

$$\rho_z = \frac{A_b}{S_t \times t_o}; \quad (\text{Eqn. 2})$$

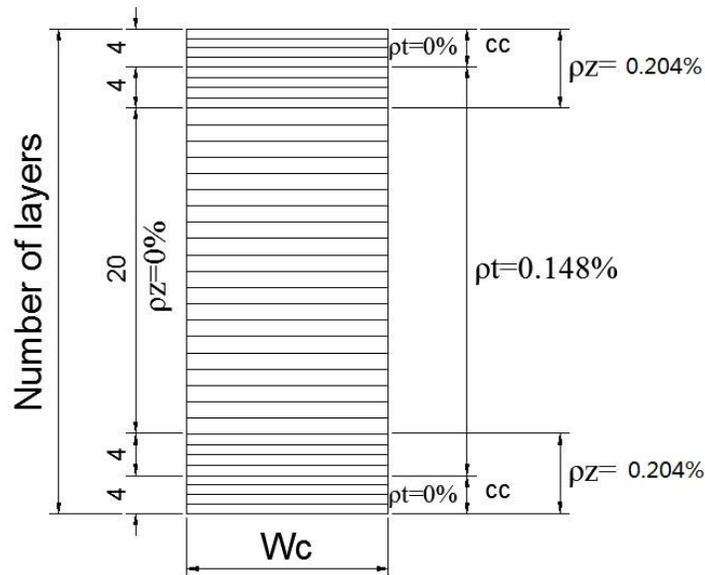
where  $A_b$  is the cross sectional area of out of plane reinforcement;  $S_t$  is the spacing of the out-of-plane reinforcement in the longitudinal direction,  $t_o$  is the distance, in the transverse direction, in which the out-of-plane reinforcement is to be assigned, and  $W_c$  is the width of the cross section.

Section A-A has a clear cover of 43 mm. Hence, the layers that correspond to this area need to have  $\rho_t = 0\%$ .

**Table 2 - Calculation of Tributary Area, Rho-z and Rho-t**

Section	Tributary Area	Rho-z	Rho-t
A - A	$Ta = 5.5 \times 5.7$ $= 31.35 \text{ mm}$ <i>* Read note 1</i>	$\rho_z = \frac{25.7}{152 \times 83} = 0.204\%$	$\rho_t = \frac{2 \times 25.7}{152 \times 229} = 0.148\%$

\*Note 1: As the concrete thickness above the centerline of the out-of-plane reinforcement is 41.5 mm ( $38.65 + 5.7/2 = 41.5$  mm), it is better to adopt a tributary area of 41.5 mm rather than 31.35 mm for this particular beam. Consequently, the number and width of the concrete layers should cover a depth of  $41.5 + 41.5 = 83$  mm ( $t_o$ ) at both the top and bottom of the cross section.



**Figure 11- Sectional Modeling: Concrete Layers - Section A-A**

In addition, other fields should be filled out, in which the user has to put the member type number (MT), thickness of each layer ( $D_c$ ), width of the cross section ( $W_c$ ) and the number of layers with the same details ( $N_x$ ). For accuracy, the number of layers allows for a clean decimal, so that no portion of  $D_c$  is lost. Both values for Rho-t and Rho-z are used in layers 4 – 8 because both the transverse and out-of-plane reinforcement affect these layers; this is similar for layers 29 – 32. **Figure 12** illustrates these steps.

(E) Concrete Layers

MT	Dc (mm)	Wc (mm)	Rho-t (%)	Rho-z (%)	Nx	
1	9.6625	229	0	0.204	4	4 layers of 9.6625mm = 38.65mm (cc)  4 * 11.0875 + 4 * 9.6625 = 83 mm (to)
1	11.0875	229	0.148	0.204	4	
1	19.3	229	0.148	0	20	
1	11.0875	229	0.148	0.204	4	
1	9.6625	229	0	0.204	4	

Figure 12 - Concrete Layers: Specifying Rho-z and Rho-t

Note that when the cross-section has U stirrups the Rho-z is 0% at the top; while cross-sections that do not have stirrups the Rho-t is 0%, and Rho-z is 0% in both regions at the top and the bottom.

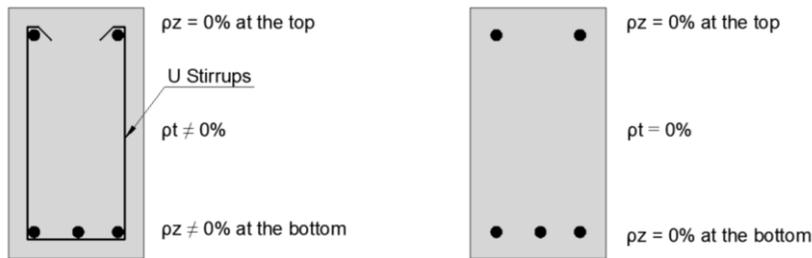


Figure 13 - Other Examples: Cross-Sections with U Stirrups and no Stirrups

The next step is specifying the rebar layers. In this example, it must be created 4 rebar layers as presented in Figure 14.

