USER BULLETIN 4

BEAM MODELING WITH VECTOR5
USING PRE- AND POST-PROCESSING FACILITIES

1. INTRODUCTION

FormWorks Plus is a pre-processor program, which creates the input text files for a VecTor5 analysis. The FormWorks Plus allows the modeler to visualize the structures which reduces the possibility for input mistakes and makes modeling more fun. In this manual, the modeling of a simply-supported beam VSA1 is discussed. When analyzing the beam, a nodal displacement is applied to the midspan of the beam, and half of the beam is modeled. This displacement-controlled analysis provides the post-peak response, which cannot be achieved with a load-controlled analysis.

This document will allow a user to gain hands-on experience with the FormWorks Plus program. Developing an understanding of this program will also allow the user to better understand the VecTor5 processor and the files it produces. The input supplied into the FormWorks Plus program directly correlates to the input required by the VecTor5 processor.

When the beam is being created, the FormWorks Plus program will create the input files required for a VecTor5 analysis: i.e., Structure Data File (.S5R), Load Data Files (.L5R), Job Data File (.JOB), and the Auxiliary Data File (.AUX). These files will be created by the FWP program. Ensure to save these files often using the save icons at the center of the toolbar located at the top of the screen. These files should be saved into the same folder.

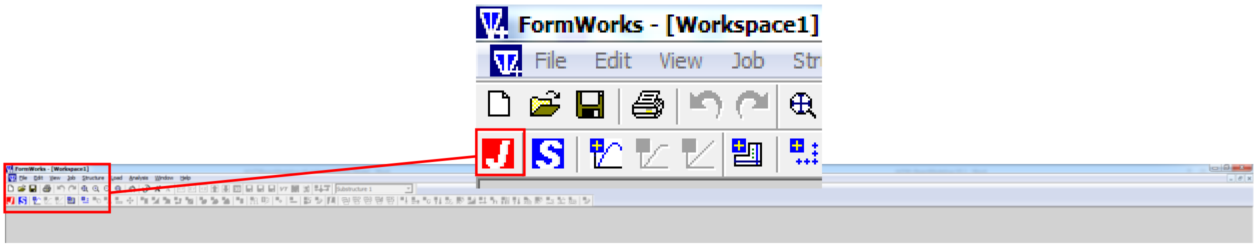
This Manual will also review the use of the Janus program, which reads the VecTor5 output files and presents them graphically. In order to get a better understanding of VecTor5, it is recommended reading the User's Manual of VecTor5, which can be downloaded from Dr. Guner's website (see User's Manuals tab).

This document is prepared by Kyle Blosser as a part of a project supervised by Dr. Serhan Guner. No software development work was undertaken during this project.

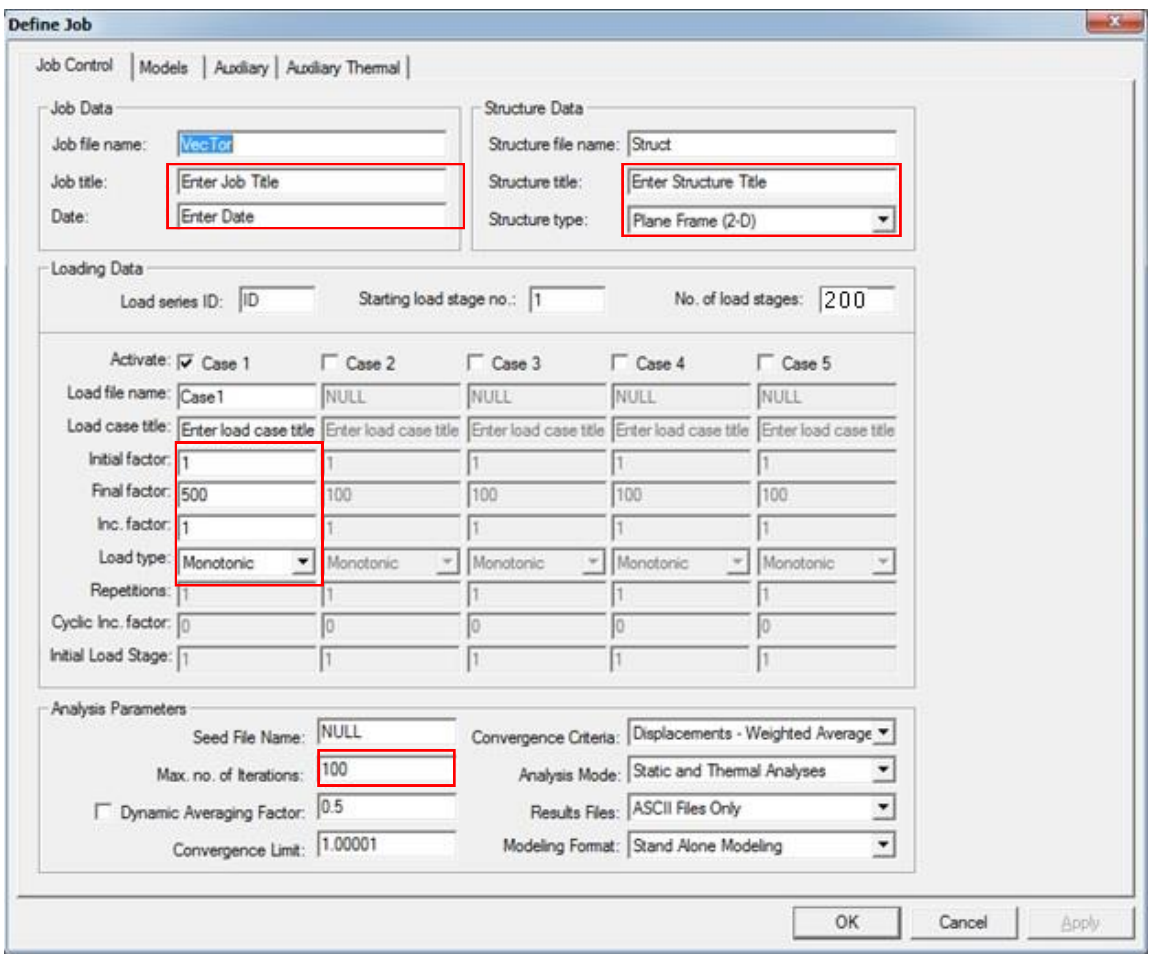
2. CREATING THE NECESSARY INPUT FILES USING THE PRE-PROCESSOR

2.1 Define Job File

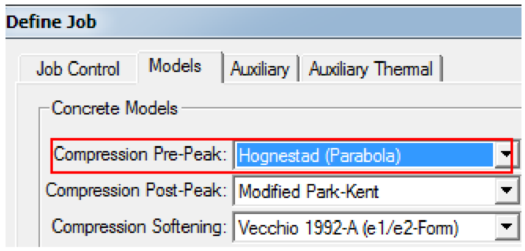
First of all, save the Workspace file into a new folder named, for example, Analysis. You may rename the Workspace files as, for example, Beam. To begin creating the model with FormWorks Plus, start with the Define Job icon located at the top-left side of the screen.



