USER BULLETIN 2

FRAME 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 frame structure is discussed.

Completing this exercise will give you hands-on experience working with VecTor5. The example to be covered is a two story frame structure subjected to a lateral load, which is applied to a second storey beam in monotonically and cyclically increasing manners until the failure of the frame, and two constant axial loads. The structure is supported by two fixed supports at the base. Experimental details can be downloaded from here.

The bulletin includes an explanation on how to create the structure and load data files and determining the failure mode. 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. Last followed by 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 should 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.

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 Chu Peng and Kyle Blosser, as parts of projects supervised by Dr. Serhan Guner.

2. GLOBAL FRAME MODELING

The process of modeling starts with the creation of the global frame model. Thus, it is necessary to create a sketch of the structural model, defining nodes and members.

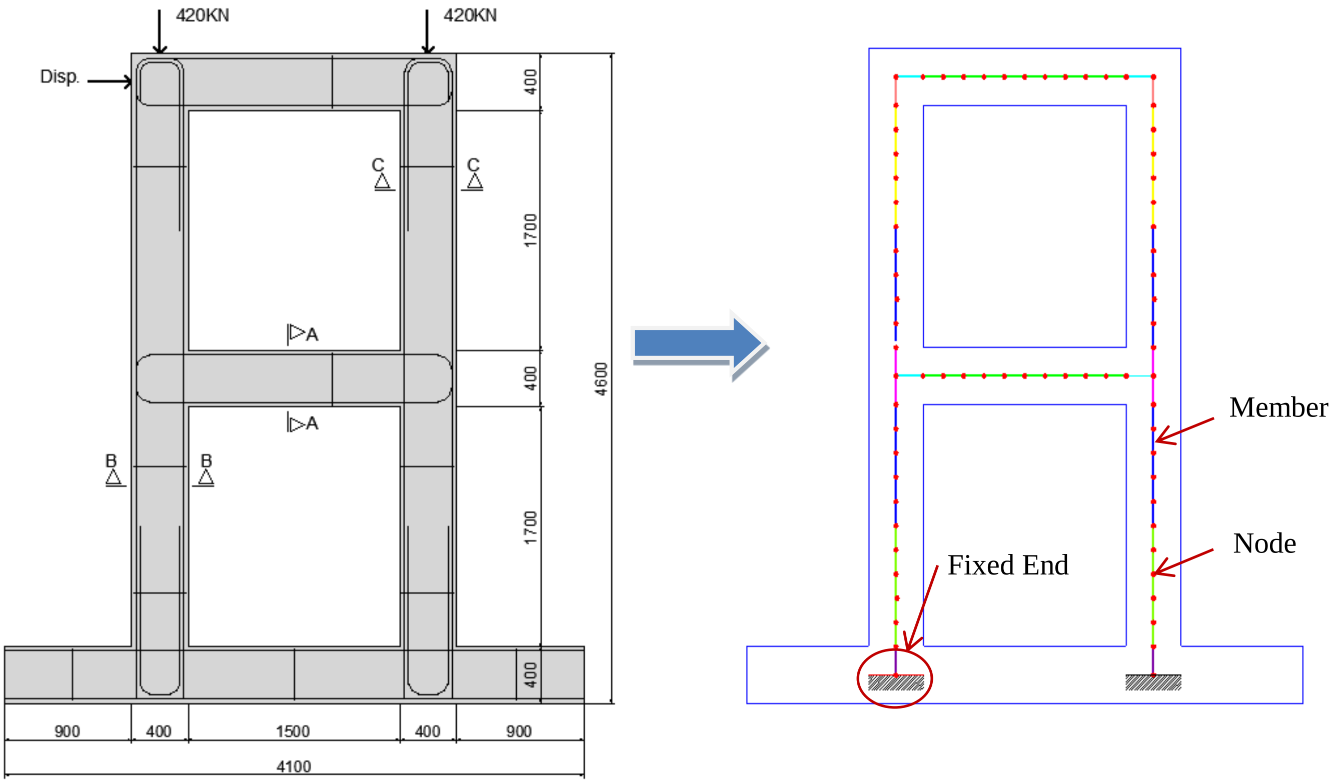


Figure 1 - Sketching a Structural Model

The orientation of the members requires special attention in the presence of asymmetric cross sections. The orientation of each member can be defined as presented below:

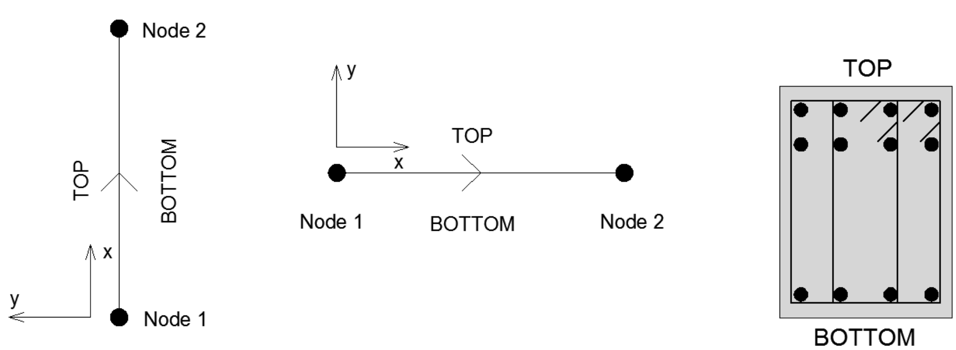
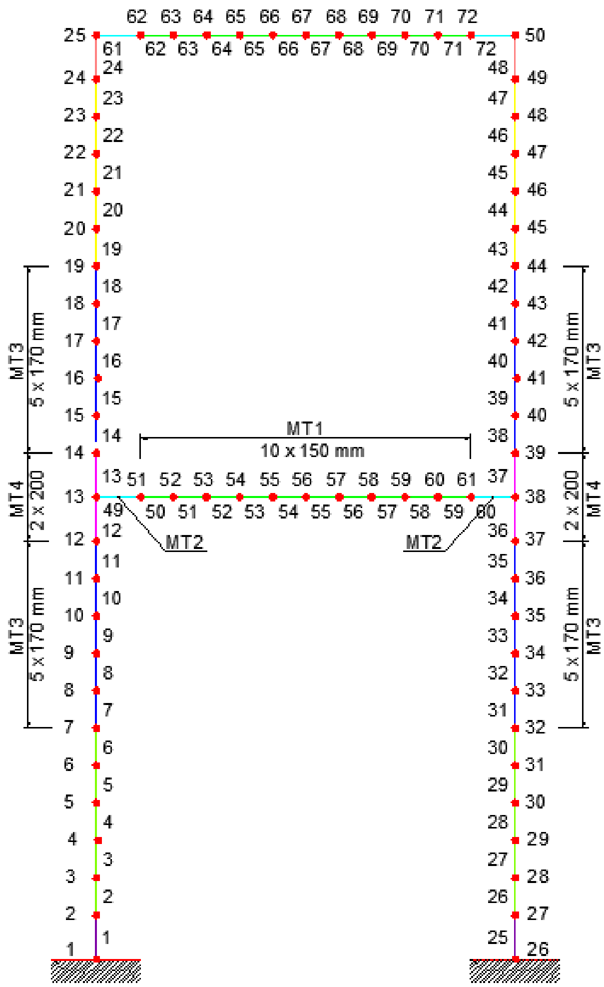


Figure 2 - Member Orientation and Local Coordinates

The length of each member should be in the range of 50% of the section height.



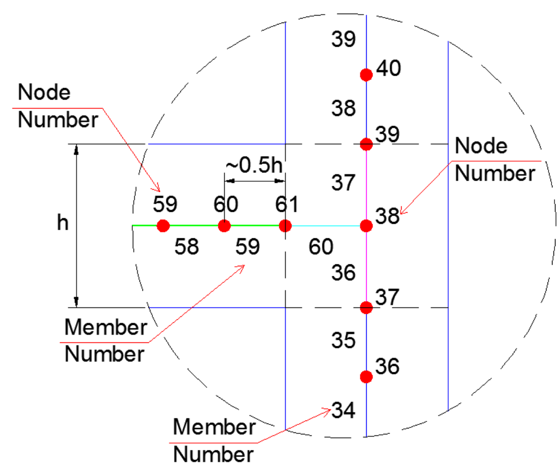


Figure 3 - Analytical Model

Figure 4 – Specification of a Beam – Column Joint

The beam-column joints are modeled as shown in **Figure 4**. The members that connect beams to columns are determined by the extension of the borders of the cross sections. It is important to insert nodes in these extensions, which are represented by dashed lines (see nodes 37, 39 and 61). In this example, the length of members at joints is equal to 200 mm (one half of the cross section height).

It's important to specify member types correctly. Member 50 to member 59 and member 62 to member 71 have the same cross section, so these members can be defined as member type 1 (MT1). Member 49, 60, 61 and 72 have the same cross section and reinforcement amounts of these members are twice of MT1. We can define these members as MT2. Same method can be used to define other member types. Specify attention need to be paid on asymmetric sections (Reinforcement locations and areas in compression and tension zones are not the same). For example, there are two longitudinal reinforcement layers in the right part of member 19 to member 23, while one layer in the left part of these members. From the orientation convention, we know left part of these members are top with one reinforcement layer. But left part of member 43 to member 47 is top with two reinforcement layers, so member 19 to 23 differ from member 43 to 47. We can define member 19 to 23 as MT5 and member 43 to 47 as MT6.

All member types that connect beams to column need to have their Rho-z, Rho-t and steel rebar areas doubled, as discussed in the **page 7** of this tutorial file.

3. SECTIONAL MODELING

The sectional modeling consists of defining the section details in the structure data file. The steel reinforcement material properties used in the cross section should be input.

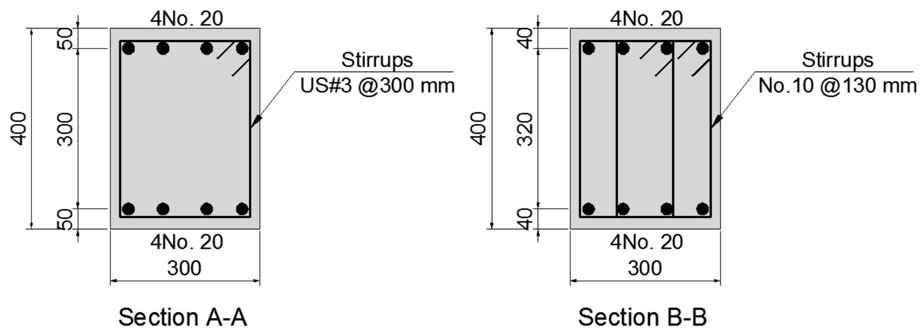


Figure 5 - Cross section A-A and B-B

The values of F_y, F_u, ε_{sh} and ε_u can be obtained from the stress-strain diagram of the reinforcing bars used. Remember that Fy and Fu are the yield and ultimate stresses of the transverse reinforcement respectively; ε_{sh} is the strain where strain hardening of the reinforcement begins; and ε_u are the strains corresponding to Fu (**Figure 6**).