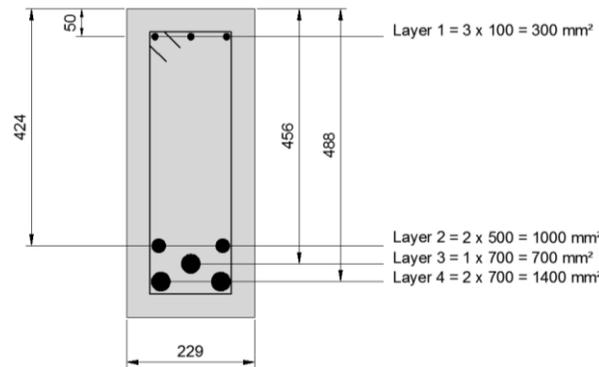


Figure 14 - Rebar Layers

VS3.S5R - Notepad

File Edit Format View Help

(F) Rebar 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	456.00	700.0	29.90	436.0	700.0	200000	1610	11.40	0.000
1	4	488.00	1400.0	29.90	436.0	700.0	200000	1610	11.40	0.000

Figure 15 - Specifying Rebar Layers

$MT$  is the member type,  $N$  is the longitudinal reinforcement component number starting from 1 and increasing in number by 1,  $Y_s$  is the location of the longitudinal reinforcement layer from the top of the cross section,  $A_s$  is the total area of the longitudinal reinforcement layer,  $Db$  is the diameter of one bar, and  $E_s$  is the modulus of elasticity of the longitudinal reinforcement.

#### 4. LOAD FILE

The *load data file* (.L5R) contains information about nodal loads, externally applied member end actions, concentrated loads, uniformly distributed loads, gravity loads, temperature loads, concrete prestrains, prescribed nodal displacements, additional lumped masses, impulse, blast and impact forces and ground accelerations.

In the example presented here, only one load file will be created since the structure is subjected to only one load applied vertically at the midspan, as shown in **Figure 1**. Thus, a monotonically increasing vertical displacement was imposed at Node 11 and the analyses were continued until failure of the beam occurred. Therefore, this is a nodal displacement with a value of -1. It shall be noted that Node 11 has 2 degrees of freedom: in the y-direction and z-direction.

```

VSB3.L5R - Notepad
File Edit Format View Help
PRESCRIBED NODAL DISPLACEMENTS
*****
<NOTE:> UNITS: mm,rad
<<<<< FORMAT >>>>>
NODE DOF DISPL [ #NODES d(NODE) ]/
11 2 -1/
/

```

Therefore, this is a nodal displacement with a value of -1. It shall be noted that Node 11 has 2 degrees of freedom: in the y-direction and z-direction.

**Figure 16** shows which load case parameters were assigned in this example.

```

VSB3.L5R - Notepad
File Edit Format View Help
*****
* 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.)   : VSB3
Load Case Data File (8 char. max.)   : VSB3
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 Nodes 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

```

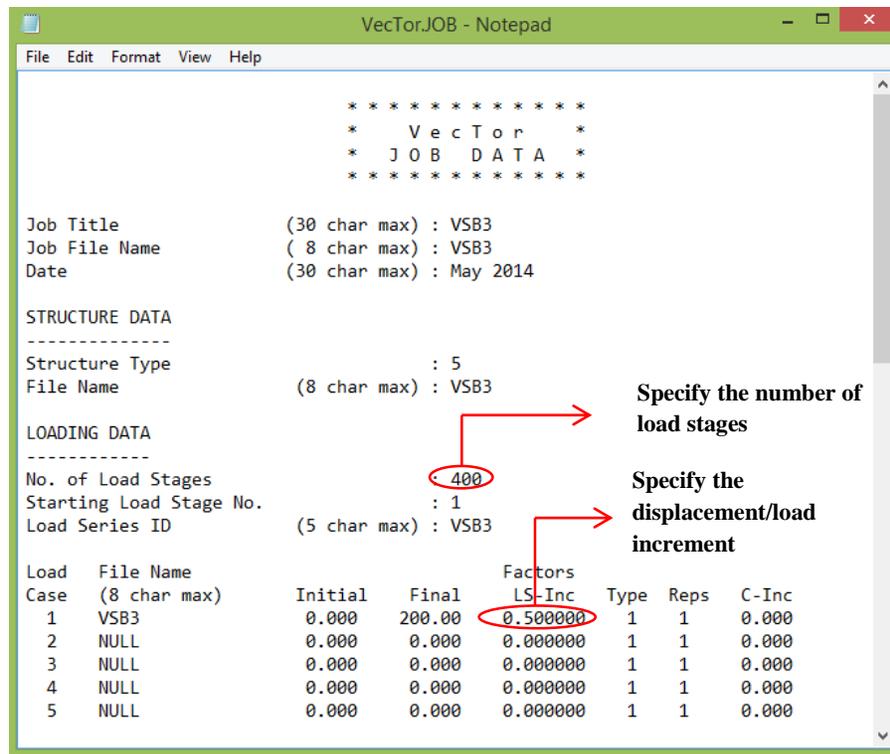
**Figure 16** - Specification of Load Parameters and Prescribed Nodal Displacements