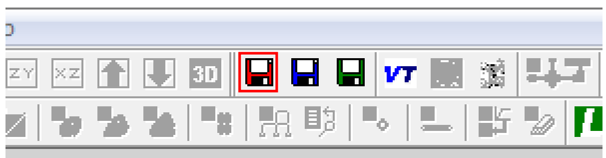
Once this icon is selected, the following dialog window will appear. In this box, the fields indicated with a red box should be input. The Job File name can remain the same but the Job Title and Date should be changed for the project. On the top-left, the Structure Data must be entered for the analysis. The Load File must be activated in this dialog, which is Case 1 for this example. The Load Case title, Initial and Final Load factors, Increment factor, and Load type must be defined. For this example, a "Monotonic" loading type is selected. The maximum number of iterations should be defined as 100. The number of load stages is not very important, but it must be large enough so that the beam can fail. In summary, the Job file specifies the displacement increments for this analysis. The other fields in the Analysis Parameters section may remain as shown below.



When modeling high-strength concrete (not applicable in this example), the user should choose an appropriate compression pre-peak model (not Hognestad) within the Models tab of the Define Job menu, as shown below. Also, input the appropriate *Ec* and *e0* values in the Define Material Properties menu. Refer to the VecTor5 User's Manual, pages 12 and 13. After the analysis is complete, review the ".S5E" expanded data file to ensure the *Ec* and *e0* values used by the program are the same as were originally intended.

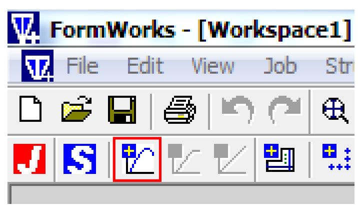


When the Job file has been defined, click "OK" to exit the dialog box. Promptly save the Job file by clicking on the first disk icon, which appears in red.