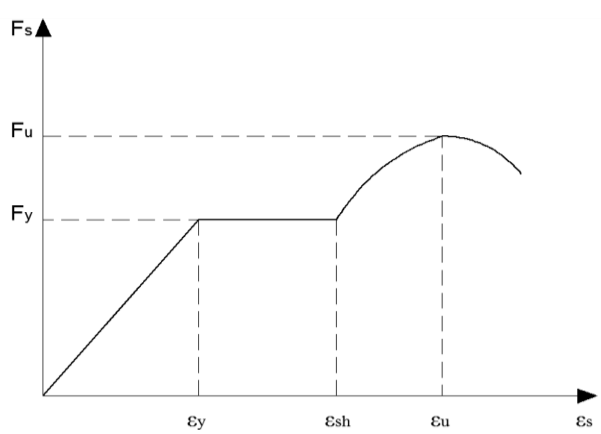


Figure 6 - Stress-Strain Diagram

Table 1 shows the steel reinforcement material properties used in this cross sections.

Table 1 - Steel Reinforcement material properties

Bar Size	Diameter (mm)	Cross Sectional Area (mm²)	ε_{sh} x 10^{-3}	Fy (MPa)	Fu (MPa)	Es (MPa)
No.10	11.3	100	22.8	455	583	192400
No.20	19.5	300	17.1	447	603	198400
US #3	9.5	71	28.3	506	615	210000

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 follows:

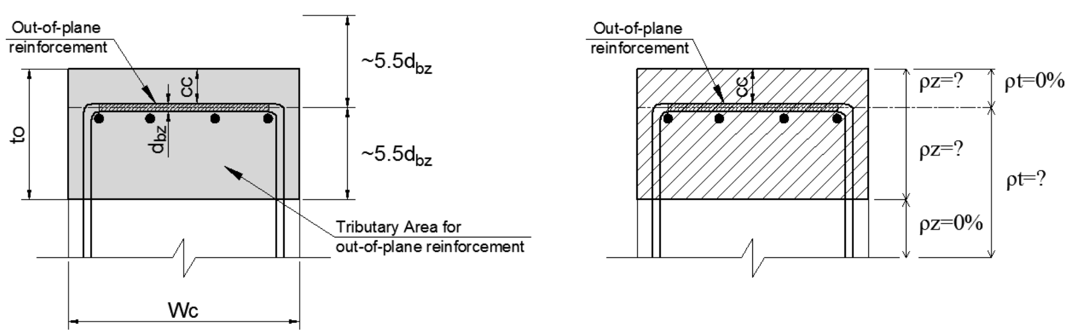


Figure 7 - Determination of the out-of-plane reinforcement tributary area

$$\rho z = \frac{Ab}{St\ x\ to}; \qquad \rho t = \frac{2.Ab}{St\ x\ Wc} \qquad (Eqn. 1\ \&\ Eqn. 2)$$

where *Ab* is the cross sectional area of out of plane reinforcement; St is the spacing of the out-of-plane reinforcement in the longitudinal direction, *to* is the distance, in the transverse direction, in which the out-of-plane reinforcement is to be assigned, and *Wc* is the width of the cross section.

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

Section	Tributary Area	Rho-z	Rho-t
A - A	Ta = 5.0 x 9.5 ≈ 50 mm	$\rho z = \frac{71}{300\ x\ 100} = 0.237\%$	$\rho t = \frac{2\ x\ 71}{100\ x\ 300} = 0.158\%$
B - B	Ta = 5.0 x 11.3 ≈50 mm	$\rho z = \frac{1.5\ x\ 100}{130\ x\ 100} = 1.157\%$	$\rho t = \frac{4\ x\ 100}{130\ x\ 300} = 1.026\%$

Note that the Section B-B has 2 stirrups; therefore, there are 2 top horizontal legs and 4 vertical legs. However, in the calculation of Rho-z, the cross sectional area of out of plane reinforcement was multiplied by 1.5 since there exist some parts on the top and bottom of this section that have only one horizontal leg (See **Figure 8**).