#### 5. JOB FILE

The *job file* contains information about the loading data, analysis parameters, and material behaviour models. Herein users specify the total number of load stages, time, temperature and displacement increment, as well as loading type (i.e, monotonic, cyclic, or reversed-cyclic) for each load case. In analysis parameters, the type of the analysis (i.e, static or dynamic) is defined. To run the program in

the default mode, it is not necessary to modify the analysis parameters and the material behaviour models contained in this file. The *job title* must have the same name as the structure data file; otherwise, VecTor5 will not run. The following figures shows which fields are to be filled.

As the failure displacement of the beam is not known before the analysis, it is recommended to consider a large number of load stages to avoid the possibility of the termination of the analysis before the failure of the beam. In this example, 400 load stages with an increment of 0.5 mm is input, which means that the displacement will gradually increase following this sequence: 0, 0.5, 1.0, and so on.



**Figure 17 - Job Data File - Specifying Load Data**

The *Concrete Compression Base Curve* is selected based on the concrete strength input. The available base curves are:

0 - Linear,

1 - Hognestad (Parabola) (Normal Strength Concrete: NSC): It is recommended to use this default model for concrete strengths up to 40 MPa. In this case, the users have to input e0 value so that the program is able to calculate the modulus of elasticity of the concrete using *Equation 3*. If the Ec value is input, the program ignores it.

$$E_c = \frac{f'c}{e_0} \quad (\text{Eqn. 3})$$

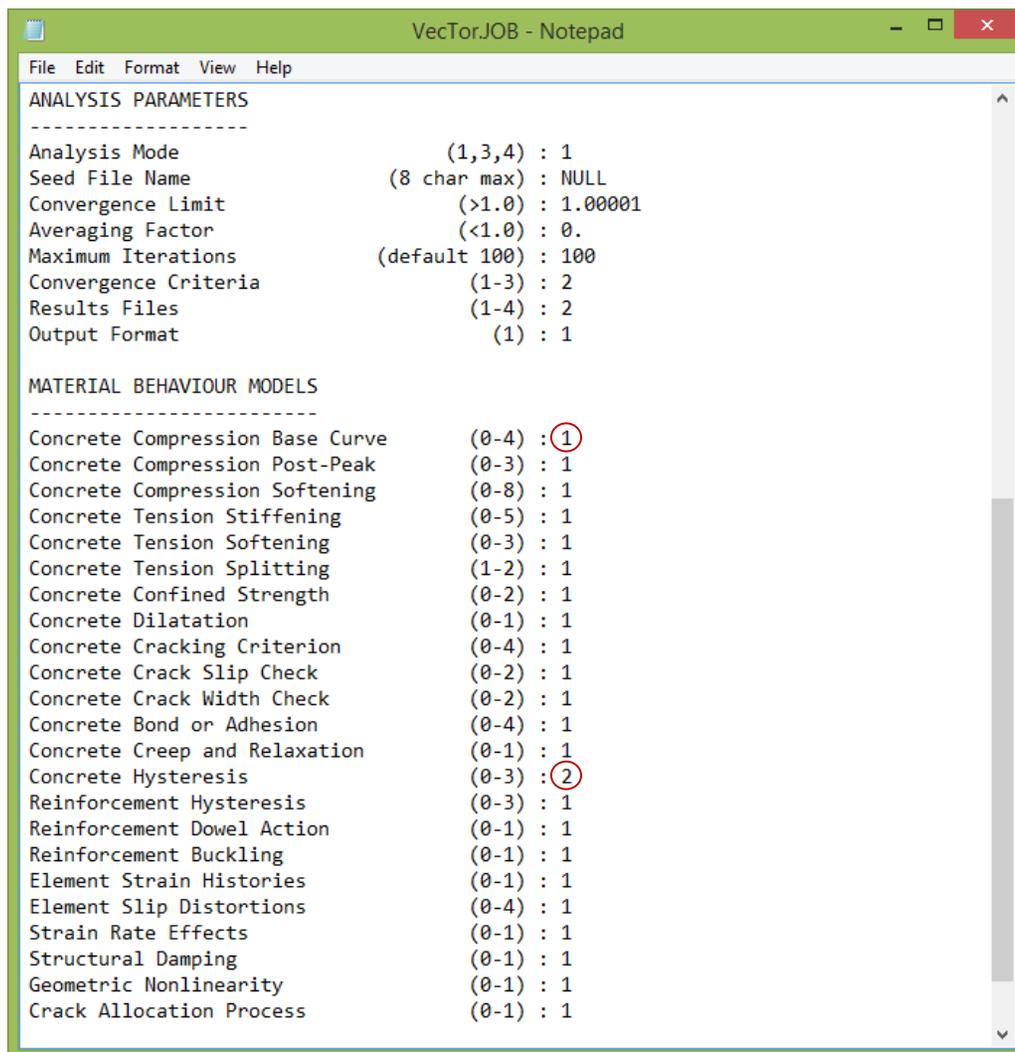
2 - Popovics (NSC): This model can be used for concrete strengths up to 50MPa. In this option, users need to input both Ec and e0 values to create the concrete compression base curve.