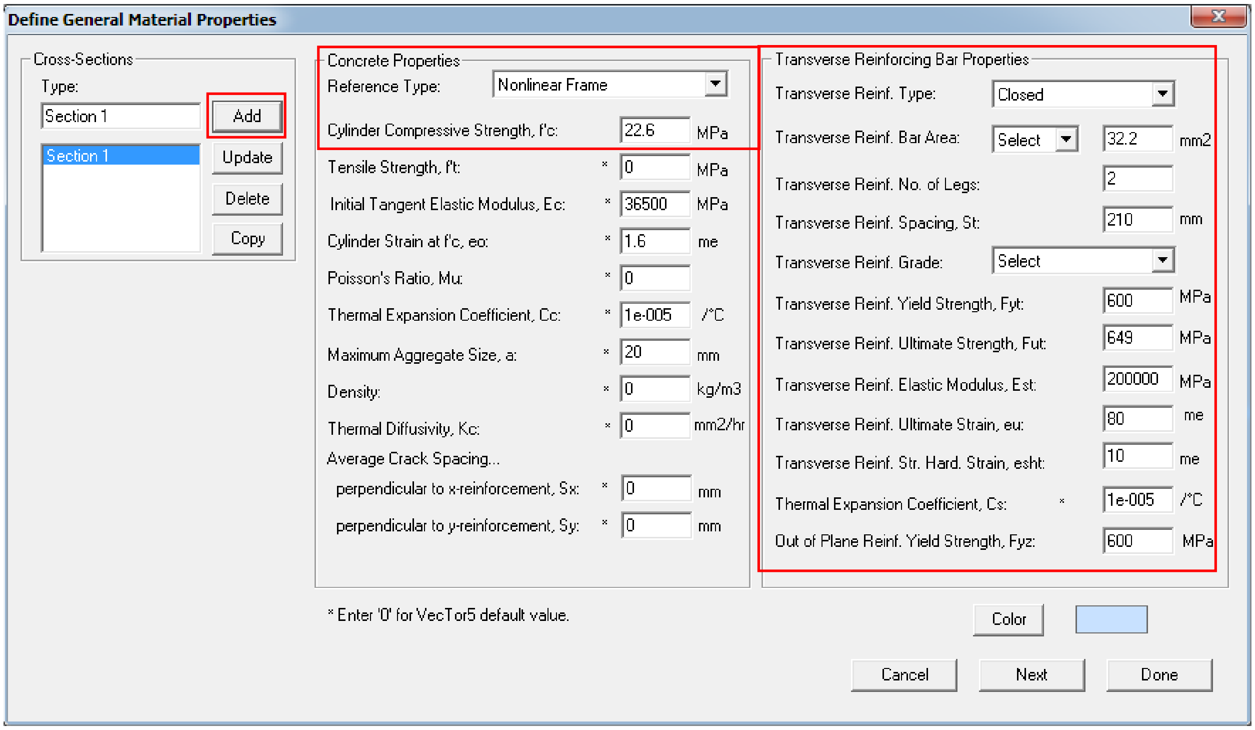


2.2 Define Cross Sections

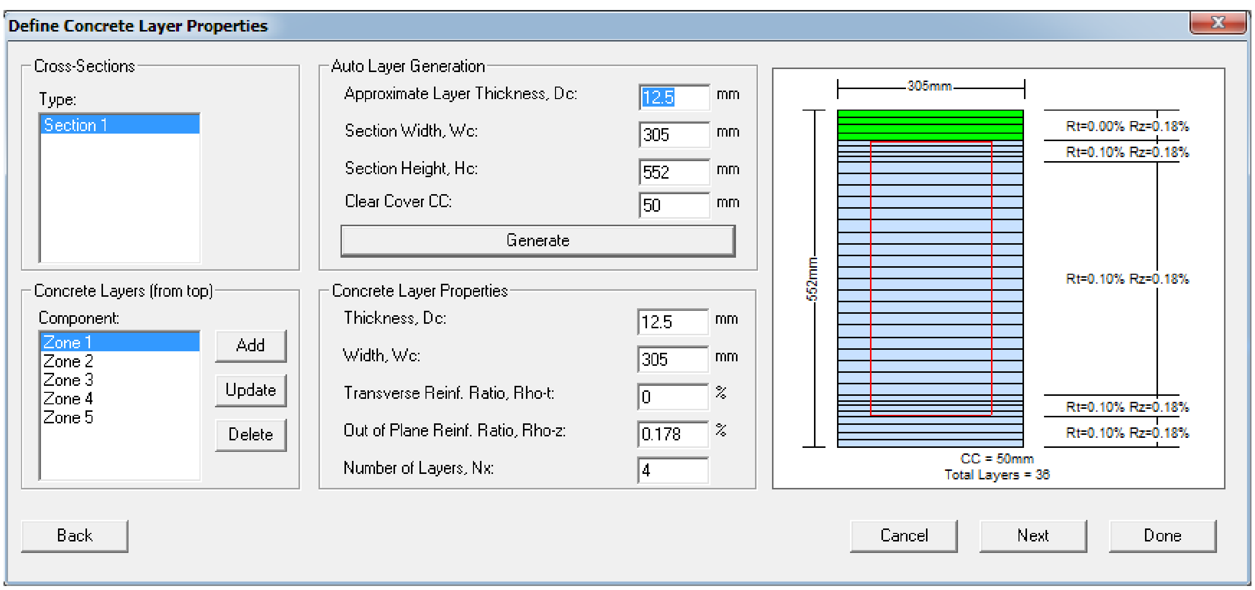
Once the Job file is saved, the next step is to create the Structure Data file. This is done by clicking on the Define Materials icon located to the right of the Job File icon.



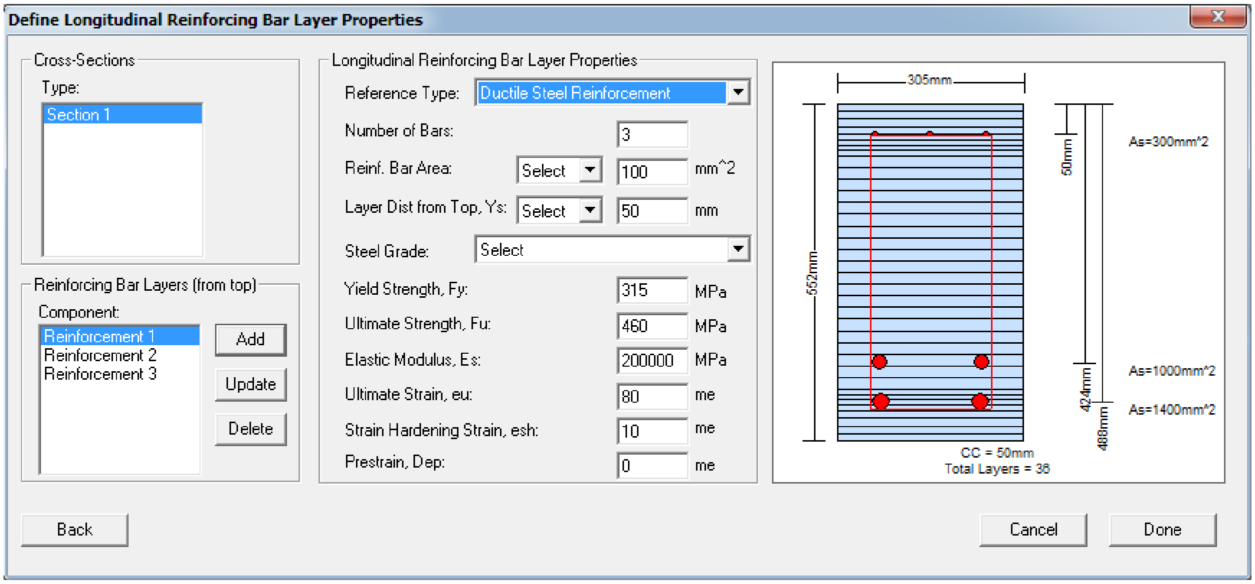
Upon clicking this icon, the "Define General Material Properties" dialog box appears, shown below. The values shown in the fields below indicate the properties that are characteristic to the beam VSA1 from the Toronto beams experiment. All of the fields boxed in red must be input into the program. For users in the United States, the English unit for compressive strength of 22.6 MPa is approximately equal to 3.28 ksi. Transverse reinforcement is synonymous to stirrups and shear reinforcement. The longitudinal reinforcement will be defined in following menus. Once the properties have been input, be sure to click "Add" before clicking "Next", or else all values that were input will be lost. If changed need to be made, click "Update".



After clicking "Next", the dialog box now allows the user to input the concrete layer properties. In order to calculate the percentages of transverse and out-of-plane reinforcement, refer to the "Bulletin 1: Beam Modeling with VecTor5" document for a quantitative example. FWP will automatically generate these layers, and no calculations are needed. Note that it is important to check these values for non-rectangular cross sections.