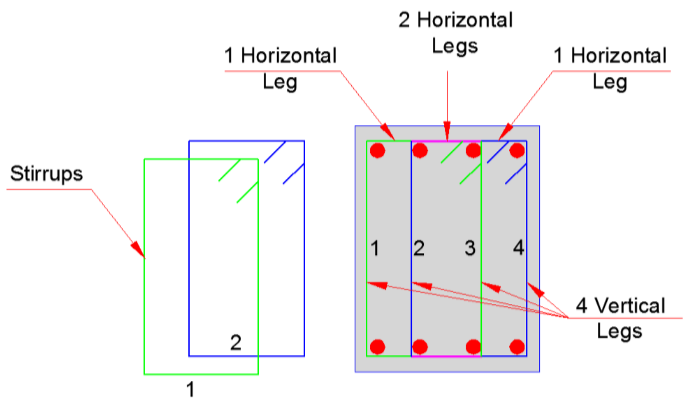


Figure 8 - Detailing of Stirrups for Section B-B

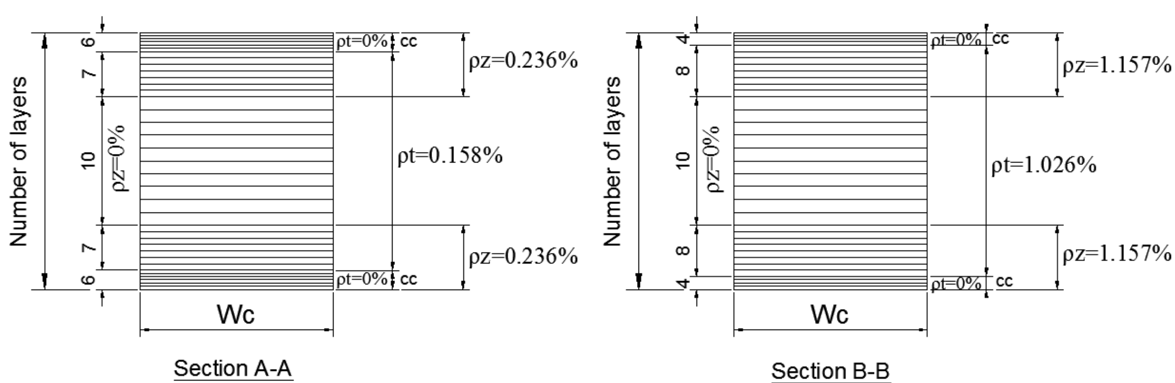


Figure 9 - Sectional Modeling: Concrete Layers

Section A-A has a clear cover of 30 mm, and for the other sections the clear cover is 20 mm. Hence, the layers corresponding to the clear cover should have $\rho t = 0\%$. In addition, other fields should be filled out, in which the user has to input the member type number (MT), thickness of each layer (Dc), width of the cross section (Wc) and the number of layers with the same details (Nx). **Figure 10** illustrates these steps.

KVD.S5R - Notepad

File Edit Format View Help

(E) Concrete Layers

MT	Dc (mm)	Wc (mm)	Rho-t (%)	Rho-z (%)	Nx
1	5	300	0.00	0.236	6
1	10	300	0.158	0.236	7
1	20	300	0.158	0.000	10
1	10	300	0.158	0.236	7
1	5	300	0.000	0.236	6
2	5	300	0.000	0.473	6
2	10	300	0.316	0.473	7
2	20	300	0.316	0.000	10
2	10	300	0.316	0.473	7
2	5	300	0.000	0.473	6
3	5	300	0.000	1.157	4
3	10	300	1.026	1.157	8
3	20	300	1.026	0.000	10
3	10	300	1.026	1.157	8
3	5	300	0.000	1.157	4
4	5	300	0.000	2.314	4
4	10	300	2.052	2.314	8
4	20	300	2.052	0.000	10
4	10	300	2.052	2.314	8
4	5	300	0.000	2.314	4

6 layers of 5 mm = 30 mm (cc)

4 layers of 5 mm = 20 mm (cc)

Figure 10 - Concrete Layers: Specifying Rho-z and Rho-t

For the members near joints (MT2, MT4), Rho-z and Rho-t should be two times the value of the ratios of the adjacent members – i.e., for MT2: Rho-z will be $2\ x\ 0.237\% = 0.473\%$ and Rho-t = $2\ x\ 0.158\% = 0.316\%$. Similarly, for MT4: Rho-z will be equal to 2.314% and Rho-t = 2.052%.

4. REBAR DEVELOPMENT (Optional)

In this example, the inner rebars have to be developed gradually.