3 - Popovics (High Strength Concrete: HSC): This model is used when the concrete has strengths greater than 50MPa. In this case, users have to input the e0 value, which allows the program to calculate Ec value as follows. If the Ec value is input, the program ignores it.

$$n = 0.80 + \frac{f'c}{17} \quad (\text{Eqn. 4})$$

$$Ec = \frac{f'c}{e0} \times \frac{n}{n - 1} \quad (\text{Eqn. 5})$$

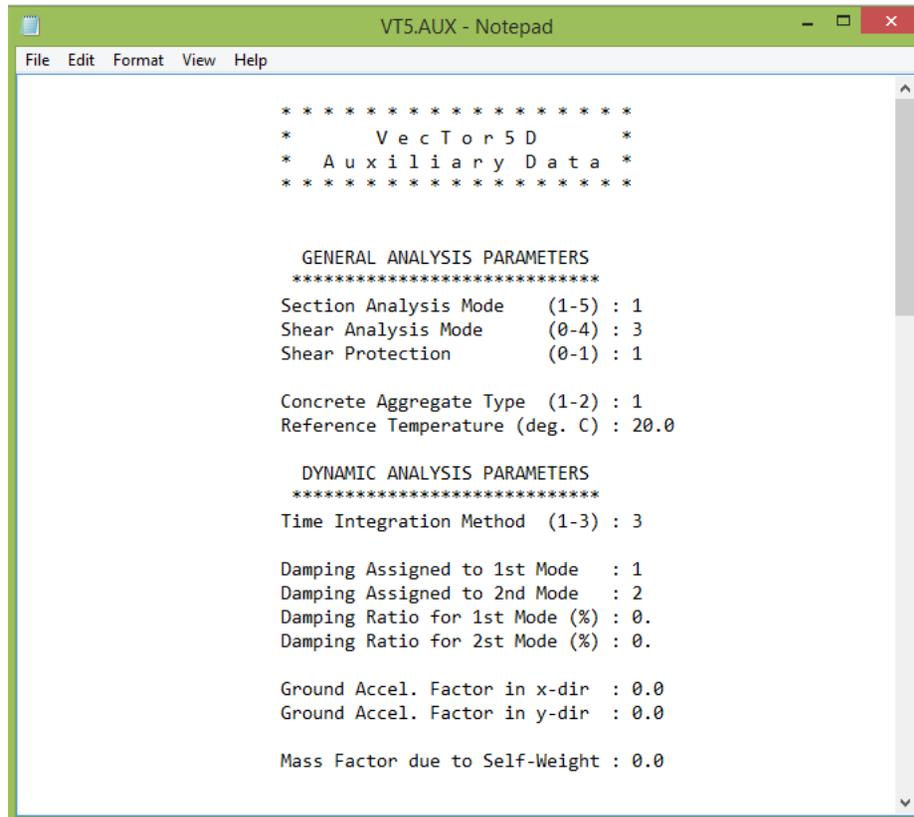
4 - Hoshikuma (HSC): This model is also used for high concrete strengths ( $\geq 50\text{MPa}$ ). In this alternative, the user needs to input Ec and e0 values to create the concrete compression base curve. In all cases, the concrete strength f'c must be input. The default values for other concrete properties are: Mu = 0.15; Cc =  $10 \times 10^{-6} \text{ 1/}^\circ\text{C}$ ; Kc =  $4320 \text{ mm}^2/\text{h}$ ; Agg =  $10 \text{ mm}$ ; and Dens =  $2400 \text{ kg/m}^3$  After performing an analysis, it is recommended to review the expanded data file, with S5E extension, to verify the actual parameters used in the analysis.