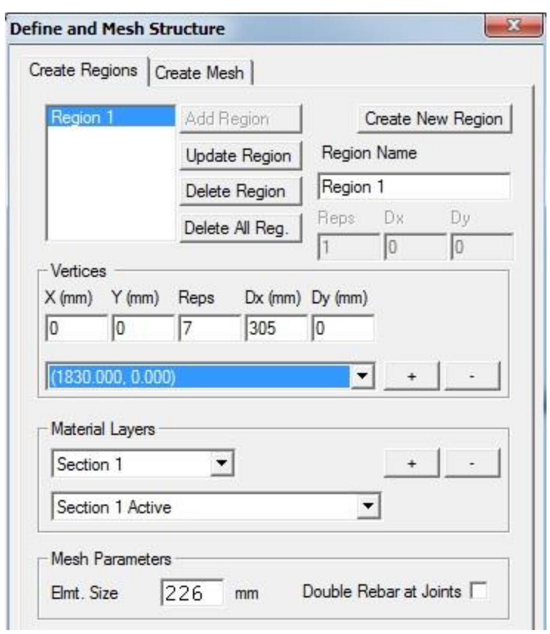
In the next dialog box, the Longitudinal Reinforcing Bar Layer Properties should be input. Each layer corresponds to a row of bars at a specified depth, Y_s. For beam VSA1, the first rebar layer is 50 mm from the top of the beam. The distances can also be input via the dropdown menu by selecting Top and Bottom. The steel grade can also be selected if the specific steel properties are unknown. Each M10 bar has a sectional area of 100 mm^2. The values for Yield Strength, Ultimate Strength, and others must be input. The value for Prestrain is required for prestressed elements only and should be left as zero for this example. For this beam, there are three rebar layers which must be defined. Once all of the layers have been added, click "Done" to exit the dialog box.



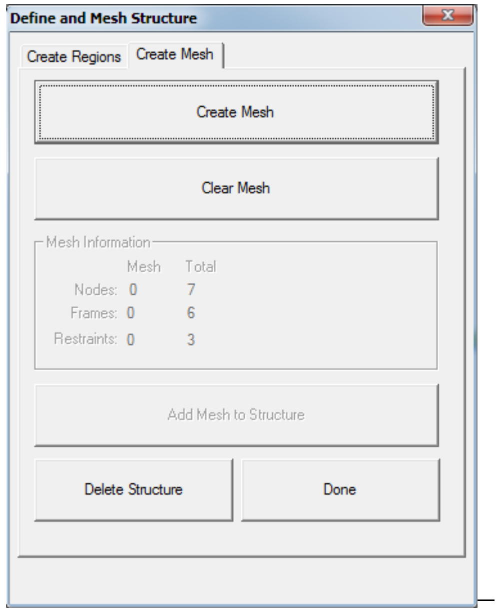
Once the dialog box closes, be sure to save the Structure Data File by clicking the second, green, save icon. Note that this does not mean the Structure Data File is complete, only that it is in progress and must be saved to avoid loss of work.

2.3 Mesh the Structure

Once the Material Properties have been defined, the next step is to create the global beam model. To do this, click on the "Define and Mesh Structure" icon. In the Vertices section, input the first coordinate (0, 0) by clicking the plus symbol; then input the end coordinate (1830, 0). Inputting the number of Reps and *Dx* is not necessary, as FormWorks Plus will perform the meshing automatically. The Material Layers should be assigned as Section 1, based on the section which was previously defined. The Element Size is automatically calculated as approximately half of the cross-section depth. There are no joints for this structure, so the Double Rebar box can remain unchecked. Once the region is added, it is time to move to the "Create Mesh" tab.



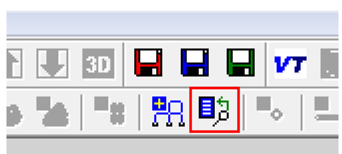
In the “Create Mesh” tab, click on the “Create Mesh” button, followed by “Add Mesh to Structure”. It can be noted that the amount of nodes is 7, with a corresponding number of 6 members, which makes sense for the beam being defined. Then, click “Done” to exit the window.

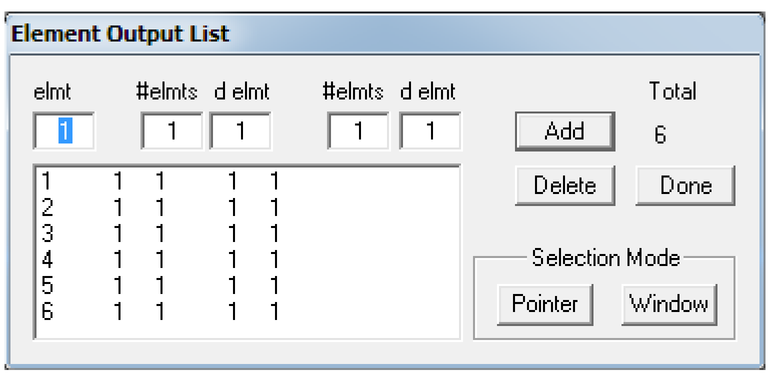


Once the framework for the structure has been defined and meshed to the beam, be sure to again save the Job Data file and Structure Data file by clicking on the red and blue save icons.

2.4 Create Element Output List

Once the elements have been added to the structure, it is necessary to create the output list. Click on the icon shown below to generate the proceeding dialog box.

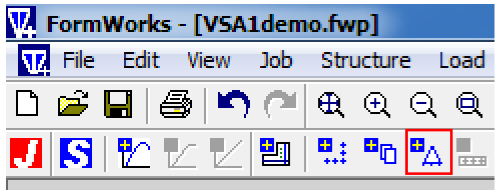


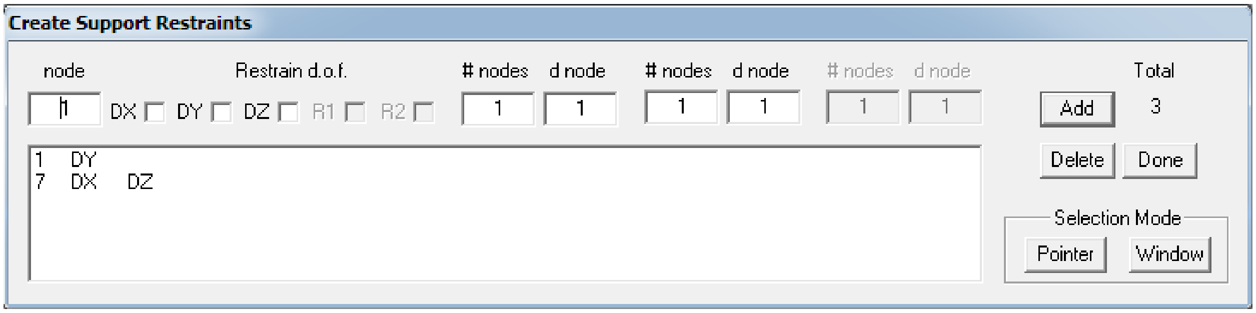


Select window option and select all elements using your cursor; then click on add. Once this is done, be sure to save the Structure Data file using the blue save icon.