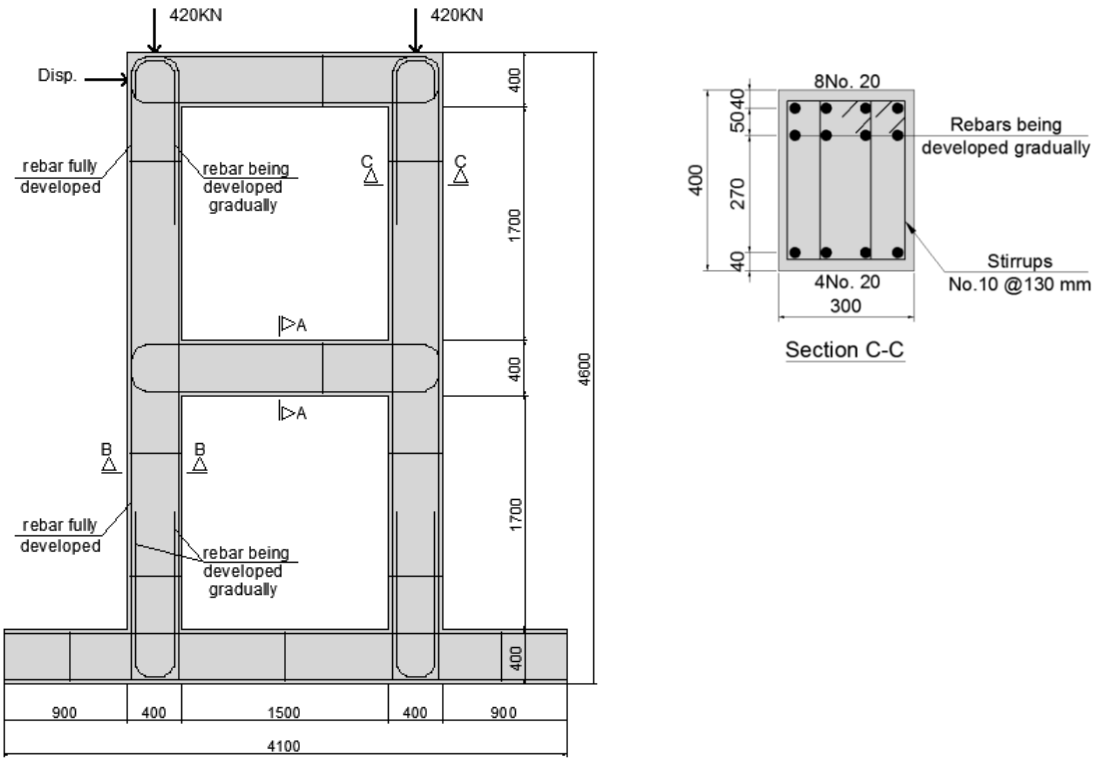


Figure 11 - Rebar Development

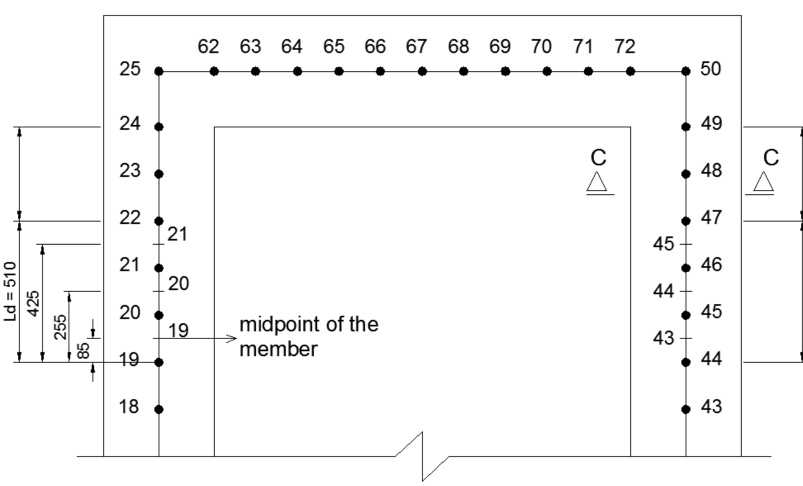


Figure 12 - Rebar Development: Calculating Multipliers for Rebar Cross-Sectional Areas

For the example shown in **Figure 12**, the development length (Ld) includes 3 members (19, 20 and 21). In order to find the multipliers for the rebar cross-sectional areas, the user has to calculate the corresponding percentage of development for each member, considering the development length at mid-points of each member.

Member 19 = $\frac{85}{510}\ x\ 100\% \approx 15\%$ $\rightarrow$ $0{,}15\ x\ 1200\ mm^2 = 180mm^2 = 4\ x\ 45mm^2$

Member 20 = $\frac{255}{510}\ x\ 100\% = 50\%$ $\rightarrow$ $0{,}50\ x\ 1200\ mm^2 = 600mm^2 = 4\ x\ 150mm^2$

Member 21 = $\frac{425}{510}\ x\ 100\% \approx 85\%$ $\rightarrow$ $0{,}85\ x\ 1200\ mm^2 = 1020mm^2 = 4\ x\ 255mm^2$

Accordingly, 4 bars will be input with diameters to give a cross-sectional area of 45mm² for member 19, 150 mm² for member 20, and 255 mm² for member 21. **Figure 13** presents the rebar development for this example.

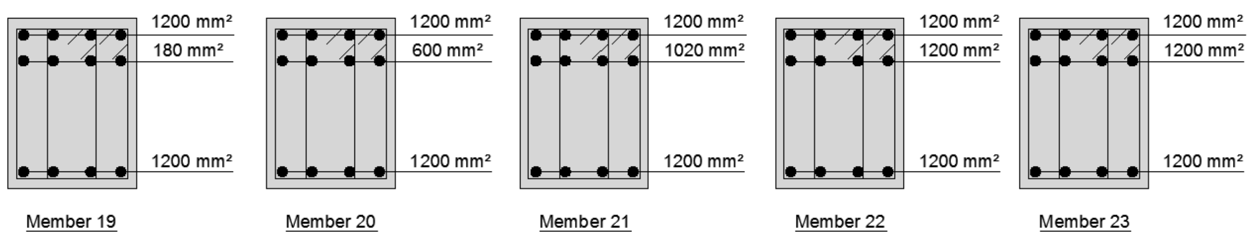


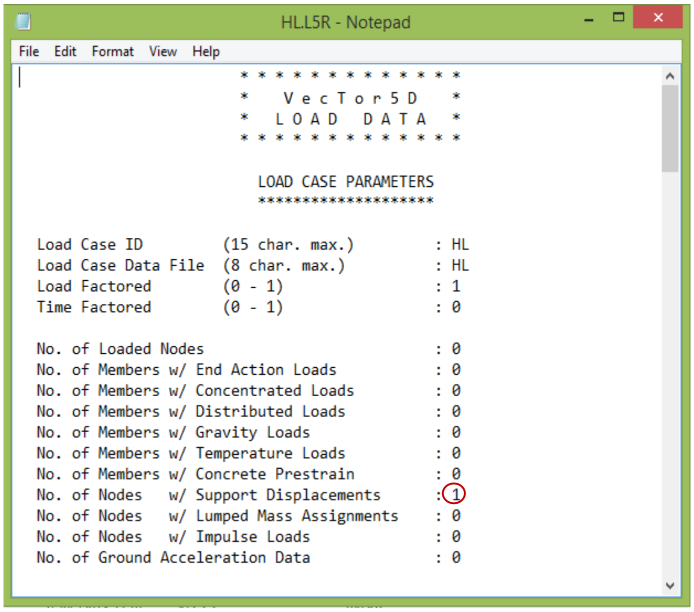
Figure 13 - Section C-C: Rebar Development