**Figure 18** - Job Data File - Specifying Analysis Parameters and Material Behaviour Models

## 6. AUXILIARY DATA FILE

The Auxiliary Data File contains information about the analysis parameters. In *general analysis parameters*, it is possible to define the sectional analysis mode, shear analysis mode, shear protection, dynamic average factor, concrete aggregate type and reference temperature. **Figure 19** shows which values are input. For Non-linear Section Analysis, this file can remain unchanged, but is required for the analysis.



```
VT5.AUX - Notepad
File Edit Format View Help
*****
*   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

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

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

**Figure 19** - Auxiliary Data File - Input Field

Notes:

### *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*

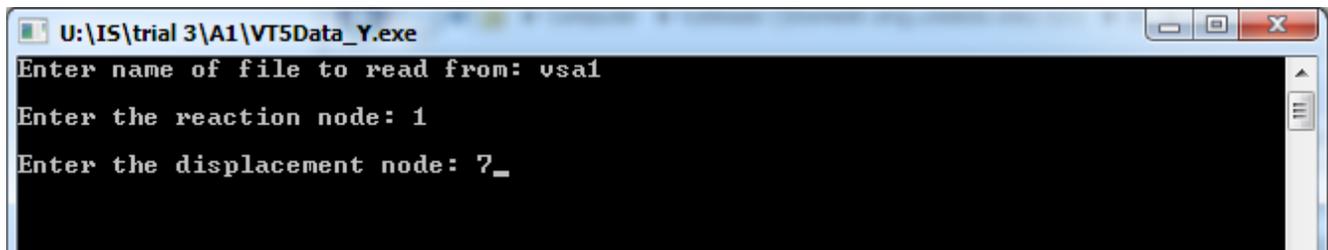
- 0 - Shear not considered,
- 1 - Uniform Shear Flow (Multi-Layer),

- 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.

## 7. RESULTS

Once the analysis is complete, it is necessary to obtain the data with displacements and equivalent loads at the midspan using VT5Data\_Y program. Users need to type the name of the file and specify the node; in this case, the results shows the displacements and reactions at the node 11. To specify the name of the file, the name must match the output files, for example, type “vsa1” for beam VSA1. The reaction node is Node 1, while the displacement is at Node 7.