2.5 Define Support Restraints

Now that the beam has been created, create the supports. Click on the icon shown below to generate the dialog box shown.

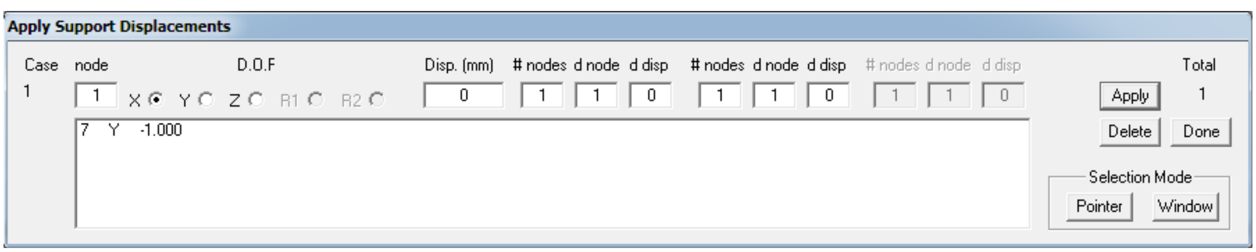




In this menu, type the node number or select the node to which the support will be assigned. Then, select the desired degrees-of-freedom to be restrained. Since half of the beam is being modeled due to symmetry, the X and Z degrees of freedom must be restrained at node 7. Click “Add” to generate a list in the window below. Continue to do this for each node which needs to be restrained. Click “Done” when finished.

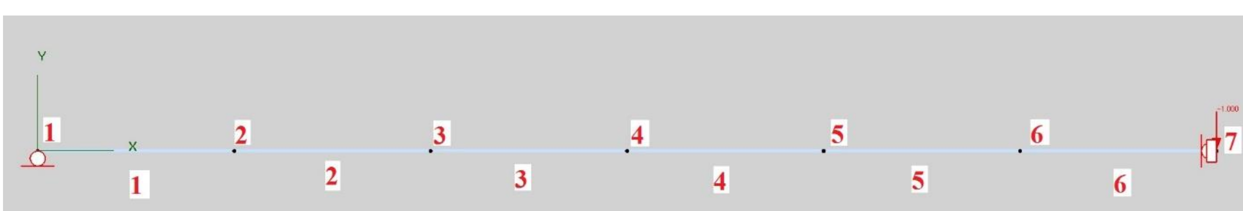
2.6 Apply Support Displacements

For the beam VSA1, a nodal displacement will be required to conduct a displacement-controlled analysis. The displacement is 1.0 mm and is applied to the midspan. To do this, click on the icon shown below to generate the dialog box shown.



Type the node number or select the node for which the displacement is to be defined. For the beam VSA1, node 7 is selected, with a displacement of 1.0 in the negative y-direction. Therefore, a displacement value of -1.0 is entered. Once the displacement is specified, click “Apply” to add it to the list. Click “Done”.

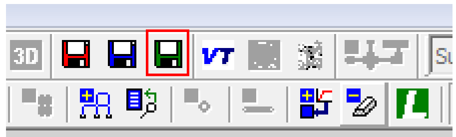
Once the displacements have been applied to the beam, the model should look like this:



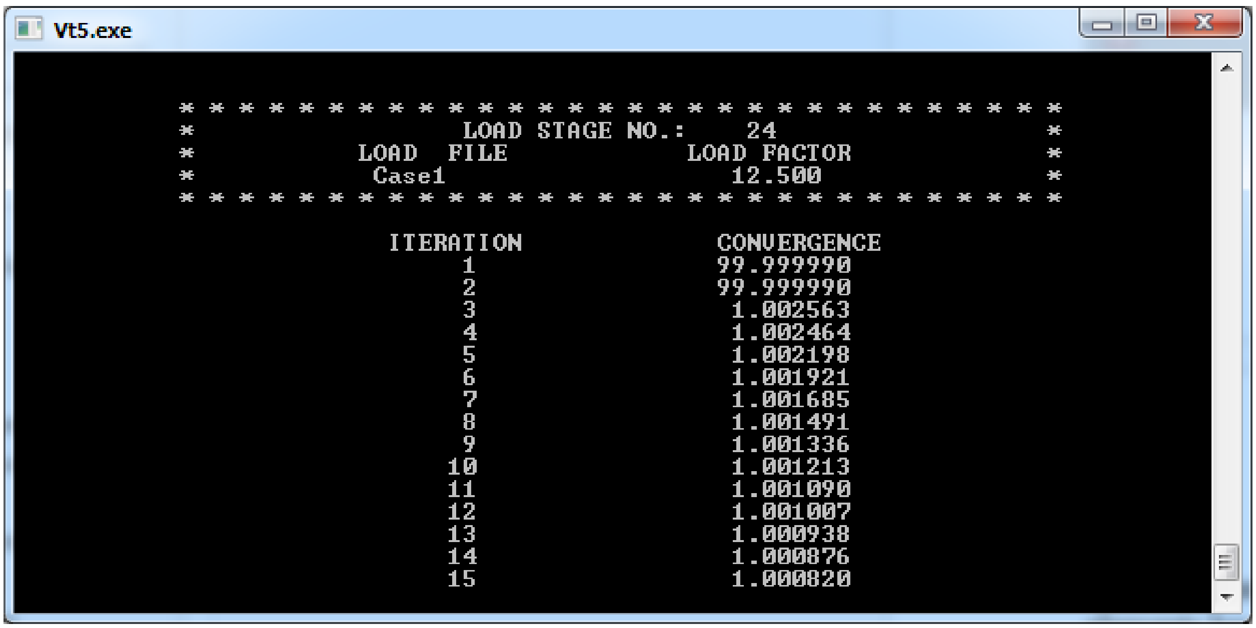
It can be seen that the supports are in place, the nodes are numbered correctly, the member elements have been defined, and the nodal displacement has been added to node 7. Save the Load file by clicking on the green save icon.