5. LOAD FILE

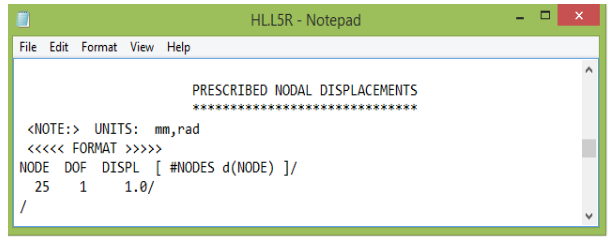
The *load data file* (.L5R) contains information about nodal loads, externally applied member end actions, concentrated loads, uniformly distributed loads, gravity loads, temperature loads, concrete prestrains, prescribed nodal displacements, additional lumped masses, impulse, blast and impact forces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

In this example, two load cases are defined: the first one for the constant gravity loads applied vertically, and the second one for the monotonically increasing horizontal loads applied gradually, as shown in **Figure 1.**

Figures 14 and **15** present the load case parameters assigned in this example for each load case (horizontal and vertical). Note that these two load cases should be specified in two load data files; it's not correct to merge two load cases in one load data file.



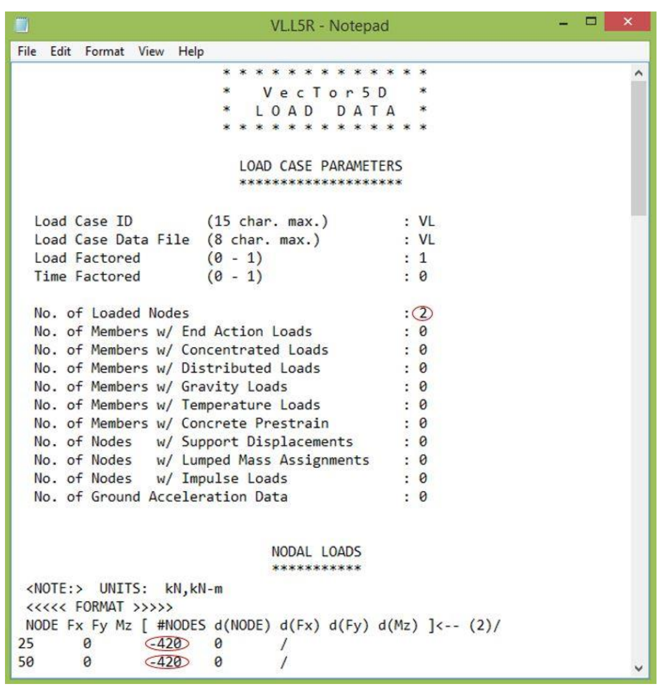
```
HLL5R - Notepad
File  Edit  Format  View  Help
                          * * * * * * * * * * * * *
                          *   V e c T o r 5 D     *
                          *   L O A D   D A T A   *
                          * * * * * * * * * * * * *

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

 Load Case ID         (15 char. max.)         : 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 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
```



```
HLL5R - Notepad
File  Edit  Format  View  Help

                    PRESCRIBED NODAL DISPLACEMENTS
                    ******************************
 <NOTE:>  UNITS:  mm,rad
 <<<<< FORMAT >>>>>
NODE  DOF  DISPL  [ #NODES d(NODE) ]/
  25    1     1.0/
/
```

Figure 14 - Specification of Load Parameters for Horizontal Load

It is possible to choose between two load application modes: a force-controlled, or a displacement-controlled mode. When using the force-controlled mode, it will be not possible to obtain the post-peak behaviour of the structure. Thus, whenever possible, perform the analysis in a displacement controlled mode. In this example, the displacement-controlled mode is used since only one load is applied in a monotonically increasing manner.