```
U:\IS\trial 3\A1\VT5Data_Y.exe
Enter name of file to read from: vsa1
Enter the reaction node: 1
Enter the displacement node: 7_
```

## 8. DETERMINATION OF FAILURE MODE

Take VSA1 for example. We need to determine at which load stage the first flexural cracking, first shear cracking and first yielding of reinforcement occurs, respectively. Then determine the failure mode of this beam.

### 8.1 First Flexural Cracking

First flexural cracking usually occurs at the mid-span of a beam. Users need to first check load stage No. 2. If WCR at any concrete layers for any members are not zero, flexural cracking occurs. For this case, at member 6, from layer 25 to layer 32, WCR is not zero (**Figure 20**), so first flexural cracking occurs at load stage No. 2 with applied mid-span load 55.9 KN.

MEMBER : 6

MOMENT: 48.9 kN-m CURVATURE: 0.413 me/m  
 AXIAL LOAD: 0.0 kN AXIAL STRAIN: 0.000 me  
 SHEAR: -14.0 kN SHEAR STRAIN: -0.011 me

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
6	0.00	0.0	0.00	0.00	0
7	0.00	0.0	0.00	0.00	0
8	0.00	0.0	0.00	0.00	0
9	0.00	0.0	0.00	0.00	0
10	0.00	0.0	0.00	0.00	0
11	0.00	0.0	0.00	0.00	0
12	0.00	0.0	0.00	0.00	0
13	0.00	0.0	0.00	0.00	0
14	0.00	0.0	0.00	0.00	0
15	0.00	0.0	0.00	0.00	0
16	0.00	0.0	0.00	0.00	0
17	0.00	0.0	0.00	0.00	0
18	0.00	0.0	0.00	0.00	0
19	0.00	0.0	0.00	0.00	0
20	0.00	0.0	0.00	0.00	0
21	0.00	0.0	0.00	0.00	0
22	0.00	0.0	0.00	0.00	0
23	0.00	0.0	0.00	0.00	0
24	0.00	0.0	0.00	0.00	0
25	0.01	136.2	-0.00	-0.00	2
26	0.01	144.9	-0.00	-0.00	2
27	0.01	180.0	-0.00	-0.00	2
28	0.01	155.3	-0.00	-0.00	2
29	0.01	155.6	-0.00	-0.00	2
30	0.02	163.8	-0.00	-0.00	2
31	0.02	187.8	-0.00	0.00	2
32	0.02	207.9	-0.00	0.00	2

**Figure 20** – Specifying how to know first flexural cracking by looking at WCR

There is another method for determining first flexural cracking. Longitudinal reinforcement stress and strain at the crack are zero at load stage No. 1 (indicating no flexural cracking) while at load stage No. 2, stress and strain at crack are larger than zero. So first flexural cracking occurred at load stage No. 2 with applied mid-span load 55.9 KN. **Figure 21** gives more details of this analyzing process.

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	0.000	0.00	0.000	0.00	0.00
2	424.0	0.00	-0.000	-0.00	0.000	0.00	-0.00
3	488.0	0.00	-0.000	-0.00	0.000	0.00	-0.00

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	-0.093	-18.58	0.000	0.00	-5.57
2	424.0	0.00	0.062	13.56	0.059	12.30	13.56
3	488.0	0.00	0.088	17.62	0.088	18.34	24.67

**Figure 21** – Specifying another method for determining first flexural cracking

Note:

WCR - average crack width

SCR – crack spacing (in mm)

### 8.2 First Shear Cracking

If cracks exist in the compression zone, these cracks must be shear cracking because no flexural cracking will occur in the compression zone. Users need to open output files, then look at member deformations and find the most critical member in shear by finding the largest absolute value of GXY. Then go to the detailed output of this member and determine if cracks exist in the compression zone.

For example, opening VSA1\_08.A5E, by looking at *Member Deformations* in **Figure 22**, we can find member 5 is the most critical member in shear, so we need to go to the detailed output of member 5. In **Figure 23**, We can see from layer 1 to layer 14, FCX is smaller than zero which indicates the compression zone is from layer 1 to 14. Then we can see from *Crack Conditions* in **Figure 24** that layer 14 (compression zone) has a crack with width 0.03 mm, and this crack must be shear cracking. But for load stage No.7, no cracks exist at compression zone. So first shear cracking occurs at load stage No.8 with applied mid-span load 203.4 KN. What’s more? At load stage No.8, for layers 15 and 16, *THETA1* (**Figure 25**) is quite smaller than 0, which indicates inclined tension field. Cracks at inclined tension field should be shear cracking.

MEMBER DEFORMATIONS							
*****							
M	ECL (me)	GXY (me)	PHI (me/m)	ESL-MAX (me)	ESL-MIN (me)	EST-MAX (me)	WCR-MAX (mm)
1	-0.002	-0.041	0.123	0.024	-0.030	0.005	
2	0.017	-0.098	0.619	0.148	-0.123	0.022	0.05
3	0.074	-0.135	1.272	0.344	-0.213	0.038	0.10
4	0.135	-0.167	1.935	0.546	-0.302	0.053	0.16
5	0.193	-0.191	2.594	0.743	-0.393	0.067	0.22
6	0.239	-0.096	3.248	0.927	-0.495	0.081	0.28