3. PERFORMING THE ANALYSIS

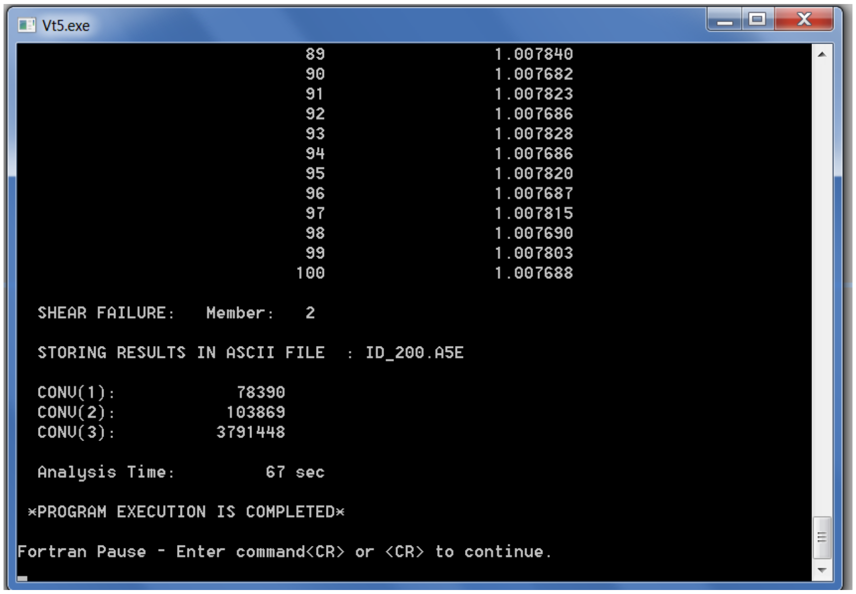
Now that all analysis files have been created, click on the icon shown below to begin the VT5.exe program which will analyze the beam. Alternatively, you can click on the VT5.exe file inside the Analysis folder that you created.



Choosing this icon will activate the program and the VT5.exe will begin to run, shown below.

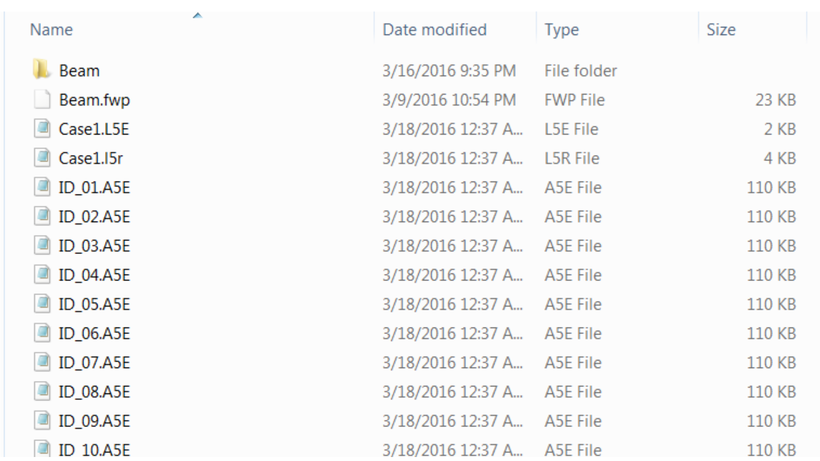


The analysis is running and the calculations are performed very quickly. With each iteration, the convergence becomes closer to a value of 1. This screenshot illustrates the point at which the program is at Load Stage 24. In the Job file, it has been specified that there will be a total of 200 Load Stages. This will ensure that the beam will fail before the analysis reaches the specified load stage limit. Once the beam fails, the program will show the failure mode, as well as which member has failed. This can be seen in the following screenshot below. You may then close the analysis without a need to wait for the remaining load stages.



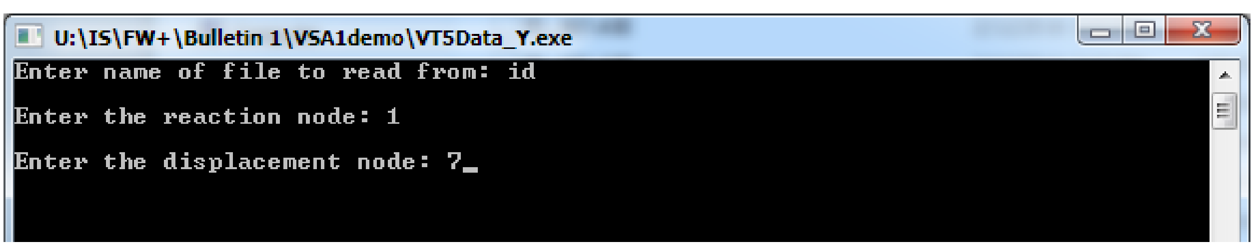
4. PLOTING THE LOAD-DEFLECION CURVE

To access the data files that VecTor5 has created for the beam analysis, go to the "Analysis" folder where you have saved the files. In this location, the following files will appear. You will see the expanded structure and load files, and output files created by VT5.exe. You can open any of these files using Notepad. You should review the expanded date file (*.S5E) to check for input mistakes, especially with the material properties.



Name	Date modified	Type	Size
Beam	3/16/2016 9:35 PM	File folder	
Beam.fwp	3/9/2016 10:54 PM	FWP File	23 KB
Case1.L5E	3/18/2016 12:37 A...	L5E File	2 KB
Case1.l5r	3/18/2016 12:37 A...	L5R File	4 KB
ID_01.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_02.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_03.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_04.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_05.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_06.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_07.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_08.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_09.A5E	3/18/2016 12:37 A...	A5E File	110 KB
ID_10.A5E	3/18/2016 12:37 A...	A5E File	110 KB

To obtain the load-deflection data, copy the "VT5Data_Y.exe" program into the analysis folder. Once the data program is copied, open the file to generate the following window. At this window, enter the information as the program prompts.