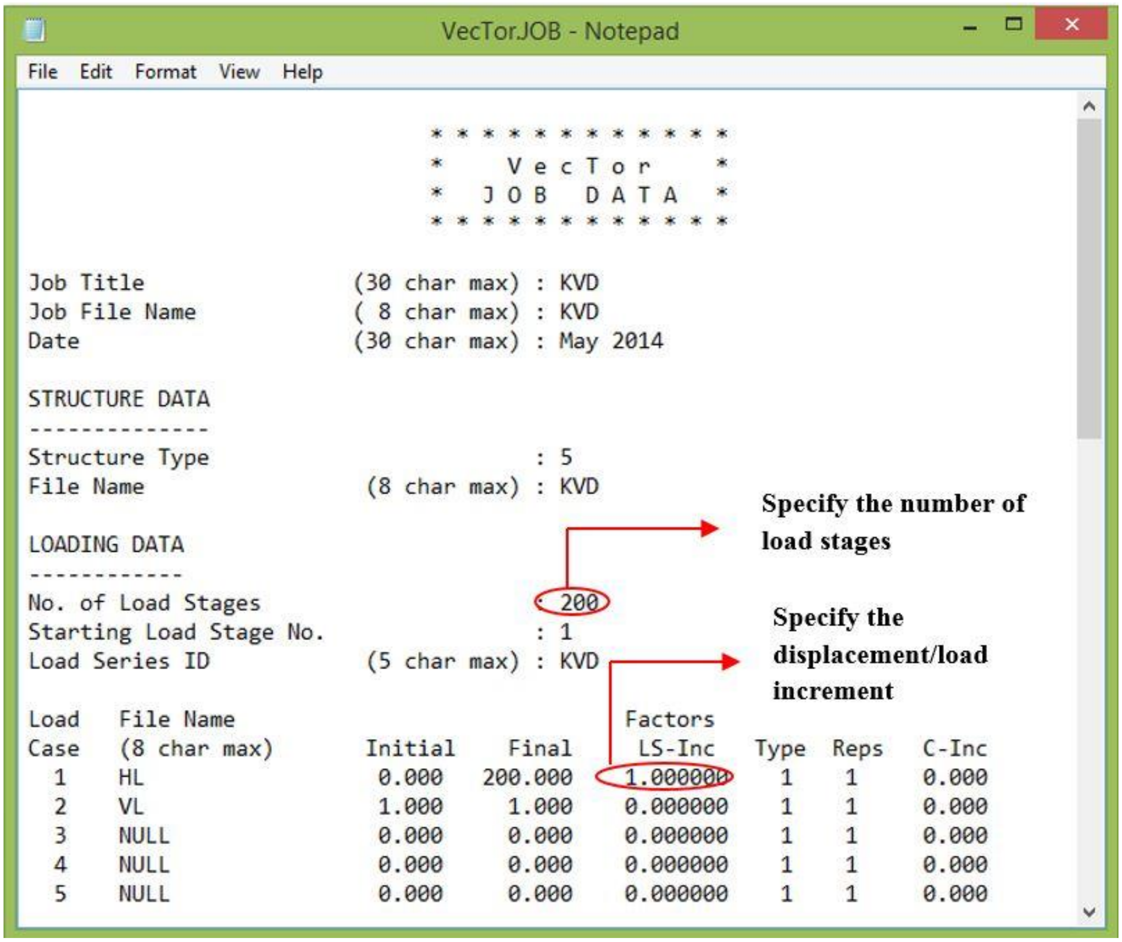


```
VLL5R - Notepad
File  Edit  Format  View  Help
                          * * * * * * * * * * * * *
                          *   V e c T o r 5 D     *
                          *   L O A D   D A T A   *
                          * * * * * * * * * * * * *

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

 Load Case ID         (15 char. max.)         : 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 Nodes   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 [ #NODES d(NODE) d(Fx) d(Fy) d(Mz) ]<-- (2)/
25      0     -420    0       /
50      0     -420    0       /
```

Figure 15 - Specification of Load Parameters for Vertical Load

6. 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 show the Job file for the monotonic analysis. For a cyclic analysis (not covered in this document), the horizontal load [which points in the positive x-direction] will increase gradually first, then decrease gradually; then increase gradually, but in the reverse direction. Finally, the force will decrease gradually. Therefore, it's quite obvious this is reversed-cyclic loading type, so loading type 3 should be input.


VecTor.JOB - Notepad

File Edit Format View Help

```
                         * * * * * * * * * * * *
                         *    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) : May 2014

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

LOADING DATA
------------
No. of Load Stages                     : 200
Starting Load Stage No.                : 1
Load Series ID            (5 char max) : KVD

Load   File Name                                 Factors
Case   (8 char max)     Initial    Final     LS-Inc     Type  Reps   C-Inc
  1    HL                 0.000   200.000    1.000000     1     1    0.000
  2    VL                 1.000     1.000    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
```


Figure 16 - Job Data File - Specifying Load Data

```
VecTor.JOB - Notepad
File  Edit  Format  View  Help
ANALYSIS PARAMETERS
-------------------
Analysis Mode                       (1,3,4) : 1
Seed File Name                 (8 char max) : NULL
Convergence Limit                    (>1.0) : 1.00001
Averaging Factor                     (<1.0) : 0.
Maximum Iterations            (default 100) : 100
Convergence Criteria                  (1-3) : 2
Results Files                         (1-4) : 2
Output Format                           (1) : 1

MATERIAL BEHAVIOUR MODELS
-------------------------
Concrete Compression Base Curve       (0-4) : 1
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
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
```

Figure 17 - Job Data File - Specifying Analysis Parameters and Material Behaviour Models

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 software ignores it.

$$Ec = \frac{f'c}{e0} \qquad (Eqn. 3)$$

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 software ignores it.

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

$$Ec = \frac{f'c}{e0} x \, \frac{n}{n-1} \qquad (Eqn. 5)$$

4 - Hoshikuma (HSC): This model is also used for high concrete strengths (≥50MPa). 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 x 10^{-6} 1/ºC; Kc = 4320 mm^2/h; Agg = 10 mm; and Dens = 2400 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.

7. 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 18** shows which values are input.