**Figure 22** – Specification of finding the most critical member in shear at load stage No.8

CONCRETE STRESSES  
-----

NC	FCX (MPa)	FCY (MPa)	VC (MPa)	FC1 (MPa)	FC2 (MPa)	FC2/FP
1	-11.025	0.000	-0.050	-0.002	-11.025	0.487
2	-10.619	0.000	-0.161	0.000	-10.621	0.470
3	-10.146	0.000	-0.281	0.006	-10.154	0.449
4	-9.664	0.000	-0.398	0.014	-9.680	0.428
5	-9.073	-0.010	-0.534	0.022	-9.105	0.403
6	-8.366	-0.009	-0.688	0.047	-8.422	0.372
7	-7.587	-0.008	-0.845	0.085	-7.680	0.340
8	-6.653	-0.007	-1.021	0.146	-6.806	0.301
9	-5.578	-0.006	-1.198	0.241	-5.825	0.258
10	-4.468	-0.005	-1.360	0.377	-4.850	0.215
11	-3.324	-0.003	-1.506	0.578	-3.906	0.173
12	-2.148	-0.002	-1.633	0.879	-3.029	0.134
13	-0.944	-0.001	-1.740	1.331	-2.276	0.101
14	-0.121	-0.003	-1.576	1.515	-1.639	0.087
15	0.397	-0.006	-1.219	1.431	-1.040	0.066
16	0.760	-0.006	-0.894	1.349	-0.596	0.049

**Figure 23** – Specification of finding compression zones at load stage No.8 by looking at FCX

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
6	0.00	0.0	0.00	0.00	0
7	0.00	0.0	0.00	0.00	0
8	0.00	0.0	0.00	0.00	0
9	0.00	0.0	0.00	0.00	0
10	0.00	0.0	0.00	0.00	0
11	0.00	0.0	0.00	0.00	0
12	0.00	0.0	0.00	0.00	0
13	0.00	0.0	0.00	0.00	0
14	0.03	390.4	0.00	0.00	2
15	0.04	391.6	0.00	0.00	2
16	0.06	398.5	-0.00	-0.00	2
17	0.08	344.3	-0.00	-0.00	2

**Figure 24** – Specifying crack exists at compression zone at load stage No.8

NET STRAINS  
-----

NC	TEMP (C)	ECT (me)	EX (me)	EY (me)	GXY (me)	E1 (me)	E2 (me)	THETA1 (Deg)	E1T
1	0.00	-0.573	-0.492	-0.000	-0.002	-0.000	-0.492	90.108	-0.000
2	0.00	-0.547	-0.474	-0.000	-0.006	-0.000	-0.474	90.360	-0.000
3	0.00	-0.518	-0.452	-0.000	-0.010	-0.000	-0.453	90.658	-0.000
4	0.00	-0.490	-0.431	-0.000	-0.015	0.000	-0.431	90.979	0.000
5	0.00	-0.455	-0.403	-0.000	-0.020	-0.000	-0.404	91.401	-0.000
6	0.00	-0.415	-0.373	-0.000	-0.025	0.000	-0.373	91.953	0.000
7	0.00	-0.372	-0.338	-0.000	-0.031	0.000	-0.338	92.639	0.000
8	0.00	-0.321	-0.296	-0.000	-0.038	0.001	-0.297	93.626	0.001
9	0.00	-0.265	-0.248	-0.000	-0.044	0.002	-0.250	95.051	0.002
10	0.00	-0.210	-0.199	-0.000	-0.050	0.003	-0.202	97.086	0.003
11	0.00	-0.154	-0.148	-0.000	-0.055	0.005	-0.153	100.300	0.005
12	0.00	-0.098	-0.095	-0.000	-0.060	0.009	-0.104	106.145	0.009
13	0.00	-0.043	-0.042	-0.000	-0.064	0.017	-0.059	118.546	0.017
14	0.00	0.013	0.013	-0.000	-0.067	0.041	-0.028	-39.379	0.041
15	0.00	0.069	0.068	0.006	-0.076	0.086	-0.012	-25.488	0.086
16	0.00	0.125	0.124	0.008	-0.081	0.137	-0.005	-17.433	0.137
17	0.00	0.180	0.180	0.007	-0.082	0.189	-0.002	-12.620	0.189

**Figure 25** – Specifying inclined tension field at load stage No.8 by looking at THETA1

Notes:

GXY - shear strain of the concrete at the mid-depth

FCX - longitudinal axial stress of the concrete

THETA1 - inclination of the principal tensile stress field based on the MCFT (Vecchio and Collins, 1986) or the DSFM (Vecchio, 2000)

### 8.3 First yielding of Reinforcement

First yielding of reinforcement can be determined by opening output files one by one and find whether or not the average longitudinal reinforcement reaches its yielding stress, if it reaches the yielding stress, find the load stage number and then find the applied mid-span load corresponding to this number.

**Figure 26** shows average longitudinal reinforcement stress of load stage No. 25, we can see clearly that no reinforcement reaches yielding stress, but for load stage No. 26 in **Figure 27**, the reinforcement in the third steel layer has reached yielding stress, so first yielding of reinforcement occurs at load stage No. 26. The corresponding applied mid-span load is 465.9 kN. We also need to consider whether or not stirrups reach yielding stress, but by inspection, the stress of stirrups is small in the output files and does not reach yielding stress.

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	-1.220	-243.93	0.000	0.00	-73.18
2	424.0	0.00	1.667	366.70	1.824	380.07	366.70
3	488.0	0.00	2.161	432.15	2.246	439.75	605.02

**Figure 26** - Average longitudinal reinforcement stress of load stage No.25

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	-1.255	-251.05	0.000	0.00	-75.31
2	424.0	0.00	1.758	386.85	1.870	389.53	386.85
3	488.0	0.00	2.274	436.00	2.354	439.75	610.40