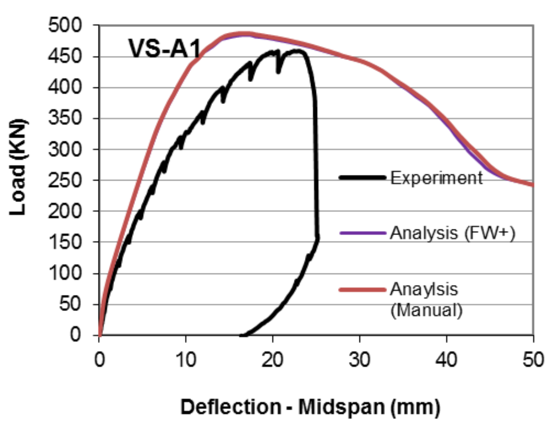


Once the information is entered, click enter to allow the program to read the output data files one by one. Once completed, the file named "results.dat" will be generated. Open this file to view a tabulated list of the load versus deflection data for the beam, respectively. Copy and paste this information into Excel or a graphing program so to create graphs to display the data. Remember to multiply the loading by a factor of 2 for this example, since the beam is being cut in half at the midspan and the load is at the midspan. The Results data file will look like this:

```
File  Edit  Format  View  Help
Extracted Data for file id

Disp at node 7 (mm), Reaction at node 1 (KN)
===================================
1.000000         43.540001
1.500000         56.009998
2.000000         67.589996
2.500000         78.849998
3.000000         89.889999
3.500000         100.809998
4.000000         111.559998
4.500000         122.180000
5.000000         132.639999
5.500000         142.910004
6.000000         152.919998
6.500000         162.610001
7.000000         171.229996
7.500000         179.029999
8.000000         186.440002
8.500000         192.860001
9.000000         199.300003
9.500000         205.600006
```

For this beam, the following graph is created for the Load versus Deflection information. The load is at node 1, the deflection is at node 7. As mentioned, the experimental data for this beam, as well as other beams, is provided in Vecchio and Shim (2004). VT5 analysis (Manual) refers to the analysis whose input data files were created manually without using the FormWorks Plus program (not conducted in this document).



5. POST-PROCESSOR JANUS

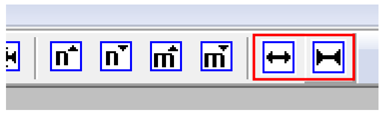
Janus is the post-processor program for VecTor5 which reads the output files created by VecTor5 and displays them graphically. In this program, various display modes can be chosen, such as rebar stresses, cross-section details, or crack pattern of the beam. You can also easily determine the failure mode using Janus. Please refer to the User Manual for Janus in Dr. Guner's website (see the User's Manual tab) for more details.

5.1 Opening the Program

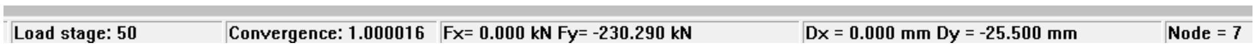
To open the Janus program, activate the "Janus.exe" program. On the left-hand side, click the folder button to open the Job file for the structure that you wish to view (i.e., beam VSA1)

5.2 The User Interface

When using the Janus program, becoming familiar with the user interface is important so that all information is correctly displayed. If the window does not fit the computer monitor screen, use the arrow keys within the menu bar to adjust the size of the window.



The information located at the bottom of the screen describes the Load Stage, the Convergence, the reaction forces, displacements, as well as the node number being examined.



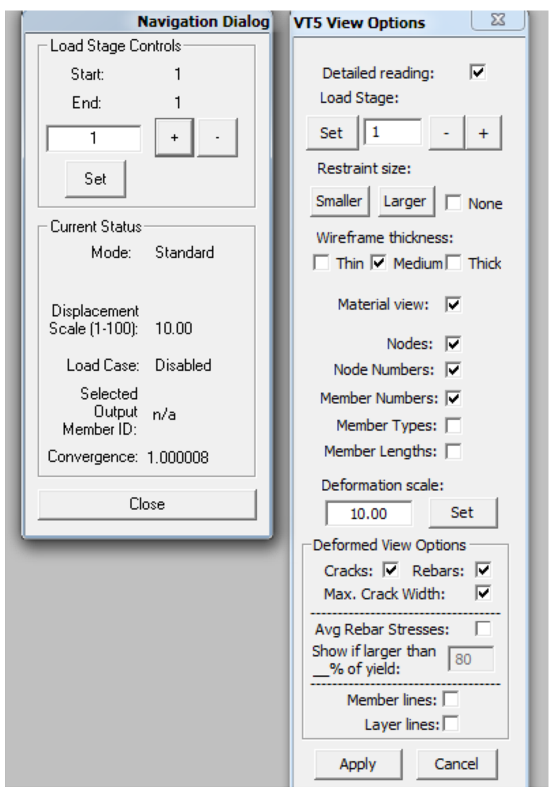
Toward the bottom of the screen, there is a Cartesian coordinate system. The top left buttons of the toolbar allow the user to change views of the beam between the 3D view of the entire beam, section view, and concrete cracking view of each member.

5.3 Viewing Load Stages

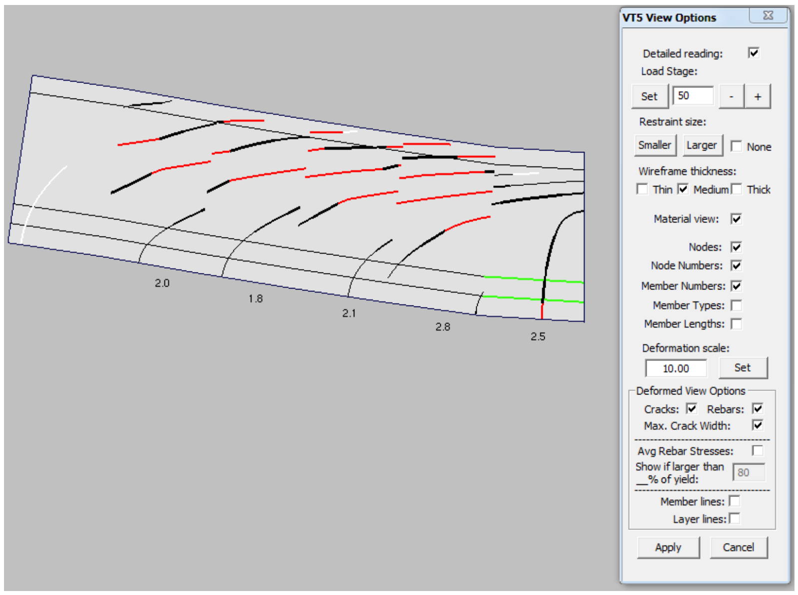
Navigation Dialog / VT5 View Options

Click on the icon below to access the Navigation Dialog and the VT5 View Options. This way, the user has control over which Load Stage is being viewed how the beam should be presented.