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,

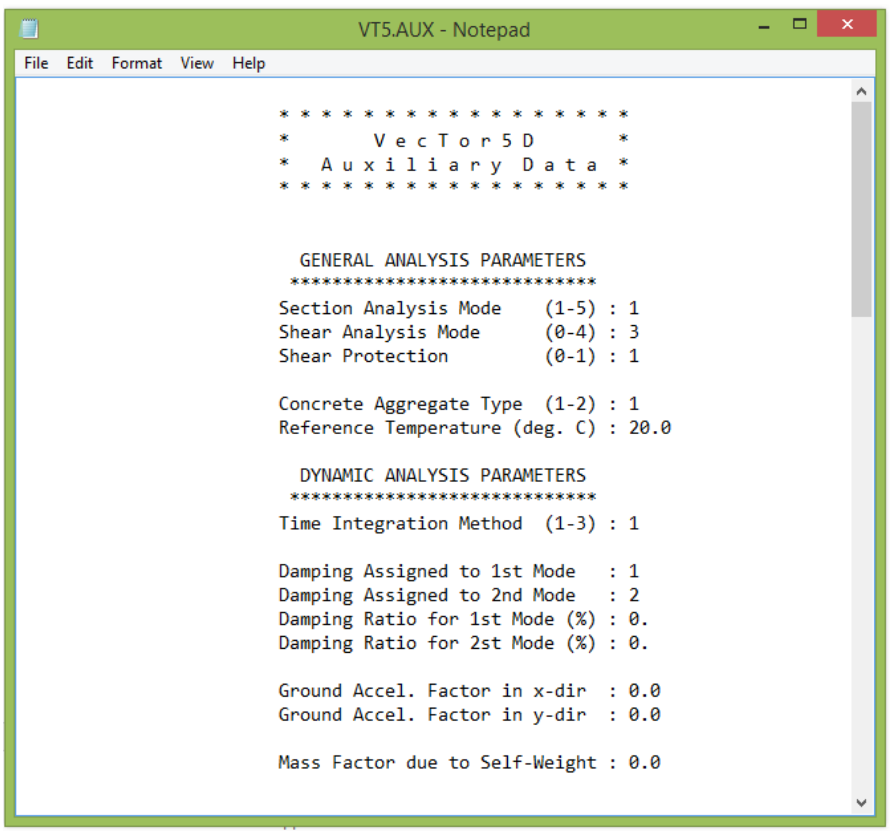
1 - Uniform Shear Flow (Multi-Layer),

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.




```
* * * * * * * * * * * * * * * * *
*       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

Damping Assigned to 1st Mode    : 1
Damping Assigned to 2nd Mode    : 2
Damping Ratio for 1st Mode (%) : 0.
Damping Ratio for 2st 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 18 - Auxiliary Data File - Input Field

8. DETERMINATION OF FAILURE MODE

First we run the program VT5.exe, once terminated, it will show what kind of failure mode happened. **Figure 19** shows that at member 58, shear failure happened.

```
                          99                   1.001684
                         100                   1.001685

 SHEAR FAILURE:    Member:  58

 STORING RESULTS IN ASCII FILE   : Duong_26.A5E

 CONV(1):             12641
 CONV(2):             18748
 CONV(3):           4229008

    Input Time:               1300
    Calc. Time:               2400
    Stiff Time:                500
   Decomp Time:              31000
   Invert Time:              71440
     Moca Time:             923130
 Load Stg Time:              16500
  Storage Time:              53710
 Analysis Time:            1107780

*PROGRAM EXECUTION IS COMPLETED*

Fortran Pause - Enter command<CR> or <CR> to continue.
```

Figure 19 – Running Results of VT5.exe

We still need to go to the output data file and check the detailed member output. Curvature, axial strain and shear strain results in **Figure 20** are asterisks, which is due to non-convergence of analyzing process. This indicates failure occured.

```
             MEMBER SECTION STRESSES AND STRAINS
             ***********************************
                       AVERAGE VALUES

                        MEMBER :  58

      MOMENT:        0.0 kN-m        CURVATURE:********* me/m
  AXIAL LOAD:        0.0 kN       AXIAL STRAIN: ******** me
       SHEAR:        0.0 kN       SHEAR STRAIN: ******** me
```

Figure 20 – Specification of non-convergence analyzing process

Moreover, by inspection of the crack conditions in **Figure 21**, we can see that cracks near neutral axis (shear crack) is obviously larger than cracks in two sides. So we can finally conclude that the failure mode of this structure is shear failure, which occurred at member 58.

CRACK CONDITIONS

NC	WCR (mm)	SCR (mm)	VCI (MPa)	SLIP (mm)	STATE
1	1.03	181.7	0.00	0.00	1
2	0.95	166.2	0.00	0.00	1
3	0.92	155.2	0.10	0.07	1
4	0.91	145.6	0.21	0.10	1
5	0.93	137.8	0.27	0.11	1
6	0.96	131.2	0.31	0.12	1
7	1.04	121.7	0.37	0.14	1
8	1.30	120.2	0.41	0.22	1
9	1.62	123.0	0.42	0.45	1
10	2.51	151.3	0.43	0.50	4
11	3.68	181.8	0.40	0.42	4
12	4.42	207.6	0.07	0.14	4
13	4.28	224.5	0.02	0.06	4
14	4.46	242.4	0.01	0.04	4
15	4.65	258.6	0.08	0.14	4
16	4.83	274.4	0.09	0.15	4
17	4.83	274.9	0.09	0.14	4
18	4.68	260.5	0.08	0.14	4
19	4.48	245.5	0.08	0.14	4
20	4.19	229.6	0.06	0.13	4
21	3.87	213.6	0.02	0.07	4
22	4.26	208.8	0.06	0.13	4
23	4.09	200.2	0.08	0.15	4
24	3.94	198.4	-0.00	-0.00	4
25	3.64	206.3	-0.00	-0.03	4
26	3.51	225.9	-0.01	-0.04	4
27	3.49	238.7	0.26	0.30	4
28	3.37	250.7	0.33	0.36	4
29	3.22	263.4	0.43	0.42	4
30	3.02	278.7	0.47	0.46	4
31	2.66	301.2	0.34	0.39	4
32	2.82	271.7	1.58	1.38	4

Figure 21 – Crack Conditions

We can look at output files one by one and find at which load stage shear failure first occur. Then, find the failure load and failure displacement corresponding to this load stage.

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 Chu Peng an Kyle Blosser, as parts of projects supervised by Dr. Serhan Guner.