**Figure 27** - Average longitudinal reinforcement stress of load stage No.26

## 8.4 Failure Mode

From the graph drawn using data calculated by VecTor5, we can see there is a sharp decrease of loading capacity for VSOA1, VS0A2 and VSOA3, this significant brittle failure indicates diagonal-tension failure mode. Because of the lacking of stirrups, only concrete can sustain shear force, thus diagonal failure occurs before flexural failure. So VSOA1, VS0A2 and VSOA3 belong to diagonal-tension failure.

The failure mode of VSA1 belong to shear-compression failure or flexure-compression failure. Both of this two failure modes are ductile failure modes, so it is difficult to tell the failure mode of VSA1 by purely looking at the load-deflection curve. It is important to look at the output files of VecTor5; one obvious difference between these two failure modes is that flexure-compression failure doesn't have significant diagonal shear cracking, while diagonal shear cracking for shear-compression failure is significant.

From the load-deflection curve of VSA1, we can see at mid-span displacement of 25 mm (load stage No. 51), the beam load capacity dropped significantly. So we must open output file VSA1\_51.A5E. Inspection of this output file (**Figure 28**) reveals that Member 5 is the most critical member in terms of shear deformations because absolute value of GXY for member 5 is the largest. And we need to further look at detailed member output for member 5. **Figure 29** shows detailed crack conditions output of member 5.

MEMBER DEFORMATIONS *****							
M	ECL (me)	GXY (me)	PHI (me/m)	ESL-MAX (me)	ESL-MIN (me)	EST-MAX (me)	WCR-MAX (mm)
1	0.012	-0.097	0.519	0.122	-0.105	0.018	0.04
2	0.402	-3.119	2.048	0.836	-0.061	3.977	1.94
3	0.431	-2.718	3.323	1.136	-0.320	4.199	1.65
4	0.489	-3.288	4.879	1.523	-0.614	4.886	2.21
5	0.568	-3.684	6.540	1.955	-0.910	5.476	2.65
6	2.104	-1.604	29.479	8.354	-4.558	2.254	2.49

**Figure 28** – Specification of the most critical member in shear by looking at GXY

CRACK CONDITIONS					
NC	WCR (mm)	SCR (mm)	VCI (MPa)	SLIP (mm)	STATE
1	0.00	0.0	0.00	0.00	0
2	1.60	442.5	-0.17	-0.31	1
3	1.80	424.7	-0.19	-0.32	1
4	1.97	405.4	-0.21	-0.32	1
5	0.65	359.3	0.02	0.03	1
6	1.30	386.3	0.00	0.03	1
7	1.40	390.3	0.01	0.07	1
8	1.41	390.4	0.02	0.11	1
9	2.08	412.8	0.01	0.07	1
10	2.27	419.4	0.02	0.08	1
11	2.51	455.9	0.00	0.02	1
12	2.61	452.7	0.00	0.02	1
13	2.65	449.0	0.00	0.02	1
14	2.62	444.7	0.00	0.02	1
15	2.53	439.6	0.00	0.02	1
16	2.37	420.8	-0.29	-0.36	1
17	1.73	361.5	-0.54	-0.58	1
18	1.35	331.1	-0.65	-0.61	1
19	1.24	313.1	-0.65	-0.47	1
20	1.11	291.9	-0.62	-0.34	1
21	0.95	266.4	-0.58	-0.24	1
22	0.78	236.1	-0.56	-0.17	1
23	0.62	201.9	-0.53	-0.13	1
24	0.47	164.7	-0.48	-0.09	1
25	0.38	143.4	-0.43	-0.07	1
26	0.36	147.0	-0.36	-0.06	1
27	0.41	174.2	-0.29	-0.06	1
28	0.34	151.0	-0.22	-0.05	1
29	0.34	150.6	-0.04	-0.02	1
30	0.35	157.9	-0.01	-0.01	1
31	0.40	180.2	-0.00	0.00	1
32	0.45	200.4	-0.00	0.00	1
33	0.51	222.4	-0.00	0.00	1
34	0.57	244.2	-0.00	0.00	1

**Figure 29** - Detailed crack conditions output of member 5

From **Figure 29**, we can see diagonal shear cracking at the mid depth of the concrete section is very large comparing with the flexural cracking at the bottom of the concrete, so diagonal shear cracking is dominant, thus shear-compression failure for VSA1 can be concluded. A similar analysis method can be used here for other beams. Be aware that diagonal-tension failure mode is mainly concluded by observing if the load-deflection curve has a sharp decrease of loading capacity.

In conclusion, we need to first look at the load-deflection graph drawn using data calculated by VecTor5; if there is a sharp decrease of loading capacity which indicates brittle failure, then diagonal-tension failure mode can be concluded. If the graph does not have a sharp decrease, failure mode will be flexural-compression failure or shear-compression failure depending on what kind of cracking is dominant. If flexural cracking is dominant, the failure mode will be flexural-compression failure; if diagonal shear cracking is dominant, the failure mode will be shear-compression failure.

## 9. REFERENCES

Guner, S. and Vecchio, F. J. (2008), "User's Manual of VecTor5," Online Publication, 88 pp. [Download](#) (see User's Manuals tab)

---

This document is prepared by Harley Viana, and later edited by Agneya Loya, Chu Peng, and Kyle Blosser as parts of projects supervised by Dr. Serhan Guner.