The following dialog box is the main interface for the Janus program. The viewing options can be changed and the load stage can be selected.

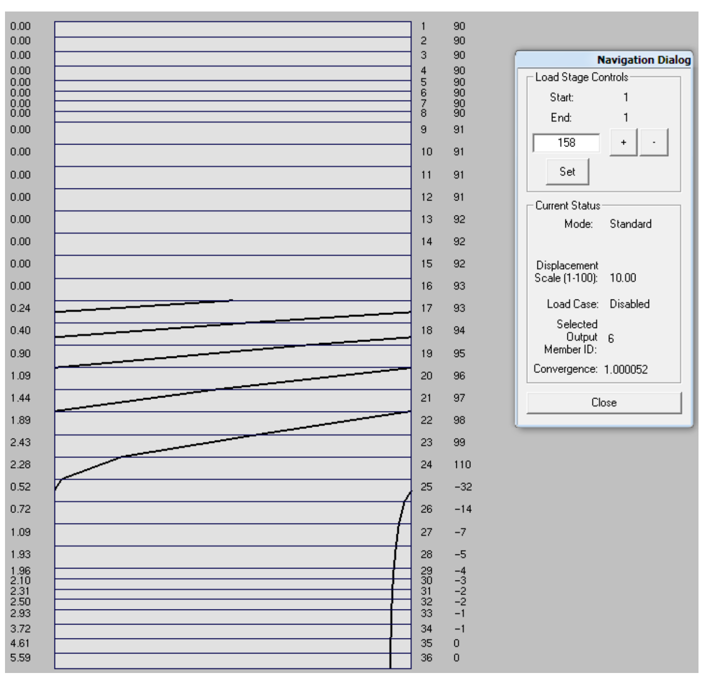


By clicking on the plus and minus symbols, the user can increase and decrease the load stages. Also, the user can input exactly which load stage they want by typing it in and then clicking "Set" next to the input field. Notice in the photo below that the beam has significantly deformed at load stage 50 and that cracking is quite apparent. The colors indicate the strain experienced in the reinforcement. Green color indicates that the reinforcement has yielded. The yellow color indicates that the reinforcement is in strain hardening. The red rebar color indicates that the reinforcement stresses are approaching the rupture strain. The dotted red line indicates that the reinforcement is ruptured. Crack widths are also represented with colors and different line weights to indicate the significance of cracking.



Section Crack View

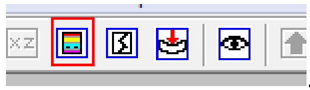
To view the beam as a section, click on the Crack View icon shown below and select the desired member in the dialog box that appears.

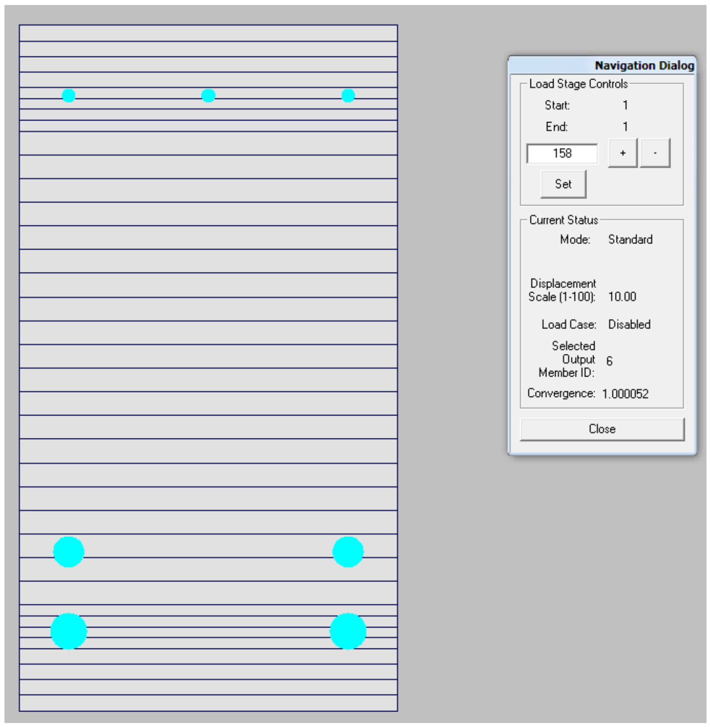


In this view, the user can see the concrete layers with the layer number on the right-hand side. Also, the crack widths are listed on the left-hand side and the cracks are visually noticeable. While in this view, it is possible to toggle between load stages to see the cracking-progression of the section at the selected member.

Section Layer View

Another useful view is the Layer View, which can be accessed by clicking the icon shown below. Clicking this icon will bring up the view shown on the next page.





This view allows the user to see the longitudinal reinforcement in the member, as well as a visual representation of the concrete layers present in the member that is selected.

6. CONCLUSION

Completing this analysis example will ensure familiarity with the analysis process with VecTor5. Understanding the process is critical in order to correctly model the structures and interpreting the analysis results. To avoid losing work, saving often is critical. Once the analysis is complete, a load-deflection or pushover curve can be created using Excel. A visual demonstration can be conducted with the Janus program. Understanding the modeling basics will allow for detailed studies of the non-linear behavior and failure conditions of concrete structures.

7. REFERENCES

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

Loya, A. S., Guner, S., and F. J. Vecchio (2015), "User's Manual of Janus for VecTor5," Online Publication, 15 pp. Download (see User's Manual tab).

Vecchio F. J. and Shim W. (2004), "Experimental and Analytical Re-examination of Classic Concrete Beam Tests, ASCE Journal of Structural Engineering, Vol. 130, No. 3, pp. 460-469. Download

---

This document is prepared by Kyle Blosser as a part of a project supervised by Dr. Serhan Guner. No software development work was undertaken during this project.